



# MSC Nastran 2022.1

## Quick Reference Guide

## Americas

5161 California Ave. Suite 200  
University Research Park  
Irvine, CA 92617  
Telephone: (714) 540-8900  
Email: [americas.contact@mscsoftware.com](mailto:americas.contact@mscsoftware.com)

## Europe, Middle East, Africa

Am Moosfeld 13  
81829 Munich, Germany  
Telephone: (49) 89 431 98 70  
Email: [europe@mscsoftware.com](mailto:europe@mscsoftware.com)

## Japan

KANDA SQUARE 16F  
2-2-1 Kanda Nishikicho, Chiyoda-ku  
1-Chome, Shinjuku-Ku  
Tokyo 101-0054, Japan  
Telephone: (81)(3) 6275 0870  
Email: [MSCJ.Market@mscsoftware.com](mailto:MSCJ.Market@mscsoftware.com)

## Asia-Pacific

100 Beach Road  
#16-05 Shaw Tower  
Singapore 189702  
Telephone: 65-6272-0082  
Email: [APAC.Contact@mscsoftware.com](mailto:APAC.Contact@mscsoftware.com)

## Worldwide Web

[www.mscsoftware.com](http://www.mscsoftware.com), [www.hexagon.com](http://www.hexagon.com)

## Support

<https://simcompanion.hexagon.com>

## Disclaimer

Hexagon reserves the right to make changes in specifications and other information contained in this document without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. Hexagon assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright © 2022 Hexagon AB and/or its subsidiaries. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of Hexagon is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from Hexagon suppliers. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at <http://www.mscsoftware.com/thirdpartysoftware> (or successor website designated by Hexagon from time to time).

PCGLSS 8.0, Copyright © 1992-2016, Computational Applications and System Integration Inc. All rights reserved. PCGLSS 8.0 is licensed from Computational Applications and System Integration Inc.

The Hexagon logo, Hexagon, MSC Software logo, MSC, Dytran, Marc, MSC Nastran, Patran, e-Xstream, Digimat, and Simulating Reality are trademarks or registered trademarks of Hexagon AB and/or its subsidiaries in the United States and/or other countries.

NASTRAN is a registered trademark of NASA. FLEXIm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. All other trademarks are the property of their respective owners.

Use, duplicate, or disclosure by the U.S. Government is subjected to restrictions as set forth in FAR 12.212 (Commercial Computer Software) and DFARS 227.7202 (Commercial Computer Software and Commercial Computer Software Documentation), as applicable.

U.S. Patent 9,361,413

March 16, 2022

NA:V2022.1:Z:Z:DC-QRG-PDF

# Documentation Feedback

At Hexagon Manufacturing Intelligence, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: [documentation-feedback@mscsoftware.com](mailto:documentation-feedback@mscsoftware.com).

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about Hexagon Manufacturing Intelligence documentation by taking a short 5-minute survey at: <http://msc-documentation.questionpro.com>.

**Note:**

The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact [Technical Support](#).



# C O N T E N T S

## MSC Nastran Quick Reference Guide

Table of Contents  
Preface

1

Executing MSC Nastran

2

NASTRAN Statement

- List of MSC Nastran Guides, xx
- Technical Support, xxi
- Training and Internet Resources, xxi

- Executing MSC Nastran, 2

- The NASTRAN Statement, 16

NASTRAN 17  
BUFFSIZE (1) 18  
F06 (2) 18  
NLINES (9) 18  
MAXLINES (14) 18  
METIME (20) 18  
APP (21) 18  
MACHTYPE (22) 18  
DIAGA (25) 18  
CONFIG (28) 18  
ADUMi (46 - 54) 18  
HEAT (56) 18  
DIAGB (61) 18  
PUNCH (64) 18  
MPYAD (66) 18  
DCMPOUT(69) 18  
DELFF (77) 19  
DBSET 19  
DMAP (82) 19  
F04 (86) 19  
RADMTX (87) 19  
RADLST (88) 19  
SMP (107) 19  
NEWHESS (108) 19  
(109) 19  
BUFFPOOL (114) 20  
(119) 20  
ATTDEL (124) 20  
NOKEEP (125) 20  
SPARSE (126) 20  
UPDTTIM (128) 20  
SMPYAD67 (129) 20  
MAXDBSET 21

AUTOASGN (133) 21  
TSTAMP (135) 21  
QUADINT (141) 21  
SCR300 (142) 21  
LOCBULK (143) 21  
(144) 21  
BFGS (145) 21  
FBGMEM (146) 22  
UWM (147) 22  
DBVERCHK (148) 22  
SCR300DEL (150) 22  
(151) 22  
DBLAMKD (155) 22  
(162) 23  
(166) 23  
LDQRKD (170) 24  
OLDQ4K (173) 24  
Q4TAPER (189) 24  
Q4SKEW (190) 24  
TETRAAR (191) 24  
SCRSAVE (196) 24  
MINFRONT (198) 24  
NSEGADD (200) 24  
CORDM (204) 24  
(205) 24  
DCMPSEQ (206) 24  
USPARSE (209) 25  
PUNCHTYPE (210) 25  
DISTORT (213) 26  
T3SKEW (218) 26  
(219) 26  
(220) 26  
(221) 26



(253 - 262) 26  
 MAXSET (263) 26  
 QUARTICDLM (270) 26  
 (273) 26  
 DBCFACT (274) 26  
 (275) 26  
 SPLINE\_METRICS (281) 26  
 MINDEF (303) 27  
 MPERTURB (304) 27  
 (309) 27  
 OLDRBE3 (310) 27  
 TBCMAG (311) 27  
 INDEX (316) 27  
 XMSG (319) 28  
 OLDDAREA (320) 28  
 RSEQCONT (357) 28  
 QLHOUL (359) 28  
 PRTPCOMP (361) 28  
 STRICTUAI (363) 28  
 STPFLG (366) 28  
 QRMETH (370) 29  
 PARAMCHK (372) 29  
 TZEROMAX (373) 29  
 NOLIN (386) 29  
 KRYLOV1 (387) 30  
 KRYLOV2 (388) 30  
 KRYLOV3 (389) 30  
 KRYLOV4 (390) 30  
 KRYLOV5 (391) 30  
 BARMASS (398) 30  
 DPBLKTOL (402) 30  
 OP2NEW (403) 31  
 DEF\_DENS (408) 31  
 DEF\_TECO (410) 31  
 DEF\_TEIJ (411) 31  
 DEF\_DAMP (412) 31  
 OPTCOD (413) 31  
 (414) 31  
 OLDTLDMTH (428) 31  
 NONLRGAP (431) 32  
 ESLNRO (443) 32  
 (444) 32  
 IFPSTAR 32

MNLQ4C (445) 32  
 (446) 33  
 PARCHILD(449) 33  
 (451) 33  
 DIFFS(614) 34  
 IFPBUFF(624) 34  
 NONUIV (653) 34  
 GPU\_MIN\_RANK (655) 35  
 GPU\_MIN\_FRONT (656) 35  
 (662) 35  
 SEGLOG(664) 35  
 CNTBKCOMP(666) 36  
 (670) 36  
 LMFBKCOMP(676) 36  
 FFOBKCOMP(677) 36  
 OL DLCNTR(678) 36  
 (679) 37  
 37  
 (684) 37  
 RDBOTH(695) 37  
 S2SDEF(701) 38  
 HDF5 (702) 38  
 OBEAMS (725) 38  
 OLDCCONE (726) 38  
 METS2S (727) 38  
 DMP (728) 39  
 N2SDOF (729) 39  
 H5NORDOF(730) 39  
 H5MTX (739) 39  
 H5MDL (740) 39  
 ACCSDLSZ(747) 39  
 H5GM34 (751) 39  
 UDEFGRID(753) 39  
 CNTBACK(758) 40  
 OP2IMPV(761) 40  
 IGNBLN(767) 40  
 BLNMEM(768) 40  
 OPENFSI(776) 41  
 H5INFO(789) 41  
 STRNCUR(778) 41  
 CNT101(786) 42  
 H5XHH (790) 42  
 DELTAU(800) 42

### 3

#### File Management Statements

- Key to Descriptions, 44
- The File Management Section (FMS), 45
  - File Management Statement Summary, 45
  - File Management Statement Descriptions, 46
  - File Management Statements, 50
    - \$, 50
    - ACQUIRE, 51
    - ASSIGN, 52
    - CONNECT, 61
    - DBCLEAN, 64
    - DBDICT, 65

- DBDIR, 79
- DBFIX, 80
- DBLOAD, 81
- DBLOCATE, 84
- DBSETDEL, 87
- DBUNLOAD, 88



## 4

### Executive Control Statements

- DBUPDATE, 90
- DEFINE, 91
- ENDJOB, 94
- EXPAND, 95
- INCLUDE, 97
- INIT, 98
- MEMLIST, 102
- PROJ, 103
- RESTART, 104

- Key to Descriptions, 108
  - Executive Control Section, 109
  - Executive Control Statement Summary, 109
- Executive Control Statement Descriptions, 110
  - \$, 111
  - ALTER, 112
  - APP, 115
  - CEND, 116
  - COMPILE, 117
  - COMPILER, 121
  - DIAG, 122
  - DOMAINSOLVER, 126
  - ECHO, 134
  - ENDALTER, 135
  - GEOMCHECK, 136

- ID, 140
- INCLUDE, 141
- LINK, 142
- MALTER, 144
- MODEL\_CHECK, 148
- SOL, 151
- SOL 600,ID, 154
- SOL 700,ID, 173
- SPARSE Solver, 185
- TIME, 190

## 5

### Case Control Commands

- Key to Descriptions, 192
- The Case Control Section, 193
  - Case Control Command Descriptions, 193
- Case Control Command Summary, 194
  - Subcase Definition, 194
  - Data Selection, 195
  - Output Selection, 200
  - Superelement Control, 204
  - Miscellaneous, 205
- Case Control Commands, 206

- \$, 206
- \$S700, 207
- A2GG (Case), 211
- ACCELERATION (Case), 212
- ACFPMRESULT (Case), 214
- ACPOWER (Case), 216
- ACTIVAT (Case), 218
- ACTRIM (Case), 219
- ADAMSMNF\* (Case), 220
- AECONFIG (Case), 232
- AERCONFIG (Case), 233
- AEROF (Case), 234
- AESYMX (Case), 235
- AESYMXZ (Case), 236
- AEUXREF (Case), 237
- ANALYSIS (Case), 238
- APRESSURE (Case), 246
- AUTOSPC (Case), 247
- AUXCASE (Case), 250
- AUXMODEL (Case), 251
- AVLEXB (Case), 252
- AXISYMMETRIC (Case), 258
- B2GG (Case), 259
- B2PP (Case), 260

- BC (Case), 261
- BCHANGE (Case), 262
- BCONCHK (Case), 263
- BCONTACT (Case), 264
- BCONTACT (Case), 268
- BCMOVE (Case), 271
- BEGIN BULK (Case), 272
- BENDL (Case), 275
- BOUTPUT (Case), 277
- BSQUEAL (Case), 279
- CAMPBELL (Case), 280
- CLOAD (Case), 281
- CMETHOD (Case), 282
- CMSENGY (Case), 283
- COSMSEL (Case), 286
- CSSCHD (Case), 287
- DBSAVE (Case), 288
- DEACTEL (Case), 289
- DEFORM (Case), 290
- DESGLB (Case), 291
- DESMOD (Case), 292
- DESOBJ (Case), 293
- DESSUB (Case), 294
- DESVAR (Case), 295
- DISPLACEMENT (Case), 296
- DIVERG (Case), 301
- DLOAD (Case), 302
- DRSPAN (Case), 303
- DSAPRT (Case), 304
- DSYM (Case), 306
- DYSTIFF (Case), 307
- ECHO (Case), 309
- EDE (Case), 311
- EKE (Case), 314
- ELAFORCES (Case), 317
- ELSDCON (Case), 319
- ELSENS (Case), 321
- ELSUM (Case), 324
- ENDSTEP (Case), 326
- ENDTIME (Case), 327
- ENDMODULE (Case), 328
- ENTHALPY (Case), 329
- EQUILIBRIUM (Case), 330
- ERP (Case), 332
- ESE (Case), 336
- EXPORTLD (Case), 340
- EXTDRIN (Case), 342
- EXTDROUT (Case), 351
- EXTSEQOUT (Case), 353
- FATIGUE (Case), 366
- FBODYLD (Case), 373
- FEMCHECK (Case), 374
- FLSFSEL (Case), 375
- FLSPOUT (Case), 377
- FLSTCNT (Case), 380
- FLUX (Case), 382
- FMETHOD (Case), 383
- FORCE (Case), 384
- FREQUENCY (Case), 387
- FRF (Case), 388
- GPFORCE (Case), 405
- GPKE (Case), 411
- GPRSORT (Case), 413
- GPSDCON (Case), 414
- GPSTRAIN (Case), 415
- GPSTRESS (Case), 417
- GROUNDCHECK (Case), 419
- GUST (Case), 421
- GVECTOR (Case), 422
- HADAPT (Case), 423
- HARMONICS (Case), 424
- HDOT (Case), 425
- HOUTPUT (Case), 426
- HTFLOW (Case), 427
- IC (Case), 428
- ICF (Case), 430
- IMPERFECT (Case), 432
- INCLUDE (Case), 433
- INTENSITY (Case), 434
- IRLoad (Case), 435
- K2GG (Case), 436
- K2PP (Case), 437
- K42GG (Case), 438
- LABEL (Case), 439
- LDLABEL (Case), 440
- LINE (Case), 441
- LOAD (Case), 442
- LOADNAME (Case), 443
- LOADSET (Case), 444
- M2GG (Case), 446
- M2PP (Case), 447
- MASSSET (Case), 448
- MASTER (Case), 449
- MAXLINES (Case), 451
- MAXMIN (Case), 452
- MAXMIN (Case), 454
- MAXMIN(DEF) (Case), 456
- MCFRACTION (Case), 459
- MCHSTAT (Case), 462
- MEFFMASS (Case), 463



- METHOD (Case), 465
- MFLUID (Case), 467
- MFREQUENCY (Case), 468
- MODALKE (Case), 472
- MODALSE (Case), 476
- MODES (Case), 480
- MODESELECT (Case), 481
- MODTRAK (Case), 487
- MONCARL (Case), 488
- MONITOR (Case), 489
- MPC (Case), 491
- MPCFORCES (Case), 492
- MPRES (Case), 495
- NLBUCK (Case), 496
- NLHARM (Case), 498
- NLIC (Case), 499
- NLLOAD (Case), 501
- NLOPRM (Case), 502
- NLPARM (Case), 506
- NLRESTART (Case), 507
- NLSTEP (Case), 510
- NLSTRESS (Case), 511
- NONLINEAR (Case), 513
- NOUTPUT (Case), 514
- NSM (Case), 515
- NVELOCITY (Case), 516
- OFREQUENCY (Case), 518
- OIMPERFECT (Case), 520
- OLOAD (Case), 521
- OMODES (Case), 524
- OTIME (Case), 526
- OUTPUT (Case), 528
- P2G (Case), 529
- PACCELERATION (Case), 530
- PAGE (Case), 532
- PARAM (Case), 533
- PARTN (Case), 534
- PEAKOUT (Case), 535
- PFGRID (Case), 536
- PFMODE (Case), 538
- PFPANEL (Case), 542
- PLOTID (Case), 546
- POST (Case), 547
- PRESSURE (Case), 551
- RANDOM (Case), 552
- RCROSS (Case), 553
- RELDISP (Case), 555
- REPCASE (Case), 557
- RESVEC (Case), 558
- RGYRO (Case), 560
- RIGID (Case), 561
- ROTBENT (Case), 563
- ROTSEKE (Case), 564
- RSDAMP (Case), 566
- SACCELERATION (Case), 567
- SDAMPING (Case), 568
- SDISPLACEMENT (Case), 570
- SEALL (Case), 571
- SEDAMP (Case), 572
- SEDR (Case), 573
- SEDV (Case), 574
- SEEXCLUDE (Case), 575
- SEFINAL (Case), 577
- SEKREDUCE (Case), 578
- SELGENERATE (Case), 579
- SELREDUCE (Case), 580
- SEMGENERATE (Case), 581
- SEMREDUCE (Case), 582
- SERESP (Case), 583
- SET (Case), 584
- SETP (Case), 586
- SETS DEFINITION (Case), 587
- SKIP (Case), 588
- SMETHOD (Case), 589
- SOLUTION (Case), 590
- SPC (Case), 591
- SPCFORCES (Case), 592
- SPLINOUT (Case), 595
- STATSUB (Case), 596
- STEP (Case), 598
- STOCHASTICS (Case), 599
- STRAIN (Case), 600
- STRESS (Case), 603
- STRFIELD (Case), 606
- SUBCASE (Case), 607
- SUBCOM (Case), 608
- SUBSEQ (Case), 610
- SUBSEQ1 (Case), 611
- SUBSTEP (Case), 612
- SUBTITLE (Case), 614
- SUPER (Case), 615
- SUPPORT1 (Case), 617
- SVECTOR (Case), 618
- SVELOCITY (Case), 619
- SYM (Case), 620
- SYMCOM (Case), 621
- SYMSEQ (Case), 622
- TACCELERATION (Case), 623
- TDISPLACEMENT (Case), 625
- TEMPERATURE (Case), 627

- TERMIN (Case), 631
- TFL (Case), 632
- THERMAL (Case), 633
- TITLE (Case), 635
- TIRE (Case), 636
- TRIM (Case), 637
- TRIMF (Case), 638
- TRIMGRP (Case), 640
- TSTEP (Case), 642
- TSTEPNL (Case), 643
- TSTRU (Case), 644
- TVELOCITY (Case), 645
- UNGLUE (Case), 647
- VCCT (Case), 648
- VECTOR (Case), 649
- VELOCITY (Case), 650
- VINTENSITY (Case), 652
- WEIGHTCHECK (Case), 655
- WETSENS (Case), 657

## ■ Case Control Applicability Tables, 658

## ■ X-Y PLOT Commands, 672

### ■ X-Y Output Command Summary, 672

- ALLEDGE TICS, 676
- BALL EDGE TICS, 677
- BLEFT TICS, 678
- BRIGHT TICS, 679
- CAMERA, 680
- CLEAR, 681
- CSCALE, 682
- CURVELINESYMBOL, 683
- DENSITY, 684
- LEFT TICS, 685
- LONG, 686
- LOWER TICS, 687
- PENSIZE, 688
- PLOTTER, 689
- RIGHT TICS, 690
- SEPLOT, 691
- SEUPPLOT, 692
- TALL EDGE TICS, 693
- TCURVE, 694
- TLEFT TICS, 695
- TRIGHT TICS, 696
- UPPER TICS, 697
- XAXIS, 698
- XBAXIS, 699
- XBGRID LINES, 700
- XDIVISIONS, 701
- XGRID LINES, 702
- XINTERCEPT, 703
- XLOG, 704
- XMAX, 705
- XMIN, 706
- XPAPER, 707
- XTAXIS, 708
- XTGRID LINES, 709
- XTITLE, 710
- XYPAPLOT, 711
- XYPEAK, 712
- XYPLOT, 713
- XYPRINT, 719
- XYPUNCH, 720
- XVALUE PRINT SKIP, 721
- YAXIS, 722
- YBDIVISIONS, 723
- YBINTERCEPT, 724
- YBGRID LINES, 725
- YBLOG, 726
- YBMAX, 727
- YBMIN, 728
- YBTITLE, 729
- YBVALUE PRINT SKIP, 730
- YDIVISIONS, 731
- YINTERCEPT, 732
- YGRID LINES, 733
- YLOG, 734
- YMAX, 735
- YMIN, 736
- YPAPER, 737
- YTAXIS, 738
- YTDIVISIONS, 739
- YTGRID LINES, 740
- YTINTERCEPT, 741
- YTITLE, 742
- YTLOG, 743
- YTMAX, 744
- YTMIN, 745
- YTTITLE, 746
- YTVALUE PRINT SKIP, 747
- YVALUE PRINT SKIP, 748

## ■ OUTPUT(POST) Commands, 749

- SURFACE, 749
- OUTPUT(PLOT) Commands, 754
  - AXES, 756
  - CAMERA, 758
  - CONTOUR, 759
  - CSCALE, 761
  - DISTORTION, 762
  - FIND, 763
  - MAXIMUM DEFORM, 765
  - OCULAR SEPARATION, 766
  - ORIGIN, 767
  - ORTHOGRAPHIC, etc., 768
  - PAPER SIZE, 769
  - PEN, 770
  - PERSPECTIVE, 771
  - PLOT, 772
  - PLOTTER, 780
  - PROJECTION, 781
  - PTITLE, 782
  - SCALE, 783
  - SEPLOT, 784
  - SET, 785
  - SEUPLOT, 788
  - STEREOSCOPIC, 789
  - VANTAGE POINT, 790
  - VIEW, 791
- Parameter Descriptions, 794
  - ACEXTMTD, 794
  - ACEXTSET, 794
  - ACMPF, 794
  - ACOUT, 794
  - ACOWEAK, 795
  - ACSYM, 795
  - ACTDMP, 795
  - ACTSMP, 795
  - ACTMEM, 796
  - ADJMETH, 796
  - ADMEXTU, 796
  - ADMPOST, 796
  - ADPCON, 796
  - ADSTAT, 797
  - AERODOF, 797
  - AESDISC, 797
  - AESMAXIT, 797
  - AESMETH, 797
  - AESRNDM, 797
  - AESTOL, 798
  - AGGROT, 798
  - ALPHA1, ALPHA2, 798
  - ALPHA1FL, ALPHA2FL, 799
  - ALTRED, 800
  - ARBMASP, 800
  - ARBMFEM, 800
  - ARBMNOW, 800
  - ARBMPS, 800
  - ARBMSS, 801
  - ARBMSTYP, 801
  - ARF, 801
  - ARS, 801
  - ASCOUP, 801
  - ASING, 801
  - AUNITS, 802
  - AUTOADJ, 802
  - AUTOGOUT, 802
  - AUTOMSET, 803
  - AUTOQSET, 803
  - AUTOSPC, 804
  - AUTOSPCR, 804
  - AUTOSPRT, 804
  - BAILOUT, 805
  - BEAMBEA, 805
  - BEIGRED, 805
  - BIGER, BIGER1, BIGER2, BIGER3, 805
  - BUCKLE, 805
  - BUSHNM, 806
  - CASIEMA, 806
  - CASIMEST, 806
  - CASPIV, 806
  - CB1, CB2, 807
  - CDIF, 807
  - CDITER, 807
  - CDPCH, 807
  - CDPRT, 807
  - CFDIAGP, 808
  - CFRANDEL, 808
  - CHECKOUT, 808
  - CK1, CK2, 809
  - CK3, 809
  - CLOSE, 810

- CM1, CM2, 810
- COMPMATT, 810
- CONFAC, 811
- COSUBCYC, 811
- COSUBMAX, 811
- COUPMASS, 812
- CP1, CP2, 812
- CQC, 812
- CURV, 812
- CURVPLOT, 815
- CWDIAGP, 816
- CWRANDEL, 816
- DBALL, 816
- DBCCONV, 817
- DBCDIAG, 817
- DBCOVWRT, 817
- DBDICT, 817
- DBDN, 817
- DBDRPRJ, 818
- DBDRVER, 818
- DBEXT, 818
- DBRCV, 818
- DBUP, 818
- DDRMM, 818
- DELCLUMP, 818
- DESPCH, 819
- DESPCH1, 819
- DFREQ, 820
- DIROUT, 820
- DOPT, 820
- DPEPS, 820
- DPHFLG, 820
- DSNOKD, 821
- DSO, 821
- DSZERO, 822
- DV3PASS, 822
- DYNSPCF, 822
- EIGFILT, 822
- ENFMETH, 822
- ENFMOTN, 823
- EPPRT, 824
- EPSILON, 824
- EPZERO, 824
- EPZERO is analogous to and described under the EPS keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPZERO., 824
- ERPC, 825
- ERPREFDB, 825
- ERPRHO, 825
- ERPRLF, 825
- ERROR, 825
- ESLFAV, 825
- ESLMOVE, 825
- ESLLCOMP, 826
- ESLMPC1, 826
- ESLOPTEX, 826
- ESLPRT, 827
- ESLPRT1, 827
- ESLRCF,user\_rc\_file, 827
- ESLSOPCV, 828
- ESLUNT2, 828
- ESLUNT1, 828
- EST, 828
- EULBND, 828
- EULSTRES, 829
- EUSUBCYC, 830
- EXCLUDE, 830
- EXTDR, 831
- EXTDROUT, 831
- EXTDRUNT, 831
- EXTOUT, 831
- EXTRCV, 831
- EXTUNIT, 831
- FACTOR, 831
- FASTFR, 831
- FBATOLR, 832
- FBLEND, 832
- FDRLDS, 833
- FIRSTKI, 833
- FIXEDB, 833
- FKSVMFAC, 834
- FLEXINCR, 834
- FLUIDMP, 835
- FLUIDNE, 835
- FLUIDSE, 835
- FMULTI, 835
- FOLLOWK, 835
- FRQDEPO, 836
- FULLSEDR, 836
- FZERO, 836
- G, GFL, 836
- GEOMU, 836
- GPECT, 836
- GRADMESH, 837
- GRAVSET, 837
- GRDPNT, 837
- GUSTAERO, 838

- GYROAVG, 838
- HEATCMD, 838
- HEATSTAT, 839
- HFREQ, HFREQFL, 839
- HTOCITS, 840
- HTOCPRT, 840
- HTOCTOL, 840
- HTSYM, 840
- ICOPT, 840
- IFP, 841
- INREL, 841
- INRLM, 841
- IRES, 842
- ITAPE, 842
- IUNIT, 842
- JWLDDET, 842
- KDAMP, KDAMPFL, 842
- KDIAG, 843
- KDMFILT, 843
- K4RITZ, 844
- K6ROT, 844
- LANGLE, 845
- LDSUM, 845
- LFREQ, LFREQFL, 846
- LGDISP, 846
- LMFACT, 847
- LMODES, LMODESFL, 847
- LOADU, 848
- LOOPID, 848
- LSTRN, 848
- MACH, 848
- MARALPHA, 848
- MARAUTO, 849
- MARBATCH, 850
- MARBK105, 850
- MARBK106, 850
- MARC4401, 851
- MARC7601, 851
- MARCASUM, 851
- MARCAUTO, 852
- MARCAXEL, 852
- MARCBEAM, 852
- MARCBODY, 853
- MARCBUSH, 853
- MARCCBAR, 853
- MARCCENT, 854
- MARCCON2, 854
- MARCCON3, 854
- MARCCPY, 854
- MARCDEF, 855
- MARCDILT, 856
- MARCDIS2, 856
- , 856
- MARCDIS3, 857
- MARCDIS4, 857
- MARCDMIG,N, 857
- MARCDUPE, 857
- MARCDYND, 858
- MARCEKND, 858
- MARCEXIT, 858
- MARCFEAT,N, 858
- MARCFILi, 858
- No Default, SOL 600 only., 858
- , 859
- MARCFRIC, 860
- MARCGAPD, D, 860
- MARCGAPN, ID, 860
- MARCGAPP, 860
- MARCGAUS, 860
- MARCGLUE, 861
- MARCGRAV, 861
- MARCHOST, 861
- MARCIAMN, 862
- MARCINTC, 862
- MARCINTF, 862
- MARCITER, 862
- MARCL001, 863
- MARCLOWE, 864
- MARCLUMP, 864
- MARCMAT2, 864
- MARCMAT3, 865
- MARCMATT, 865
- MARCMEM, Value, 866
- MARCMID3, 866
- MARCMNF, 866
- MARCMPCC, 866
- MARCMPiI, 867
- MARCND99, 867
- , 867
- MARCNOER, 868
- MARCOFFT, 868
- MARCONTF, 868
- MARCOOCC, 869
- MARCOPT, 869
- MARCOSET, 869
- MARCOTIM, 870
- MARCOUTR, 870
- MARCPARR, 870
- MARCPENT, 871
- MARCPINN, 871

- MARCPLAS, n, 871
- MARCPOS, 872
- MARCPOST, 872
- MARCPR99, 873
- MARCPRN, 873
- MARCPRNG, 873
- MARCPRNR, 874
- MARCPRNG, 874
- MARCRACC, 874
- MARCRBAL, 875
- MARCRBAR, 875
- MARCRBE2, 875
- MARCRBE3, 876
- MARCREVR, 876
- MARCREVRX, 876
- MARCRIGD, 877
- MARCSAME, 877
- MARCSCLR, 878
- MARCSETS, 878
- MARCSETT, 879
- MARCSINC, 879
- MARCSIZ3, Value, 879
- MARCSIZ4, Value, 879
- MARCSIZ5, Value, 879
- MARCSIZ6, Value, 880
- MARCSLHT, 880
- MARCSOLV, 880
- MARCSPCC, 881
- MARCSTIFF, Time, 881
- MARCSTOP, 881
- MARCSUMY, 881
- MARCT16, 882
- MARCT19, 883
- MARCTABL, 883
- MARCTEDF, 883
- MARCTEDN, 884
- MARCTEMP, 884
- MARCTETT, 884
- MARCTIEC, 885
- MARCTOL, 885
- MARCTOTD, 885
- MARCTOTL, 885
- MARCTOTT, 886
- MARCTUBE, 886
- MARCTVL, Value, 886
- MARCUSUB, chr, 887
- MARCVERS, 887
- MARCWDIS, 887
- MARCWELD, 888
- MARELSTO, 888
- MARGPFEL, 888
- MARFACEA, 888
- MARFACEB, 889
- MARFATAL, 889
- MARGPFOR, 889
- MARHEATM, 889
- MARHTPRT, 890
- MARIBOOC, 890
- MARIPROJ, 890
- MARLDCMB, 890
- MARLDRMV, 891
- MARMPCCHK, 891
- MARMPCID, 892
- MARMTLCK, 892
- MARNOCID, 893
- MARNOSET, Name, 893
- MAROFSET, 893
- MARPLANE, 893
- MARPROCS, 894
- MARRBAR2, 894
- MARROUTT, 894
- MARSHRII, 895
- MARTET10, 895
- MARTETIN, 896
- MARUPDAT, 896
- MARVFCUT, 897
- MAUTOSPC, 897
- MAXAPL, 897
- MAXDAMP, 897
- MAXIREVV, 898
- MAXLP, 898
- MAXRATIO, 898
- MBENDCAP, 899
- MCNLPARM, 899
- MCON2D3D, 899
- MCORDUPD, 899
- MCSSHLCK, 900
- MCSSHORR, 900
- MDAREAMD, 901
- MDEFSEPP, 901
- MDK4OPT, 902
- MDOPT14, 902
- MDOTM, 902
- MDOTMFAC, 902
- MDREDOPT, 903
- MDUMLoad, 903
- MECHFIL, 903
- MECHFIX, 903
- MECHPRT, 903
- MESH, 904

- METHCMRS, 904
- MEXTRNOD, 904
- MEXTSEE,N, 904
- MFASTCMP, 904
- MFEA5701, 905
- MFORCOR1, 905
- MFORDUPE, 905
- MFSKIPPP, 906
- MGAPINIT, 906
- MGLUETOL, 906
- MHEATSHL, 906
- MHEATUNT, 907
- MHEMPIX, 907
- MHOUBOLT, 907
- MHRED, 907
- MICRO , 908
- MIDNODE , 908
- MINCLDSB, 908
- MINIGOA, 909
- MINRECCC, N, 909
- MINSOUTT, 909
- MINVASHF, 909
- MINVCITR, 909
- MINVCSHF, 910
- MINVCTOL, 910
- MINVFMAX, 910
- MINVNMOD, 910
- MLDALLOW, 910
- MLSTRAIN, 910
- MLSTRAN2, 911
- MMAT2ANI, 912
- MMBOLTUS, 912
- MMEMDETT, 912
- MMFIL, 913
- MMMETH, 913
- MNASTLDS, 913
- MODACC, 914
- MODEL, 914
- MOFFCORE, 915
- MOUTRCOR, 915
- MP1SET, 915
- MPCX, 916
- MPERMPRT, 916
- MPTUNIT, 916
- MQUATERN, 916
- MRADUNIT, 917
- MRAFFLOR, N, 917
- MRAFFLOT, N, 917
- MRAFFLOW, Name, 917
- MRALIAS ID (MALIAS02, MALIAS03, etc.), 918
- MRALLOCG, 918
- MRALLOCS, 918
- MRBE3SNG, 919
- MRBEAMB, 919
- MRBEPARM, IJK, 919
- MRBDYCVT, 920
- MRBIGMEM, 920
- MRBUKMTH, 920
- Default = 2, SOL 600 only, 920
- MRC2DADD, 921
- MRCKBODY, 921
- MRCKLOAD, 921
- MRCONADD, 922
- MRCONRES, 922
- MRCONTAB, 922
- MRCONUSE, 923
- MRCONVER, 923
- MRCOORDS, 923
- MRCQUAD4, 924
- MRCTRIA3, 924
- MRCWANGL, 924
- MRDELTTT, 924
- MRDISCMB, 925
- MRDUPMAT, 926
- MRDYNOLD, 926
- MREIGMTH, 926
- MREL1103, 927
- MRELRB, 927
- MRELSPCD, 927
- MRENULEE, 928
- MRENUGRD, 928
- MRENUMBR, 929
- MRESTALL, 929
- MRESULTS, 929
- MREVPLST, 930
- MRFINITE, 930
- MRFOLLOW, 930
- MRFOLLOW2, 930
- MRFOLLOW1, 930
- MRFOLLOW3, 931
- MRFOLLOW4, 931
- MRGAPUSE, 931
- MRHERRMN, 932
- MRHYPMID, 932
- MRITTYPE,, 932
- MRMAT8A3, Value, 933
- MRMAT8E3, Value, 933
- MRMAT8N1, Value, 933
- MRMAT8N3, Value, 933

- MRMAXISZ, 933
- MRMAXNUM, 933
- MRMEMSUM, 933
- MRMTXKGG, Name, 934
- MRMTXNAM, Name, 934
- MRNOCOMP, 934
- MRNOCOR, 935
- MRNOECHO, 935
- MRORINTS, 935
- MROUTLAY, 936
- MRPARALL, 937
- MRPBUSHT, 937
- MRPIDCHK, 938
- MRPELAST, 938
- MRPLoad4, 938
- MRPLoD4R, 939
- MRPLSUPD, 939
- MRPOISCK, 939
- MRPREFER, 940
- MRPRSFAC, 940
- MRPSHCMP, 941
- MRPSHELL, 941
- MRRBE3TR, 942
- MRRCFILE, RCF, 942
- MRRELNOD, 943
- MRRSTOP2, 943
- MRSCMOD, 943
- MRSETNA1, N, 944
- MRSETNA2, M, 944
- MRSETNAM, N, 944
- MRSPAWN2, CMD, 945
- MRSPLINE, 945
- MRSPRING, 945
- MRSPRVER, 946
- MRSTEADY, 946
- MRT16STP, N, 946
- MRTABLIM, 946
- MRTABLS1, 947
- MRTABLS2, 948
- MRTABPRC, 948
- MRTFINAL, 948
- MRTHREAD, 948
- MRTIMING, 949
- MRTSHEAR, 949
- MRUSDATA, 949
- MRVFIMPL, 950
- MRV09V11, 950
- MSG10625, 950
- MSIZOVRD, 951
- MSOLMEM, MBYTE, 951
- Default = Program determined value, SOL  
600 only., 951
- MSPCCCHK, 952
- MSPCKEEP, 952
- MSPEEDCB, 952
- MSPEEDCW, 953
- MSPEEDOU, 953
- MSPEEDP4, 953
- MSPEEDPS, 954
- MSPEEDSE, 954
- MSPEEDS2, 954
- MSPEEDSH, 954
- MSPLINC0, 955
- MSTFBEAM, 955
- PARFREE, 955
- MSTTDYNE, 956
- MTABIGNR, 956
- MTABLD1M, 957
- MTABLD1T, 957
- MTEMPCHK, 957
- MTEMPDWN, 958
- MTET4HYP, 958
- MTLD1SID, 958
- MUALLUDS, 959
- MULRFORC, 959
- MUMPSOLV, 959
- MUSBKEEP, 960
- MVERMOON, 960
- MWINQUOT, 960
- MWLDGSGA, 960
- MXICODE0, 961
- MXLAGM1, 961
- NASPRT, 961
- NDAMP, NDAMPM, 961
- NEWMARK, 963
- NFDOPts, 963
- NHPLUS, 963
- NINTPTS, 963
- NLAYERS, 963
- NLHTLS, 963
- NLHTOL, 963
- NLHTWK, 964
- NLMAX, 964
- NLMIN, 964
- NLPACK, 964
- NLTOL, 965
- NMLOOP, 965
- NMNLFrq, 966
- NOCOMPS, 966
- NODCMP, 966



- NOELOF, 966
- NOELOP, 966
- NOGPF, 967
- NONCUP, 967
- NQSET, 967
- NUMOUT, NUMOUT1, NUMOUT2,  
NUMOUT3, 967
- OELMOPT, 967
- OELMSET, 968
- OG, 968
- OGEOM, 968
- OGRDOPT, 968
- OGRDSET, 968
- OLDSEQ, 968
- OLDWELD, 971
- OMACHPR, 971
- OMAXR, 971
- OMID, 971
- OMSGLVL, 971
- OP2GM34, 971
- OPCHSET, 972
- OPGEOM, 972
- OPGTKG, 972
- OPPHIB, 972
- OPPHIPA, 972
- OPTEXIT, 973
- OPTION, 973
- OSETELE, 973
- OSETGRD, 974
- OSWELM, 974
- OSWPPT, 974
- OUGCORD, 974
- OUGSPLIT, 974
- OUNIT1, 975
- OUNIT2, 975
- OUTOPT, 975
- PACINFID, 975
- PANELMP, 975
- PATPLUS, 975
- PATVER, 975
- PCOMPRM, 975
- PDRMSG, 976
- PEMDMP, 976
- PEMFRIM, 976
- PEMMAPC, 976
- PEMNCOP4, 977
- PEMNPART, 977
- PEMSKIN, 978
- PENFN, 978
- PERCENT, 978
- PH2OUT, 978
- PKRSP, 978
- PLTMSG, 978
- POST, 979
- POSTEXT, 984
- POSTADF, 985
- POSTU, 985
- PREFDB, 985
- PRGPST, 985
- PRINT, 986
- PROUT, 986
- PRPA, 986
- PRPJ, 986
- PRPHIVZ, 986
- PRTMAXIM, 986
- PRTRESLT, 987
- PSENPC, 987
- PTHRES, 987
- Q, 987
- RADMOD, 987
- RBSETPRT, 987
- RESLTOPT, 987
- RESVEC, 988
- RFORSET, 988
- RHOC, 988
- RIMINTP, 988
- RKSCHME, 988
- RMSINT, 989
- ROHYDRO, 989
- ROMULTI, 990
- ROSTR, 990
- RSPECTRA, 991
- RSPRINT, 991
- RSTTEMP, 991
- RVLDFILT, 991
- S1, S1G, S1M, 992
- S1AG,S1AM, 993
- SBSPFORM, 994
- SCRSPEC, 994
- SDCSV, 995
- SDUNIT,n, 995
- SEKD, 995
- SEMAP, SEMAPOPT, SEMAPPRT, 995
- SENSU00, 997
- SEP1XOVR, 997
- SEQOUT, 997
- SERST, 997
- SESDAMP, 998
- SESEF, 998
- SHIFT1, 998

- SHLDAMP, 999
- SIGMA, 999
- SKINOUT, 999
- SKPAMP, 999
- SLOOPID, 1000
- SMALLQ, 1000
- SNORM, 1000
- SNORMPRT, 1001
- SOFTEXT, 1001
- SOLADJC, 1001
- SOLID, 1002
- SPARSED, 1002
- SPARSEDR, 1002
- SPARSEPH, 1003
- SPDRRAT, 1003
- SPCGEN, 1003
- SPDMRAT, 1003
- SQSETID, 1003
- SRCOMPS, 1003
- SRTELTY, 1004
- SRTOPT, 1004
- START, 1004
- STEPFACT, 1004
- STEPFACTL, 1004
- STIME, 1004
- STRUCTMP, 1005
- SUBCASID, 1005
- SUBID, 1005
- SUPAERO, 1005
- SUPDOF, 1005
- SUPER, 1006
- TABID, 1006
- TABS, 1006
- TDAMP, 1006
- TESTNEG, 1006
- TFSYMFAC, 1007
- TINY, 1007
- TOLRSC, 1007
- TOPOCONV, 1008
- TRMBIM, 1008
- TSTATIC, 1008
- UGASC, 1008
- UNSYMF, 1009
- UNSYMK, 1009
- UPDTBSH, 1009
- USETPRT, 1009
- USETSEL, 1010
- USETSTRI, 1010
- VARPHI, 1011
- VELCUT, 1011
- VMOPT, 1012
- VREF, 1012
- WHIRLOPT, 1012
- WMODAL, 1012
- WRBEAMB, 1013
- WTMAS600, 1013
- WTMAS, 1013
- W3, W4, W3FL, W4FL, 1014
- WR3, WR4, WRH, 1015
- XFACT, 1015
- XFACTX, 1015
- XFLAG, 1015
- XYMPCH, 1016
- XYUNIT, n, 1016
- ZROCMAS, 1016
- ZROVEC, 1017

■ Parameter Applicability Tables, 1017

■ Item Code Description, 1052

■ Element Stress (or Strain) Item Codes, 1052

- CAXIF2 (47) 1053
- CAXIF3 (48) 1053
- CAXIF4 (49) 1053
- CAXISYM (241) 1054
- CBAR (34) 1054
- CBAR (100) 1055
- CBAR (238) 1056
- CBEAM (2) 1056
- CBEAM (94) 1056
- CBEAM (239) 1057
- CBEAM3 (184) 1057
- CBEND (69) 1058

- CBUSH (102) 1058
- CBUSH1D (40) 1059
- CCONEAX (35) 1059
- CDUM3 1060
- CDUM9 (55-61) 1060
- CELAS1 (11) 1060
- CELAS2 (12) 1060
- CELAS3 (13) 1060
- CGAP (86) 1060
- CHEXA (67) 1060
- CHEXA (93) 1061
- CHEXAFD (202) 1062

CHEXAFD (207) 1062  
CIFHEX (65) 1062  
CIPFENT (66) 1063  
CIFQDX (73) 1063  
CIFQUAD (63) 1063  
CONROD (10) 1064  
CONROD (92) 1064  
CPENTA (68) 1064  
CPENTA (91) 1065  
CPENTA FD (204) 1066  
CPENTA FD (209) 1066  
CPYRAM (242) 1067  
CPYRAM (242) 1068  
CQUAD4 (33) 1068  
CQUAD4 (90) 1069  
CQUAD42 (95) 1069  
CQUAD4 (144) 1070  
CQUAD8 (64) 1071  
CQUAD82 (96) 1071  
CQUADFD (201) 1072  
CQUADFD (208) 1072  
CQUADR (82) 1072  
CQUADR (172) 1072  
CQUADR (232) 1072  
CQUADX(18) 1073  
CQUADXFD (214) 1073

CQUADXFD (215) 1074  
CROD (1) 1074  
CROD (89) 1074  
CSHEAR (4) 1074  
CSLOT3 (50) 1074  
CSLOT4 (51) 1074  
CTETRA (39) 1075  
CTETRA (85) 1076  
CTETRAFD (205) 1076  
CTETRAFD (210) 1076  
CTRIA3 (74) 1077  
CTRIA32 (97) 1077  
CTRIA3 (88) 1077  
CTRIA6 (75) 1077  
CTRIA62 (98) 1077  
CTRIA FD (206) 1077  
CTRIA FD (211) 1078  
CTRIAR (70) 1078  
CTRIAR (173) 1078  
CTRIAR (233) 1078  
CTRIAX(17) 1078  
CTRIAX6 (53) 1078  
CTRIAXFD (212) 1079  
CTRIAXFD (213) 1079  
CTUBE (3) 1079  
CTUBE (87) 1079

■ Element Force Item Codes, 1080

CWELDP 1080  
(118) 1080  
CWELDC 1080  
(117) 1080  
CWELD 1080  
(200) 1080  
CBAR (34) 1080  
CBAR (100) 1081  
CBEAM (2) 1081  
CBEAM3 (184) 1082  
CBEND (69) 1083  
CBUSH (102) 1083  
CCONEAX (35) 1083  
CDAMP1 (20) 1084  
CDAMP2 (21) 1084  
CDAMP3 (22) 1084  
CDAMP4 (23) 1084  
CDUM3 thru CDUM9 (55 - 61) 1084  
CELAS1 (11) 1084  
CELAS2 (12) 1084  
CELAS3 (13) 1084  
CELAS4 (14) 1084

CGAP (38) 1085  
CONROD (10) 1085  
CQUAD4 (33) 1085  
CQUAD4 (95) 1085  
CQUAD4 (144) 1086  
CQUAD8 (64) 1087  
CQUAD82 (96) 1088  
CQUADR (82) 1088  
CQUADR (235) 1088  
CROD (1) 1088  
CSHEAR (4) 1088  
CTRIA3 (74) 1089  
CTRIA32 (97) 1089  
CTRIA6 (75) 1089  
CTRIA62 (98) 1090  
CTRIAR (70) 1090  
CTRIAR (236) 1090  
CTUBE (3) 1090  
CVISC (24) 1090  
CWELDP (118) 1090  
CWELDC (117) 1090  
CWELD (200) 1091

■ Fluid Virtual Mass Pressure Item Codes, 1091

■ 2D Slideline and 3D Surface Contact Item Codes, 1092

CSLIFID (116) 1092                      203 1093  
203 1093

■ Element Strain Energy Item Codes, 1094

■ Fatigue Item Codes, 1095

■ Equivalent Radiated Power (ERP) Item Codes, 1101



# 8

## Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions, 1104
- Degree-of-Freedom Set Bulk Data Entries, 1108

# 9

## Bulk Data Entries

- Key to Descriptions, 1110
  - The Bulk Data Section, 1111
- Bulk Data Entry Descriptions, 1112
  - Format of Bulk Data Entries, 1113
  - Continuations, 1118
  - Bulk Data Entry Summary, 1126
  - Constraints and Partitioning, 1127
  - Elements, 1129
  - Geometry, 1138
  - Loads, 1143
  - Materials, 1146
  - Miscellaneous, 1148
  - Nastran Implicit Nonlinear (SOL 600), 1153
  - Explicit Nonlinear (SOL 700), 1156
  - Solution Control, 1167
- - \$, 1172
  - /, 1173
  - ABINFL, 1177
  - ABSNMVB, 1179
  - ABSNMVI, 1180
  - ABSNMVR, 1181
  - ABSNMVS, 1182
  - ABSTRCT, 1184
  - ACCEL, 1185
  - ACCEL1, 1187
  - ACCSSPT, 1188
  - ACLOAD, 1189
  - ACMODL, 1190
  - ACPEMCP, 1196
  - ACSRCE, 1199
  - ACTIVAT, 1201
  - ACTRIM, 1202
  - ADUMi, 1203
  - AECOMP, 1204
  - AECOMPL, 1205
  - AEDW, 1206
  - AEFACT, 1207
  - AEFORCE, 1208
  - AEGRID, 1209
  - AELINK, 1210
  - AELIST, 1211
  - AELISTC, 1212
  - AEPARM, 1213
  - AEPRESS, 1214
  - AEQUAD4, 1215
  - AERO, 1216
  - AEROS, 1217
  - AESCALE, 1218
  - AESTAT, 1219
  - AESURF, 1221
  - AESURFS, 1223
  - AETRIA3, 1224
  - ALIASM, 1225
  - ASET, 1227
  - ASET1, 1228
  - ATBACC, 1230
  - ATBJNT, 1231
  - ATBSEG, 1233
  - AXIC, 1234
  - AXIF, 1236
  - AX SLOT, 1238
  - BAROR, 1239
  - BARRIER, 1241
  - BCAUTOP, 1243
  - BCBDRP, 1246
  - BCBMRAD, 1253
  - BCBODY, 1255

- BCBODY, 1267
- BCBODY1, 1282
- BCBOX, 1284
- BCBIER, 1286
- BCELIPS, 1288
- BCGRID (SOL 700 only), 1289
- BCGRID, 1290
- BCHANGE, 1291
- BCHANGE, 1293
- BCMATL, 1295
- BCMOVE, 1296
- BCMOVE, 1297
- BCNUB2, 1299
- BCNUB3, 1301
- BCONECT, 1303
- BCONP, 1305
- BCONPRG, 1307
- BCONPRG-700, 1316
- BCONPRP, 1322
- BCONPRP-700, 1327
- BCONUDS, 1329
- BCONUDS, 1331
- BCPARA, 1333
- BCPARA, 1342
- BCPATCH, 1348
- BCPFLG, 1349
- BCPROP, 1352
- BCPROP, 1353
- BCRIGID, 1354
- BCRGSRF, 1356
- BCSCAP, 1357



- CAABSF, 1442
  - CACINF3, 1444
  - CACINF4, 1445
  - CAERO1, 1446
  - CAERO2, 1448
  - CAERO3, 1450
  - CAERO4, 1452
  - CAERO5, 1454
  - CAMPBLL, 1458
  - CAXIFI, 1460
  - CAXISYM, 1461
  - CBAR, 1463
  - CBARAO, 1468
  - CBEAM, 1470
  - CBEAM3, 1474
  - CBEND, 1478
  - CBUSH, 1481
- BCSEG, 1359
  - BCSURF, 1360
  - BCTABLE, 1370
  - BCTABLE - Glued Option, 1385
  - BCTABL1, 1389
  - BCTRIM, 1391
  - BDYLIST, 1392
  - BDYOR, 1393
  - BEADVAR, 1394
  - BEAMOR, 1398
  - BFRIC, 1400
  - BIAS, 1401
  - BJOIN, 1404
  - BLSEG, 1406
  - BNDFIX, 1408
  - BNDFIX1, 1409
  - BNDFREE, 1411
  - BNDFRE1, 1412
  - BNDGRID, 1414
  - BOLT, 1415
  - BOLT1, 1417
  - BOUTPUT, 1420
  - BRKPROP, 1422
  - BRKSQL, 1423
  - BRKSYS, 1426
  - BSET, 1428
  - BSET1, 1430
  - BSQUEAL, 1432
  - BSURF, 1438
  - BWIDTH, 1440
- CBUSH1D, 1485
  - CBUSH2D, 1486
  - CCONEAX, 1487
  - CDAMP1, 1488
  - CDAMP1D, 1489
  - CDAMP2, 1490
  - CDAMP2D, 1491
  - CDAMP3, 1492
  - CDAMP4, 1493
  - CDAMP5, 1494
  - CDUMi, 1495
  - CELAS1, 1496
  - CELAS1D, 1497
  - CELAS2, 1499
  - CELAS2D, 1500
  - CELAS3, 1502
  - CELAS4, 1503

- CFAST, 1504
- CFLUIDi, 1506
- CGAP, 1508
- CHACAB, 1510
- CHACBR, 1512
- CHBDYE, 1514
- CHBDYG, 1518
- CHBDYP, 1522
- CHEXA, 1526
- CIFHEX, 1530
- CIFPENT, 1532
- CIFQDX, 1534



- COHESIV, 1550
- COHFRIC, 1552
- COMPUDS, 1552
- CONCTL, 1556
- CONM1, 1557
- CONM2, 1558
- CONROD, 1561
- CONTRLT, 1563
- CONV, 1567
- CONVM, 1569
- CORD1C, 1572
- CORD1R, 1574
- CORD1S, 1576
- CORD2C, 1578
- CORD2R, 1580
- CORD2S, 1582
- CORD3G, 1584
- CORD3R, 1586
- COSMGRP, 1587
- COSMINP, 1588
- COSMOUT, 1589
- COSMSEL, 1590
- COSMSRV, 1591
- COUCOHF, 1591
- COUOPT, 1593
- COUP1FL, 1595
- COUPINT, 1596
- COUPLE, 1597
- CPENTA, 1601
- CPYRAM, 1605
- CQUAD, 1607
- CQUAD4, 1609
- CQUAD8, 1613
- CQUADR, 1617



- DAMPING, 1703
- DAREA, 1706

- CIFQUAD, 1536
- CINTC, 1538
- CLOAD, 1539
- CMARKB2, 1540
- CMARKN1, 1541
- CMASS1, 1542
- CMASS2, 1543
- CMASS3, 1544
- CMASS4, 1545
- CMREBAI, 1546
- CMREBAR, 1548

- CQUADX, 1620
- CRAC2D, 1622
- CRAC3D, 1624
- CREEP, 1628
- CROD, 1634
- CSEAM, 1636
- CSET, 1645
- CSET1, 1647
- CSHEAR, 1649
- CSLOT3, 1652
- CSLOT4, 1653
- CSPR, 1654
- CSSCHD, 1655
- CSSHL, 1656
- CSSHLH, 1658
- CSSHLM, 1659
- CSSHLP, 1660
- CSUPER, 1661
- CSUPEXT, 1663
- CTETRA, 1664
- CTRIA3, 1667
- CTRIA6, 1670
- CTRIAR, 1674
- CTRIAX, 1677
- CTRIAX6, 1679
- CTUBE, 1683
- CVISC, 1684
- CWELD, 1685
- CYAX, 1696
- CYJOIN, 1697
- CYLINDR, 1699
- CYSUP, 1700
- CYSYM, 1702

- DCONADD, 1707
- DCONSTR, 1708
- DDVAL, 1710
- DEACTEL, 1712
- DEACTEL, 1713
- DEFORM, 1714
- DEFUSET, 1715
- DELAY, 1716
- DEQATN, 1717
- DESVAR, 1722
- DETSPH, 1723
- DIVERG, 1724
- DLINK, 1725
- DLOAD, 1726
- DMI, 1727
- DMIAX, 1731
- DMIG, 1733
- DMIG,UACCEL, 1736
- DMIGOUT, 1738
- DMIGROT, 1742
- DMIJ, 1744
- DMIJI, 1747
- DMIK, 1750
- DOPTPRM, 1753
- DPHASE, 1761
- DRESP1, 1762
- DRESP2, 1775
- DRESP3, 1781
- DSCREEN, 1785
- DTABLE, 1787
- DTABLE2, 1788
- DTI, 1789
- DTI,ESTDATA, 1791
- DTI,INDTA, 1795
- DTI,SETREE, 1798
- DTI,SPECSEL, 1800
- DTI,SPSEL, 1801
- DTI,UNITS, 1802
- DVBSHAP, 1804
- DVCREL1, 1805
- DVCREL2, 1807
- DVGRID, 1809
- DVLREL1, 1811
- DVMREL1, 1813
- DVMREL2, 1815
- DVPREL1, 1817
- DVPREL2, 1819
- DVSHAP, 1821
- DVPSURF, 1822
- DYFSISW, 1823
- DYPARAM, ATBAOUT, 1824
- DYPARAM, ATBHOUT, 1825
- DYPARAM, ATBTOUT, 1826
- DYPARAM,AUTOCOUP, 1827
- DYPARAM,AXIALSYM, 1829
- DYPARAM,AXREMAP, 1831
- DYPARAM,BULKL, 1832
- DYPARAM,BULKQ, 1833
- DYPARAM,BULKTY, 1834
- DYPARAM,CFULLRIG, 1835
- DYPARAM,CLUFLIM, 1836
- DYPARAM,CLUMPENR, 1837
- DYPARAM,CONM2OUT, 1838
- DYPARAM,CONTACT, 1839
- DYPARAM,COSUBMXT, 1842
- DYPARAM,COUFRIC, 1843
- DYPARAM,COHESION, 1844
- DYPARAM,DMPOPT, 1845
- DYPARAM,DUMSEGS, 1846
- DYPARAM,ELDLTH, 1847
- DYPARAM,EULERCB, 1848
- DYPARAM,EULERPR, 1849
- DYPARAM,EULTRAN, 1851
- DYPARAM,EUSUBMAX, 1852
- DYPARAM,EUSUBCYC, 1854
- DYPARAM,FAILDT, 1855
- DYPARAM,FAILOUT, 1856
- DYPARAM,FLOWMETH, 1857
- DYPARAM,GEOCHECK, 1860
- DYPARAM,FASTCOUP, 1861
- DYPARAM,HYDROBOD, 1863
- DYPARAM,HGCMEM, 1864
- DYPARAM,HGCOEFF, 1865
- DYPARAM,HGCSOL, 1866
- DYPARAM,HGCTWS, 1867
- DYPARAM,HGCWRP, 1868
- DYPARAM,HGSHELL, 1869
- DYPARAM,HGSOLID, 1870
- DYPARAM,HGTYPE, 1871
- DYPARAM,HICGRAV, 1872
- DYPARAM,HVLF, 1873
- DYPARAM,IMM, 1874
- DYPARAM,INFOBJ, 1875
- DYPARAM,INISTEP, 1876
- DYPARAM,LAGPR, 1877
- DYPARAM,LIMCUB, 1878
- DYPARAM,MATRMERG, 1879
- DYPARAM,MATRMRG1, 1880
- DYPARAM,MAXSTEP, 1881
- DYPARAM,MESHPLN, 1882

- DYPARAM,MINSTEP, 1883
- DYPARAM,MIXGAS, 1884
- DYPARAM,NZEROVEL, 1885
- DYPARAM,OLDLAGT, 1886
- DYPARAM,JWLDET, 1887
- DYPARAM,LIMITER, 1888
- DYPARAM,PARALLEL, 1889
- DYPARAM,PLCOVCUT, 1890
- DYPARAM,PMINFAIL, 1891
- DYPARAM,RBE2INFO, 1892
- DYPARAM,RHOCUT, 1893
- DYPARAM,RJSTIFF, 1894
- DYPARAM,SCALEMAS, 1895
- DYPARAM,SHELLFRM, 1896
- DYPARAM,SHELMSYS, 1897
- DYPARAM,SHPLAST, 1898
- DYPARAM,SHSTRDEF, 1899
- DYPARAM,SHTHICK, 1900
- DYPARAM,SLELM, 1901
- DYPARAM,SMP,BATCHSIZ, 1902
- DYPARAM,SMP,CPUINFO, 1904
- DYPARAM,SNDLIM, 1905
- DYPARAM,SPREMAP, 1906
- DYPARAM,SPHERSYM, 1907
- DYPARAM,STRNOUT, 1908
- DYPARAM,TOLCHK, 1909
- DYPARAM,VDAMP, 1910
- DYPARAM,VELMAX, 1911
- DYPARAM,VELMAX1, 1912
- DYPARAM,VISCPLAS, 1914
- ECHOOFF, 1915
- ECHOON, 1916
- EIGB, 1917
- EIGC, 1919
- EIGP, 1924
- EIGR, 1925
- EIGRL, 1929
- ELEMUDS, 1933
- ELEMUDS, 1935
- ELIST, 1937
- ENDDATA, 1938
- ENDDYNA, 1939
- ENTUDS, 1940
- EOSDEF, 1942
- EOSGAM, 1945
- EOSIG, 1947
- EOSJWL, 1953
- EOSMG, 1955
- EOSNA, 1957
- EOSPOL, 1959
- EOSUDS, 1961
- EPOINT, 1962
- ERPPNL, 1963
- EULFOR, 1964
- EULFOR1, 1965
- EULFREG, 1966
- EXCLUDE, 1968
- EXTRN, 1969
- FAILJC, 1970
- FAILMPS, 1972
- FAILUDS, 1973
- FBADLAY, 1975
- FBALOAD, 1976
- FBAPHAS, 1977
- FBODYLD, 1978
- FBODYSB, 1979
- FFCONTR, 1980
- FLFACT, 1982
- FLOW, 1984
- FLOWC, 1986
- FLOWDEF, 1989
- FLOWT, 1991
- FLOWUDS, 1994
- FLSYM, 1996
- FLUTTER, 1997
- FORCE, 2000
- FORCE1, 2001
- FORCE2, 2002
- FORCEAX, 2003
- FORCUDS, 2004
- FREEPT, 2006
- FREQ, 2007
- FREQ1, 2008
- FREQ2, 2009
- FREQ3, 2011
- FREQ4, 2014
- FREQ5, 2017
- FRFCOMP, 2019
- FRFCONN, 2022
- FRFFLEX, 2024
- FRFRELS, 2026
- FRFSPC1, 2027
- FRFXIT, 2028
- FRFXIT1, 2029
- FSICTRL, 2030
- FSLIST, 2031



- FTGDEF, 2032
- FTGEVNT, 2037
- FTGLOAD, 2039
- FTGPARM, 2044
- FTGSEQ , 2058
- GBAG, 2062
- GBAGCOU, 2065
- GENEL, 2066
- GENUDS, 2069
- GMBNDC, 2070
- GMNURB, 2071
- GMOV3, 2074
- GMQVOL, 2075
- GRAV, 2076
- GRDSET, 2078
- GRID, 2079
- GRIDA, 2081
- GRIDB, 2082
- GRIDF, 2083
- GRIDS, 2084
- GUST, 2085
- HADACRI, 2086
- HADAPTL, 2090
- HEATLOS, 2092
- HGSUPPR, 2094
- HTRCONV, 2096
- HTRRAD, 2097
- HYBDAMP, 2098
- HYDROC, 2100
- HYDROS, 2104
- HYDSTAT, 2107
- IMPCASE, 2109
- IMPGEOM, 2110
- INCLUDE, 2114
- INFLCG, 2115
- INFLFRC, 2117
- INFLGAS, 2119
- INFLHB, 2121
- INFLTNK, 2122
- INFLTR, 2124
- INITGAS, 2126
- IPSTRAIN, 2127
- IPSTRN, 2128
- ISTRESS, 2129
- ITER, 2131
- LBSH3DG, 2134
- LEAKAGE, 2136
- LOAD, 2139
- LOADCLID, 2141
- LOADCNAM, 2142
- LOADCSUB, 2143
- LOADCYH, 2144
- LOADCYN, 2146
- LOADCYT, 2147
- LOADOF, 2148
- LOADT, 2149
- LORENZI, 2151
- LSEQ, 2155

■ , 2157

- MACREEP, 2157
- MARCIN, 2159
- MARCOUT, 2161
- MARPRN, 2169
- MASSSET, 2171
- MATBV, 2172
- MAT1, 2173
- MAT1A, 2177
- MAT1F, 2178
- MAT2, 2179
- MAT2F, 2182
- MAT3, 2184
- MAT4, 2186
- MAT5, 2188
- MAT8, 2189
- MAT8A, 2193
- MAT8F, 2197
- MAT9, 2198
- MAT9F, 2201
- MAT10, 2202
- MAT10F, 2204
- MATDEUL, 2205
- MATDIGI, 2207
- MATEP, 2209
- MATEP, 2220
- MATEP, 2235
- MATF, 2246
- MATF, 2260
- MATF, 2271
- MATF1, 2277
- MATFAB, 2278
- MATFTG , 2282
- MATG, 2294
- MATHE, 2297
- MATHE, 2305
- MATHED, 2309
- MATHP, 2312
- MATNLE, 2315

- MATORT , 2320
- MATPE1, 2325
- MATRIG, 2327
- MATS1, 2329
- MATS3, 2334
- MATS8, 2336
- MATSMA, 2337
- MATSORT, 2341
- MATT1, 2343
- MATT2, 2345
- MATT3, 2347
- MATT4, 2348
- MATT5, 2349
- MATT8, 2350
- MATT9, 2352
- MATTEP, 2353
- MATTEP, 2355
- MATTF, 2357
- MATTG, 2360
- MATTHE, 2361
- MATTORT, 2364
- MATTUSR, 2365
- MATTVE, 2366
- MATUDS, 2370
- MATUDS, 2373
- MATUSR, 2376
- MATVE, 2379
- MATVE, 2384
- MATVP, 2386
- MATVP, 2390
- MAUXCMD, 2394
- MBOLT, 2395
- MBOLTUS, 2397
- MCHSTAT, 2399
- MCOHE, 2401
- MDBCATP, 2403
- MDBCNCCT, 2406
- MDBCPCAR, 2408
- MDBCTB1, 2409
- MDBKSYS, 2411
- MDBNDRY, 2413
- MDBOLT, 2414
- MDBULK, 2416
- MDCONCT, 2419
- MDDMIG, 2421
- MDELAM, 2423
- MDEXCLD, 2424
- MDFAST, 2425
- MDFSET, 2427
- MDLABEL, 2428
- MDLOC, 2429
- MDLPRM, 2431
- MDMIAUX, 2446
- MDMIOUT, 2447
- MDMIR1, 2449
- MDMIR2, 2450
- MDMOVE, 2451
- MDMPC, 2452
- MDMPLN, 2453
- MDRBE2, 2454
- MDRBE3, 2455
- MDRJNT, 2457
- MDROT1, 2458
- MDROT2, 2459
- MDRROD, 2460
- MDSEAM, 2461
- MDSSET, 2463
- MDTRAN, 2464
- MDWELD, 2465
- MESH, 2470
- MESUPER, 2477
- METADATA, 2480
- MFLUID, 2481
- MGRSPR, 2483
- MINSTAT, 2484
- MISLAND, 2486
- MIXTURE, 2487
- MKAERO1, 2489
- MKAERO2, 2490
- MLAYOUT, 2491
- MNF600, 2492
- MODTRAK, 2494
- MOMAX, 2495
- MOMENT, 2496
- MOMENT1, 2497
- MOMENT2, 2498
- MONCARL, 2499
- MONCNCM, 2501
- MONDSP1, 2502
- MONGRP, 2504
- MONPNT1, 2505
- MONPNT2, 2507
- MONPNT3, 2509
- MONSUM, 2512
- MONSUM1, 2515
- MONSUMT, 2518
- MOTION, 2520
- MPC, 2521
- MPCADD, 2522
- MPCAX, 2523

- MPCD, 2524
- MPCREEP, 2525
- MPCY, 2527
- MPHEAT, 2529
- MPROCS, 2530
- MREVERS, 2532
- MRSSCON, 2534
- MSTACK, 2536
- MT16SEL, 2537
- MT16SPL, 2539
- MTABRV, 2542
- MTCREEP, 2544
- MOTHERM, 2546
- NHRMPRM, 2547
- NLADAPT, 2548
- NLAUTO, 2550
- NLBSH3D, 2555
- NLCYSYM, 2556
- NLDAMP, 2560
- NLFREQ, 2561
- NLFREQ1, 2562
- NLHARM, 2563
- NLHEATC, 2564
- NLMOPTS, 2565
- NLOUT, 2581
- NLOUTUD, 2584
- NLPARM, 2586
- NLPCI, 2595
- NLRGAP, 2597
- NLRSD, 2601
- NLSTEP, 2604
- NLSTRAT, 2619
- NOLIN1, 2629
- NOLIN2, 2631
- NOLIN3, 2633
- NOLIN4, 2635
- NSM, 2637
- NSM1, 2638
- NSMADD, 2640
- NSML, 2641
- NSML1, 2642
- NTHICK, 2644
- OMIT, 2645
- OMIT1, 2646
- OMITAX, 2648



- PAABSF, 2649
- PACABS, 2651
- PACBAR, 2652
- PACINF, 2653
- PAER01, 2654
- PAER02, 2655
- PAER03, 2657
- PAER04, 2659
- PAER05, 2661
- PANEL, 2665
- PARAM, 2666
- PARAMARC, 2668
- PAXISYM, 2671
- PAXSYMH, 2673
- PBAR, 2675
- PBARL, 2678
- PBARN1, 2684
- PBCOMP, 2688
- PBEAM, 2694
- PBEAM3, 2699
- PBEAMD, 2702
- PBEAML, 2703
- PBELT, 2711
- PBEMN1, 2714
- PBEND, 2718
- PBMARB6, 2727
- PBMNUM6, 2730
- PBMSECT, 2734
- PBRSECT, 2740
- PBSH2DT, 2743
- PBUSH, 2745
- PBUSH1D, 2750
- PBUSH2D, 2755
- PBUSHT, 2757
- PCOHE, 2763
- PCOMP, 2764
- PCOMPA, 2769
- PCOMPF, 2771
- PCOMPFQ, 2773
- PCOMPG, 2774
- PCOMPLS, 2779
- PCONEAX, 2784
- PCONV, 2786
- PCONVM, 2790
- PCONV1, 2792
- PDAMP, 2794
- PDAMP5, 2795
- PDAMPT, 2796
- PDISTB, 2797
- PDISTBM, 2800

- PDUMi, 2802
- PELAS, 2803
- PELAS1, 2804
- PELAST, 2805
- PERMEAB, 2807
- PERMGBG, 2809
- PEULER, 2811
- PEULER1, 2813
- PFAST, 2814
- PFASTT, 2819
- PFTG , 2821
- PGAP, 2825
- PHBDY, 2828
- PLCOMP, 2829
- PLOAD, 2833
- PLOAD1, 2835
- PLOAD2, 2838
- PLOAD4, 2840
- PLOADB3, 2843
- PLOADX1, 2845
- PLOTEL, 2848
- PLPLANE, 2849
- PLSOLID, 2851
- PLTSURF, 2852
- PMARKER, 2853
- PMASS, 2854
- PMINC, 2855
- PMREBAI, 2857
- PMREBAR, 2861
- POINT, 2864
- POINTAX, 2865

■ , 2937

- QBDY1, 2937
- QBDY2, 2938
- QBDY3, 2939
- QHBDY, 2940
- QSET, 2942
- QSET1, 2943
- QVECT, 2945
- QVOL, 2948
- RADBC, 2950
- RADBND, 2951
- RADC, 2952
- RADCAV, 2954
- RADCOL, 2957
- RADCT, 2958
- RADLST, 2959
- RADM, 2961
- RADMT, 2962
- RADMTX, 2963
- RADSET, 2964
- RANDPS, 2965
- RANDT1, 2967
- RBAR, 2968
- RBAR1, 2970
- RBAX3D, 2972
- RBE1, 2974
- RBE2, 2976
- RBE2GS, 2978
- RBE3, 2981
- RBE3U, 2984
- RBJOINT, 2985
- RCONN, 2988
- RCPARM, 2990
- RCROSS, 2993
- REFROT, 2995

- REFRTSE, 2997
- RELEASE, 2999
- RELEX, 3000
- RESTART, 3001
- RFORCE, 3004
- RGYRO, 3008
- , 3022
  - ROTHYBD, 3023
  - ROTOR, 3024
  - ROTORAX, 3026
  - ROTORG, 3028
  - ROTORSE, 3030
  - RR0D, 3031
  - RSPINR, 3033
  - RSPINT, 3041
  - RSPLINE, 3045
  - RSSCON, 3047
  - RTRPLT, 3050
  - RTRPLT1, 3052
  - RVDOF, 3054
  - RVDOF1, 3055
  - SANGLE, 3056
  - SEBNDRY, 3057
  - SEBSET, 3058
  - SEBSET1, 3059
  - SEBULK, 3061
  - SECONCT, 3064
  - SECSET, 3066
  - SECSET1, 3067
  - SECTAX, 3069
  - SEDLINK, 3070
  - SEDRSP2, 3072
  - SEDRSP3, 3075
  - SEELT, 3078
  - SEEXCLD, 3079
  - SELABEL, 3080
  - SELOAD, 3081
  - SELOC, 3083
  - SEMPLN, 3085
  - SENQSET, 3086
  - SEQGP, 3087
  - SEQSEP, 3088
  - SEQSET, 3090
  - SEQSET1, 3092
  - SESET, 3094
  - SESUP, 3095
  - SET1, 3096
  - SET2, 3097
- , 3172
  - RINGAX, 3011
  - RINGFL, 3013
  - RJOINT, 3014
  - RLOAD1, 3015
  - RLOAD2, 3017
  - ROTBENT, 3019
  - SET3, 3098
  - SET4 , 3100
  - SETREE, 3101
  - SEUSET, 3103
  - SEUSET1, 3104
  - SHREL, 3105
  - SHRPOL, 3106
  - SHRUDS, 3107
  - SLBDY, 3108
  - SLOAD, 3109
  - SLOADN1, 3110
  - SNORM, 3111
  - SPBLND1, 3113
  - SPBLND2, 3115
  - SPC, 3116
  - SPC1, 3118
  - SPCADD, 3120
  - SPCAX, 3121
  - SPCD, 3122
  - SPCOFF, 3125
  - SPCOFF1, 3126
  - SPCR, 3127
  - SPHERE, 3129
  - SPLINE1, 3130
  - SPLINE2, 3132
  - SPLINE3, 3134
  - SPLINE4, 3136
  - SPLINE5, 3138
  - SPLINE6, 3140
  - SPLINE7, 3143
  - SPLINEX, 3145
  - SPLINRB, 3147
  - SPOINT, 3148
  - SPRELAX, 3149
  - STOCHAS, 3150
  - SUPAX, 3152
  - SUPORT, 3153
  - SUPORT1, 3155
  - SUPORT6, 3157
  - SURFINI, 3159
  - SWLDPRM, 3161

- TABD1MD, 3172
- TABDMP1, 3174
- TABL3D, 3177
- TABLE3D, 3182
- TABLED1, 3183
- TABLED2, 3186
- TABLED3, 3188
- TABLED4, 3190
- TABLED5, 3191
- TABLEHT, 3195
- TABLEH1, 3196
- TABLEL1, 3197
- TABLEM1, 3198
- TABLEM2, 3200
- TABLEM3, 3202
- TABLEM4, 3204
- TABLES1, 3205
- TABLEST, 3207
- TABLFTG, 3208
- TABLRPC, 3209
- TABLUDS, 3211
- TABRND1, 3212
- TABRNDG, 3215
- TABSCTL, 3216
- TEMP, 3218
- TEMPAX, 3220
- TEMPB3, 3221
- TEMPBC, 3223
- TEMPD, 3226
- TEMPN1, 3228
- TEMPP1, 3229
- TEMPP3, 3232
- TEMPRB, 3233
- TERMIN, 3236
- TF, 3238
- THPAD, 3240
- TIC, 3246
- TICEL, 3247
- TICEUL1, 3249
- TICEUDS, 3250
- TICREG, 3251
- TICVAL, 3253
- TIC3, 3255
- TIM2PSD, 3257
- TIRE, 3260
- TLOAD1, 3261
- TLOAD2, 3264
- TMPSET, 3268
- TODYNA, 3269
- TOMVAR, 3270
- TOPVAR, 3273
- TRIM, 3277
- TRIM2, 3278
- TRMCPL, 3280
- TSTEP, 3282
- TSTEPNL, 3283
- TTEMP, 3290
- UDNAME, 3291
- UDSESV, 3292
- UNBALNC, 3293
- UNGLUE, 3297
- UNGLUE, 3298
- USET, 3300
- USET1, 3301
- USRSUB6, 3303
- UXVEC, 3304
- VCCT, 3306
- VCCT, 3308
- VIEW, 3311
- VIEW3D, 3313
- VIEWEX, 3315
- WALL, 3320
- WEAR, 3322
- WETELME, 3324
- WETELMG, 3325
- WETLOAD, 3326
- WETSURF, 3328
- YLDHY, 3329
- YLDJC, 3330
- YLDMC, 3332
- YLDMSS, 3333
- YLDPOL, 3335
- YLDRPL, 3336
- YLD SG, 3337
- YLDTM, 3339
- YLDUDS, 3341
- YLDVM, 3343
- YLDZA, 3346

## A

Configuring the Runtime Environment

- Specifying Parameters, 3350
  - Command Initialization and Runtime Configuration Files, 3350
  - Environment Variables, 3354



- User-Defined Keywords, 3355
  - General Keywords, 3355
  - PARAM Keywords, 3356
  - Value Descriptors, 3357
  - Examples:, 3358
- Resolving Duplicate Parameter Specifications, 3360
- Customizing Command Initialization and Runtime Configuration Files, 3363
  - Examples, 3365
- Symbolic Substitution, 3369
  - Introduction, 3369
  - Simple Examples, 3369
  - Detailed Specifications, 3371
  - Examples, 3380





# Preface

- List of MSC Nastran Guides xx
- Technical Support xxi
- Training and Internet Resources xxi

## List of MSC Nastran Guides

A list of some of the MSC Nastran guides is as follows:

Installation and Release Guides
■ Installation and Operations Guide
■ Release Guide
Reference Guides
■ Quick Reference Guide
■ DMAP Programmer's Guide
■ Reference Guide
■ Utilities Guide
Demonstration Guides
■ Linear Analysis
■ Implicit Nonlinear (SOL 400)
■ Explicit Nonlinear (SOL 700)
■ MSC Nastran Verification Guide
User's Guides
■ Getting Started
■ Linear Static Analysis
■ Dynamic Analysis
■ Embedded Fatigue
■ Embedded Vibration Fatigue
■ Thermal Analysis
■ Superelements and Modules
■ Design Sensitivity and Optimization
■ Rotordynamics
■ Implicit Nonlinear (SOL 400)
■ Explicit Nonlinear (SOL 700)
■ Aeroelastic Analysis
■ User Defined Services
■ Non Linear (SOL 600)
■ High Performance Computing
■ DEMATD

You may find any of these documents from Hexagon at:

<https://simcompanion.hexagon.com/customers/s/article/MSC-Nastran-Support-Home-Page>

## Technical Support

For technical support phone numbers and contact information, please visit:

<https://simcompanion.hexagon.com/customers/s/article/support-contact-information-kb8019304>

**Support Center** (<http://simcompanion.hexagon.com>)

The SimCompanion link above gives you access to the wealth of resources for Hexagon products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from Hexagon, allows you to manage your support cases, and participate in our discussion forums.

## Training and Internet Resources

Hexagon corporate site has information on the latest events, products and services for the CAD/CAE/CAM marketplace.

<http://simcompanion.hexagon.com>

The SimCompanion link above gives you access to the wealth of resources for Hexagon products. Here you will find product and support contact information, product documentations, knowledge base articles, product error list, knowledge base articles and SimAcademy Webinars. It is a searchable database which allows you to find articles relevant to your inquiry. Valid MSC customer entitlement and login is required to access the database and documents. It is a single sign-on that gives you access to product documentation for complete list of products from Hexagon, allows you to manage your support cases, and participate in our discussion forums.

<http://www.mssoftware.com/msc-training>

The link above will point you to schedule and description of seminars. Following courses are recommended for beginning Nastran users.

### **NAS120 - Linear Static Analysis using MSC Nastran and Patran**

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC Nastran and Patran. MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran will be an integral part of the examples and workshops and will be used to generate and verify illustrative MSC Nastran models, manage analysis submission requests, and visualize results. This seminar provides the foundation required for intermediate and advanced MSC Nastran applications.



# 1

## Executing MSC Nastran

- Executing MSC Nastran

## Executing MSC Nastran

MSC Nastran is executed from the command line using the command `nast_ver` (where `nast_ver=nast20221`). This command sets environment variables in order for MSC Nastran to correctly execute on your system and it also has capabilities to predict memory and solvers (as of MSC Nastran 2022.1 with `solve=auto`). The basic format of the `nast_ver` command is

```
nast_ver input_data_file keywords
nast_ver input_data_file [keyword1=value1 keyword2=value2 ...]
```

where `input_data_file` is the name of the file containing the input data and `keyword=valuei` is one or more optional keyword assignment arguments. For example, to run an MSC Nastran job using the data file `example1.dat`, enter the following command:

```
nast_ver example1
```

Most keyword assignments can be specified as command line arguments and/or included in RC files. There are some exceptions which may be specified on the command line or the User RC files, but not in the system RC files (`MSC_BASE/conf/RCfile`).

There are two RC files controlled by the user:

- The user RC file is in your home (or login) directory. This file should be used to define parameters that are applied to all jobs run by the user.
  - The local RC file is `nast20221rc` on Linux and `NAST20221.rcf` on Windows, and is located in the same directory as the input data file. If the “rcf” keyword is used, this local RC file is ignored. This file should be used to define parameters that are applied to all jobs contained in the input data file directory.
1. The tilde (`-`) character is not recognized within RC files.
  2. Environment variables are only recognized when used in the context of a logical symbol (on Windows `NAST20221.rcf`) (see [Using Filenames and Logical Symbols](#) in the *MSC Nastran Installation and Operations Guide*).
  3. When a keyword is specified on the command line, embedded spaces or special characters that are significant to the shell must be properly quoted; quotes should not be used within RC files.

The keywords listed as follows are the most common for various computers, but are not available on all computers. Also, the defaults may be site-dependent. Please consult your [Keywords and Environment Variables](#) in the *MSC Nastran Installation and Operations Guide* for keyword applicability, specialized keywords, and further discussion of the keywords and their defaults. Keywords that use yes/no values accept partial specification and case-independent values. For example, “yes” may be specified as “y”, “ye”, or “yes” using uppercase or lowercase letters. The examples assume the jobs are run under a Linux operating system.

after	after = <i>time</i>	Default: <i>None</i>
(Linux)	Holds the job’s execution until the time specified by <i>time</i> . See the description of the “at” command in your system documentation for the format of <i>time</i>	
	Example:	<code>nast_ver example after=10:00</code>
	The job is held until 10:00 AM.	

**append**      `append={yes|no}`      Default: no

Combines the .f04, .f06, and .log files into a single file after the run completes. If “no” is specified, the files are not combined. If “yes” is specified, the files are combined into one file with the suffix “.out”.

Example:                      `nast_ver example append=yes`

Result: The .f04, .f06, and .log files are combined into a file named example.out.

**batch**      `batch={yes|no}`      Default: yes

(Linux)

Indicates how the job is to be run. If “yes” is specified, the job is run as a background process. If “no” is specified, the job is run in the foreground. If the “aft” or “queue” keywords are specified, the batch keyword is ignored. Jobs submitted with “batch=yes” will run under nice(1).

**Note:** If the job is already running in an NQS or NQE batch job, the default is “no”.

Example:                      `nast_ver example batch=no`

Result: The job is run in the foreground.

**bpool**      `bpool=value`      Default: *See text below.*

Specifies the number of GINO and/or executive blocks, or memory size in MB, GB, etc., that are placed in buffer pool.

The size is specified as the number of blocks (BUFFSIZE words long), a percentage of MEM, or the number of words or bytes followed by one of the modifiers: "T", "TW", "TB", "G", "GW", "GB", "M", "MW", "MB", "K", "KW", "KB", "W", "B". See Specifying Memory Sizes, 63 for a description of these modifiers.

If `mem=max` (which is the default) or/and `solve=auto` is not used, the default of `bpool` is 150 GINO blocks.

If `solve=auto` is used, `bpool` will be set automatically.

If `mem=max` is used, the default of `bpool` will be set to:

- 25% of memory for non SOL 101 or SOL 400.
- the remaining memory after memory estimate needed for the solver for SOL 101 or SOL 400.

Example:                      `nast_ver example bpool=100mb`

**buffsize**      `buffsize=value`      Default: 32769 words

Use to control the number of words per I/O record.

For large models, it is recommended to change buffsize to 65537, which is the maximum allowed.

**casi** If set to "no", this flag will disable the Casi solver as a possible option when "solve=auto" is specified.

**db**s `db=pathname` Default= . {Current directory}  
Creates database files (see [Using the NASTRAN Statement](#) in the *MSC Nastran Installation and Operations Guide*) using an alternate file prefix. If "db" is not specified, database files are created in the current directory using the basename of the input data file as the prefix. If the "db" value is a directory, database files are created in the specified directory using the basename of the input data file as the filename.

**Note:** If "db" is specified and "scratch=yes" is specified, a warning will be issued and "scratch=no" is assumed.

In the following examples, assume the current directory includes subdirectories "mydir" and "other", and that an "example.dat" exists in both the current directory and "other". That is, ./example.dat, ./mydir, ./other, and ./other/example.dat exist on Linux, and .\example.dat, .\mydir, .\other, and .\other\example.dat exist on Windows.

Example: `nast_ver example`

Result: Database files are created in the current directory with the name "example" e.g., ./example.DBALL on Linux; and .\example.DBALL on Windows.

Example: `nast_ver other/example`

Result: Database files are created in the "other" directory with the name "example", e.g., ./other/example.DBALL on Linux, and .\other\example.DBALL on Windows.

Example: `nast_ver example db=myfile`

Result: Database files are created in the current directory with the name "myfile", e.g., ./myfile.DBALL on Linux, and .\myfile.DBALL on Windows.

Example: `nast_ver example db=mydir`

Result: Database files are created in the mydir directory with the name "example", e.g., ./mydir/example.DBALL on Linux, and .\mydir\example.DBALL on Windows.

Example: `nast_ver example db=mydir/myfile`

Result: Database files are created in the mydir directory with the name "myfile", e.g., ./mydir/myfile.DBALL on Linux, and .\mydir\myfile.DBALL on Windows

Example: `nast_ver example dmp=4 host=a:b:c:d  
db=/aa:/bb:/cc:/dd`

Result: This example will set the "db" directory to "/aa" on host a, "/bb" on host b, "/cc" on host c, and finally "/dd" on host d.

**Note:** The use of distinct per-task database directories can have a significant impact on elapsed time performance of DMP jobs on SMP and NUMA systems.



dmp                    dmp=*dmparallel*                    Default: No parallelism based upon dividing the model

The dmp keyword specifies the number of processors used to efficiently obtain a solution to the simulation. These processors may be on the same node or on multiple nodes specified by the HOST command line option.

Specifying dmparallel > 1 and not including a DOMAINSOLVER Executive Command Statement has the following effects.

- The extraction of real eigenvalues using the default Lanczos method will be performed in parallel in SOL 103 and 111.
- The frequency response using SOL 111 will be performed in parallel.

Specifying dmp > 1 and using the DOMAINSOLVER provides the following addition user controlled performance gains using parallel processing

- The extraction of eigenvalues may be performed using Automated Component Modal Synthesis (ACMS) which is an approximate method that is very computational efficient when a large number of modes (high frequencies) are required.
- Design sensitivity may be performed in parallel in SOL 200
- The formation of using advanced nonlinear elements stiffness matrices may be performed in parallel for SOL 400.

It may also be necessary to define a host file if a network based distributed computing is used. See the *MSC Nastran Installation and Operators Guide*.

gpuid

gpuid=id,id or gpuid=id:id                   Default: none

id: the ID of a licensed GPU device to be used in the analysis.

GPUs are used for two types of operations: matrix factorization and matrix multiplication. In the routines that use GPUs for matrix factorization, only one GPU will be used per DMP process. So in order to use multiple GPUs, the user must also use multiple DMP processes:

```
nast_ver job gpuid=0,1,...,ngpu-1 dmp=ndmp, where ndmp≥ngpu.
```

Each DMP process will be assigned a GPU ID in round robin fashion.

In MUMPS, MPYAD and FASTFR modules, on the other hand, multiple GPUs can be used for matrix multiplication by a single process, as long as the number of SMPthreads is equal to or larger than the number of GPUs. These modules also use multiple parallel streams for data transfer between the GPU and the host system, so it is recommended to use the maximum number of available CPU cores/threads because that helps reduce the overhead of data transfer between the host and the GPUs.

So, for example, in order to achieve the best performance on a system with 20 CPU cores and 2 GPUs the user is advised to run Nastran with:

```
nast_ver job dmp=2 smp=10 gpuid=0,1
```

or

```
nast_ver job dmp=4 smp=5 gpuid=0,1.
```

Only up to 8 CUDA-compatible GPUs can be supported.

gpu\_min\_rank

gpu\_min\_rank=value                   Default: 32

The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via `gpu_min_front` and its companion parameter, `gpu_min_rank`. The value specified must be an integer greater or equal to 1. If the rank of the frontal matrix is smaller than `value`, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. This keyword may also be set via SYSTEM cell 655.

gpu_min_front	gpu_min_front=value	Default: 2048
	The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via <code>gpu_min_front</code> and its companion parameter, <code>gpu_min_rank</code> . The value specified must be an integer greater or equal to 1. If the front size is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. This keyword may also be set via SYSTEM cell 656.	
memorymax	memorymax= <i>maximum user memory</i>	Default: 0.5 x the amount of physical memory on your machine
	The user can enter the amount of memory as <i>n</i> GB or as <i>f</i> xphysical; <i>f</i> is the fraction between 0 and 1.	
	When <code>dmp</code> is used, the amount of memory per processor is <code>memorymax / dmp</code> .	
	If the machine is going to be used by a single user or a single job, it is recommended to set <code>memorymax</code> to <code>0.75xPhysical</code> .	
	If multiple Nastran jobs are running simultaneously on the same node of a computer system, it is recommended to set <code>memorymax</code> to <code>0.8*physical_memory</code> divided by the number of possible Nastran jobs running on the same node. Otherwise, one might encounter an error due to lack of memory on small models. The error can be avoided by using above setting in the system RC file or <code>mem=size</code> specified properly for the job in the command line.	
memory	memory= <i>memory_size</i>	Default=max
	Specifies the amount of memory to allocate. The <i>memory_size</i> can be specified either as a number of words, or as a number followed by one of the following modifiers:	
	G or Gw	Multiply <i>memory_size</i> by $1024^{**3}$ .
	Gb	Multiply <i>memory_size</i> by $(1024^{**3})/bytes\_per\_word$ .
	where <i>bytes_per_word</i> is 8 or you can enter a fraction of the physical memory	
	If "memory=estimate" is specified, ESTIMATE will be used to determine size. If "memory=max" is specified, then the amount of memory allocated depends on the "memorymaximum" keyword.	
	Example:	<code>nast_ver example memory=5gb</code>
	Result: The job is run using a memory size of 5 gigabytes.	
	Example:	<code>nast_ver example memory=0.5xPhysical</code>
	Result: The job is run using one-half the amount of physical RAM.	
	For a more detailed description, see the <a href="#">memory</a> in the <i>MSC Nastran Installation and Operations Guide</i> .	

**news**                    news={yes|no|auto}                    Default=yes

Displays the news file (*install\_dir*/msc20221/nast/news.txt on Linux and *install\_dir*\msc20221\nast\news.txt on Windows) in the .f06 file. If “auto” is specified, the news file is only displayed if it has been modified since the last time it was displayed for you. If “yes” is specified, the news file is displayed in the .f06 file regardless of when it was last changed. If “no” is specified, the news file is not displayed in the .f06 file.

Example:                                    `nast_ver example news=yes`

Result: The news file is displayed in the .f06 file after the title page block.

**notify**                    notify={yes|no}                    Default=yes

Sends notification when the job is completed. See the “ncmd” keyword to define an alternate notification command.

**Note:** If the job is queued using the queue keyword, or the job is already running in an NQS batch job, the default is “no”.

Example:                                    `nast_ver example notify=yes`

**old**                        old={yes|no}                        Default=yes

Saves previous copies of the .f04, .f06, .log, .op2, .out, pch, and .plt output files using sequence numbers (additional user-specified file types can be versioned with the “oldtypes” keyword). Sequence numbers are appended to the keyword filenames and are separated by a period.

If “yes” is specified, the highest sequence number of each of the output files is determined. The highest sequence number found is incremented by one to become the new sequence number. Then, all current output files that do not include sequence numbers are renamed using the new sequence number as a type.

Example:                                    `nast_ver example old=yes`

For example, assume your current working directory contains the following files:

```
v2401.dat v2401.f04.1 v2401.f06 v2401.log v2401.log.1
v2401.f04 v2401.f04.2 v2401.f06.1 v2401.log.1 v2401.log.3
```

Apparently, the user ran the job four times, but deleted some of the files; e.g.

```
v2401.f04.3
v2401.f06.3
```

When the job is run again with “old=yes”, the files are renamed as follows:

```
v2401.f04 is renamed to v2401.f04.4
v2401.f06 is renamed to v2401.f06.4
v2401.log is renamed to v2401.log.4.
```

The sequence number 4 is used because it is one greater than the highest sequence number of all of the selected files (the highest being v2401.log.3).

out                    *out=pathname*                    Default= .

Saves the output files using a different file prefix or in a different directory. If “out” is not specified, the output files are saved in the current directory using the basename of the input data file as a prefix. If the “out” value is a directory, output files are created in the specified directory using the basename of the input data file as the filename.

In the following examples, assume the current directory includes subdirectories “mydir” and “other”, and that an “example.dat” exists in both the current directory and “other”. That is, `./example.dat`, `./mydir`, `./other`, and `./other/example.dat` exist on Linux; and `.\example.dat`, `.\mydir`, `.\other`, and `.\other\example.dat` exist on Windows.

Example:                    `nast_ver example`

or:                    `nast_ver other/example`

Result: Output files are created in the current directory with the name “example”, e.g., `./example.f06` on Linux and `.\example.f06` on Windows.

Example:                    `nast_ver example out=myfile`

Result: Output files are created in the current directory with the name “myfile”, e.g., `./myfile.f06` on Linux and `.\myfile.f06` on Windows.

Example:                    `nast_ver example out=mydir`

Result: Output files are created in the mydir directory with the name “example”, e.g., `./mydir/example.f06` on Linux and `.\mydir\example.f06` on Windows.

Example:                    `nast_ver example out=mydir/myfile`

Result: Output files are created in the mydir directory with the name “myfile”, e.g., `./mydir/myfile.f06` on Linux and `.\mydir\myfile.f06` on Windows.

rcf	<p>rcf=pathname                      Default=no</p> <p>Specifies the name of the local RC file. If this keyword is not specified, the nast20221rc file on Linux and NAST20221.rcf on Windows located in the input data file's directory is used.</p> <p>Example:                              <code>nast_ver example rcf=nast.rc</code></p> <p>Result: The nastran command will process <code>./nast.rcf</code> on Linux, or <code>.\nast.rcf</code> on Windows in lieu of the default local RC file <code>./nast20221rc</code> on Linux, and <code>.\NAST20221.rcf</code> on Windows.</p>
scratch	<p>scratch={yes no mini post}      Default=no</p> <p>Deletes the database files at the end of the run. If the database files are not required, "scratch=yes" can be used to remove them, thereby preventing cluttering of the directory with unwanted files. If "mini" is specified, a reduced size database (that can only be used for data recovery restarts) will be created. See <a href="#">Database Concepts</a> in the <i>MSC Nastran Reference Guide</i> for further details on the "mini" database. If scratch=post is specified, a reduced size database intended for use by Patran or the toolkit will be created. Scratch=post also performs the actions of NASTRAN INDEX=19. But with scratch=post the resulting database is not restartable.</p> <p>Example:                              <code>nast_ver example scratch=yes</code></p> <p>All database files created by the run are deleted at the end of the job in the same way as the FMS statement INIT MASTER(S).</p>
sdirectory	<p>sdirectory=directory              Default: <i>See the description below.</i></p> <p>See <a href="#">Determining Resource Requirements</a> in the <i>MSC Nastran Installation and Operations Guide</i> for information on estimating a job's total disk space requirements.</p> <p>Specifies the directory to use for temporary scratch files created during the run. MSC Nastran can create very large scratch files; therefore, the scratch directory should contain sufficient space to store any scratch files created during a run. You must have read, write, and execute privileges to the directory.</p> <p>Linux: The default value is taken from the TMPDIR environment variable if it is set to a nonnull value. Otherwise, the computer's default temporary file directory is chosen; this is usually /tmp.</p> <p>Windows: The default value is taken from the TEMP environment variable.</p> <p>Linux Example:                      <code>nast_ver example sdir=/scratch</code></p> <p>Result: Scratch files are created in the directory /scratch.</p>

smp	smp= <i>value</i>	Default = 0
	Specifies the maximum number of processors selected for shared-memory parallel (SMP) processing in several numeric modules. In some cases SMP parallelization can be combined with DMP. In such cases, the smp keyword specifies the number of threads per process used in the simulation.	
	smp parallelization is available for mathematical kernels.	
	smp parallelization is available for the Intel MKL PARDISO solver in SOL 101, 107, 108, 111, 200, and 400; see the SPARSE SOLVER section of this document.	
	smp parallelization is available for the NLEMG modules and is disabled in the following cases:	
	<ul style="list-style-type: none"> <li>■ heat transfer and coupled analysis simulations;</li> <li>■ models that have contact between Advanced elements and Nastran elements;</li> <li>■ models that have QUADR/QUAD4 or TRIAR/TRIA3 element types arranged into the same element groups.</li> <li>■ models that have friction.</li> </ul>	
	If models have contact between Nastran and Advanced elements resulting in a SYSTEM FATAL MESSAGE (SFM) with smp > 1, then rerun the job with sys107=65536+ <i>number</i> . For example, if smp=8 is desired but an SFM results, then rerun with sys107=65544 in place of smp=8. This turns off the SMP parallelization for NLEMG.	
solve	solve={auto train}	Default (not set)
	If solve=auto then Nastran will automatically select the solver/parallel/memory. If solve=train then Pardiso memory coefficients will be updated. These options may be placed on the command line or in the User's RC files.	
spotweld	<path>/spotweld.sys	Default: <i>None</i> .
	Points to the location of the spotweld.sys file used in fatigue analysis of spotwelds to look up the diameter of the spotwelds based on the thickness of the top/bottom sheets. See the definition of PFTG bulk data for more information.	
symbol	symbol= <i>name=string</i>	Default: <i>None</i>
	Defines a symbolic (or logical) name used in ASSIGN and INCLUDE statements and in command line arguments. This keyword may be specified in initialization or RC files and on the command line. The symbol definition may include references to previously defined symbols or environment variables using the standard "\$name" or "\${name}" syntax on Linux or %name% syntax on Windows. For convenience, the character separating the "symbol" and "name" specification and the "name" and "string" specification may be either an equal sign ("=") or a hash mark ("#"). The use of a hash mark allows this keyword to be specified as an argument to a Windows .bat file.	

If "node" is specified, symbolic names defined using this keyword are not used on the local system. Instead the specified values are passed to the remote system. This means that any pathnames must be valid on the remote system. Use the "lsymbol" keyword to specify symbolic names for the local system.

If "node" is not specified, symbolic names defined using the "lsymbol" keyword are processed as if they were defined using the "symbol" keyword.

Symbolic names are processed in the order they are encountered while processing the initialization and RC files and the command line. If a duplicate symbolic name is encountered, the new value replaces the previously specified value.

Symbolic names must be 16 characters or less. The value assigned to the symbolic name must be 256 characters or less. If the symbolic name used in an ASSIGN or INCLUDE statement or in command line arguments is not defined, it is left in the filename specification as is.

For example, many of the .tpl and .demo input data files have ASSIGN statements such as the following:

```
ASSIGN 'MASTER=DBSDIR:abc.master'
```

The string "DBSDIR:" specifies a symbolic name that is to be replaced by another string. The replaced string is defined by the "symbol=" keyword (or "lsymbol=" keyword if "node" was not specified) in an initialization or RC file, on the command line, or as environment variable. For example,

(Linux) `symbol=DBSDIR=/dbs`

(Windows) `symbol=DBSDIR=d:\dbs`

When the previous ASSIGN statement is processed, the filename assigned to the logical name MASTER is /dbs/abc.master on Linux and d:\dbs\abc.master on Windows. An alternate way of defining symbolic names is through the use of environment variables. For example, typing the following command

```
export DBSDIR=/dbs
```

at a Korn shell prompt, or

```
setenv DBSDIR /dbs
```

at a C-shell prompt, or

at a Windows shell prompt, is equivalent to the "symbol" keyword definition.

**Note:** If a symbolic name is defined by both a symbol statement in an RC file and by an environment variable, the symbol statement value will be used.

The section titled [Environment Variables](#) in the *MSC Nastran Installation and Operations Guide* contains a list of environment variables that are automatically created by the nastran command. Of particular interest to the logical symbol feature are the OUTDIR and DBSDIR variables. These variables refer to the directory that will contain the output files (set using the "out" keyword) and the directory that will contain the permanent database files (set using the "dbs" keyword), respectively.



uds	uds={filename   model}	Default: None
	<p>The file name is the source file that defines user subroutines. This keyword is used to build user service at nastran job submittal time. The user subroutines implementations are from the specified file and the user service name is from the connect service statement in input file.</p> <p>On windows the UDS files being created may not be in "C:\Program Files". Please copy files to a writable directory to build.</p> <p>Example:</p> <p>If user routine is written in Fortran: <code>nast_ver example uds=mysource.F</code></p> <p>If user routine is written in C++: <code>nast_ver example uds=mysource.cpp</code></p> <p>The mysource.F includes all user defined subroutines. There may be multiple user subroutines from multiple interfaces in the file. The user subroutine names are predefined and documented in User Defined Services.</p> <p>Another option to define user subroutines is to put them in BEGIN BULK UDS section in the input file. The content in this section will be used to create source file when building user service. If this option is used, the uds keyword should be set to model.</p> <p>Example: <code>nast_ver example uds=model</code></p>	
udssave	udssave=pathname	Default: output directory
	<p>The udssave is the location to build user service. If udssave is given, the path will be used to build and save user service. If it is not given, the nastran output directory will be used and the user service will be removed after nastran job run.</p> <p>Example: <code>nast_ver example uds=mysource.F udssave=/scratch/mydir</code></p>	
xdbold	xdbold={yes no}	Default=yes
	<p>Keeps or deletes the previous copy of the .xdb.</p> <p>If "yes" is specified, the previous copy is kept and appended by the current run.</p> <p>If "no" is specified, the previous copy is deleted and a new file is created. This gives the same behavior as below FMS statement:</p> <p>ASSIGN dbc=&lt;file_name&gt;.xdb delete</p> <p>Example: <code>nast_ver example xdbold=no</code></p>	



# 2

## NASTRAN Statement

- The NASTRAN Statement

## The NASTRAN Statement

The NASTRAN statement is used to specify values for certain Executive System operational parameters. These parameters are also called system cells. The NASTRAN statement is used for exceptional circumstances and is therefore not needed in most runs. The NASTRAN statement may also be specified in the runtime configuration (RC) files at the system, user, and job level as described in the .

## NASTRAN Executive System Parameter Modification

Specifies values for certain Executive System operational parameters called system cells.

### Format:

```
NASTRAN cellnamei=expressioni, ..., cellnamen=expressionn
```

or

```
NASTRAN SYSTEM(i)=expressioni, ..., SYSTEM(n)=expressionn
```

Descriptor	Meaning
cellnamei	System cell names from <a href="#">Table 2-1</a> .
SYSTEM	Specifies the system cell number.
expression	See DEFINE statement for description.
i	System cell number from <a href="#">Table 2-1</a> or from the SYSTEM common block described in the <i>MSC Nastran User Modifiable User's Guide</i> .

### Remarks:

1. The NASTRAN statements may appear anywhere in the File Management Section. The NASTRAN statement may also be specified in runtime configuration (RC) files. See [Customizing Command Initialization and Runtime Configuration Files](#) (App. A) in the *MSC Nastran Installation and Operations Guide*.
2. System cell values and their associated cell names may also be set with the DEFINE statement. They may also be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See [PUTSYS](#), [GETSYS](#) in the *DMAP Programmer's Guide*.
3. More than one NASTRAN statement and/or DEFINE statement may be present and, if a system cell is specified more than once among these statements, then the last specification takes precedence.
4. The expression will use type conversion rules based on the type (i.e., integer, real, or logical) of the cellname, as defined on a previous DEFINE statement (see the DEFINE statement for conversion rules).
5. If expression is omitted, the system cell associated with the cellname will be assigned the value as set on a previous DEFINE statement.

### Examples:

1. Either of the following statements could be used to change the default value for block size.

```
NASTRAN SYSTEM (1) = 4097
```

or

```
NASTRAN BUFFSIZE = 4097
```

or, if a prior DEFINE statement had defined a keyword MY\_SYSBUF to the value 4097, then the following code could be used:

NASTRAN SYSTEM(1)=MY\_SYSBUF

or

NASTRAN BUFFSIZE=MY\_SYSBUF

2. [Table 2-1](#) gives a summary of the recommended system cells. System Cells

Table 2-1 System Cell Summary

System Cell Name (Number)	Function and Reference
<a href="#">BUFFSIZE (1)</a>	Specifies the number of words in a physical record. Also called block length.
<a href="#">F06 (2)</a>	Specifies FORTRAN unit number for standard output file (.f06). (Integer $\geq$ 0; a value of 0 sends the results to the log file; Default=6). This system cell may not be set for SOL 600 or SOL 700.
<a href="#">NLINES (9)</a>	Specifies the number of lines printed per page of output. <a href="#">LINE (Case)</a> .
<a href="#">MAXLINES (14)</a>	<a href="#">MAXLINES (Case)</a>
<a href="#">METIME (20)</a>	Minimum time for execution summary table message. <a href="#">Output Description</a> in the <i>MSC Nastran Reference Guide</i> .
<a href="#">APP (21)</a>	Approach Flag. See the <a href="#">APP</a> Executive Control statement. If APP HEAT is specified, then this system cell is set to 1.
<a href="#">MACHTYPE (22)</a>	Machine type
<a href="#">DIAGA (25)</a>	Alternate method to set DIAGs 1 through 32. <a href="#">DIAG</a> .
<a href="#">CONFIG (28)</a>	Machine subtype
<a href="#">ADUMi (46 - 54)</a>	Dummy element flag, i=1 through 9.
<a href="#">HEAT (56)</a>	<a href="#">APP</a>
	0 Structural analysis (Default).
	1 Heat transfer
<a href="#">DIAGB (61)</a>	Alternate method to set diagnostics 33 through 64. <a href="#">DIAG</a> .
<a href="#">PUNCH (64)</a>	Specifies FORTRAN unit number for PUNCH file (.f07). (Default=7)
<a href="#">MPYAD (66)</a>	Selects/deselects multiplication methods. <a href="#">MPYAD</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> .
<a href="#">DCMPOUT(69)</a>	Controls matrix decomposition for MSCLDL and MSCLU. Same as <a href="#">DECOMP</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> and the <a href="#">Option Selection</a> in the <i>MSC Nastran Numerical Methods User's Guide</i> .
	0 or -1 Print up to 50 messages for null columns and zero diagonals (Default=-1).
	1 Terminates execution when first null column is encountered.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	2	Suppress printing of message when a null column is encountered.
	4	Terminates execution when first zero diagonal term is encountered.
	16	Place 1.0 in diagonal position for all null columns and proceed with the decomposition.
	32	Terminates execution on zero diagonal term.
	64	Exit after execution of preface for symmetric decomposition.
	256	Print MAXRATIO messages (SIM 4159) from DCMF.
DELFF (77)	Deletes form feeds.	
DBSET	Database neutral file set. <a href="#">SubDMAP DBFETCH</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> .	
DMAP (82)	Allows NOGO to operate. See Processing of User Errors in the <i>MSC Nastran DMAP Programmer's Guide</i> .	
F04 (86)	Specifies FORTRAN unit number for Execution Summary Table (.f04). <a href="#">Output Description</a> in the <i>MSC Nastran Reference Guide</i> (Integer $\geq 0$ ; a value of 0 sends the results to the log file; Default = 4).	
RADMTX (87)	Type of radiation exchange coefficients, <a href="#">RADMTX</a> .	
	1	Direct input of a symmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the symmetry, only one-half of the RADMTX may be entered.
	2	Direct input of a unsymmetric SCRIPT-AF matrix on RADMTX and RADLST entries is allowed. Due to the unsymmetry, the full matrix must be specified on the RADMTX entries.
	3	If you are running a view factor calculation in an opened enclosure, NASTRAN assumes that the radiation will be lost to space at absolute zero degrees Kelvin. You can set SYSTEM(87)=3 so that radiation will not be lost to space.
RADLST (88)	Print radiation area summary. <a href="#">RADLST</a> .	
SMP (107)	Number of processors used for Shared Memory Parallel	
NEWHESS (108)	Request complex eigenvalue method. See the <a href="#">EIGC</a> entry, <i>MSC Nastran Numerical Methods User's Guide</i> .	
(109)	Controls DMAP execution:	
	0	Do not execute DMAP instruction if all outputs are previously computed.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	1	Always execute DMAP instruction (Default).
BUFFPOOL (114)	Bufferpool size. <a href="#">Keywords and Environment Variables</a> in the <i>MSC Nastran Installation and Operations Guide</i> .	
(119)	Controls the type of bufferpool method to be used:	
	4	Use Version 2012.2 bufferpool method. Default = 4.
	Not 4	Use Version 2012.1 bufferpool method.
ATTDEL (124)	Controls the automatic assignment of the delivery database. <a href="#">Database Concepts</a> in the <i>MSC Nastran Reference Guide</i> . See also <a href="#">Creating and Attaching Alternate Delivery Databases</a> in the <i>MSC Nastran Installation and Operations Guide</i> .	
	0	Enables automatic assigning (Default).
	-1	Disables automatic assigning.
NOKEEP (125)	Controls NOKEEP option of the RESTART File Management statement.	
	0	Disable NOKEEP (Default)
	1	Enable NOKEEP
SPARSE (126)	Sparse matrix method selection for MSCLDL and MSCLU. For unsymmetric sparse matrix decomposition method selection, see cell number 209. The following values may be summed in order to select sparse matrix methods in the operations listed below:	
	0	Deactivate sparse methods
	1	Multiplication
	8	Symmetric decomposition
	16	Forward-backward substitution
	The default is 25, which is the sum of all values.	
UPDTTIM (128)	Specifies database directory update time. <a href="#">DBUPDATE, 90</a> FMS statement.	
	0	Do not update.
	>0	Time, in minutes, between database directory updates. (Default 5 min.)
SMPYAD67 (129)	Select pre-Version 67 method in the SMPYAD module. <a href="#">SMPYAD</a> in the <i>MSC Nastran DMAP Programmer's Guide</i> .	
	0	Use current method (Default).
	1	Use pre-Version 67 method.



Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
MAXDBSET	The maximum number of online DBsets attached to the run.
AUTOASGN (133)	Controls autoassigning of dbsets. Sum the desired values. (Default=7). <a href="#">Database Concepts</a> in the <i>MSC Nastran Reference Guide</i> .
	0 No databases are automatically assigned.
	1 Only the primary database is automatically assigned.
	2 Only the delivery database is automatically assigned.
	4 Only located databases are automatically assigned.
TSTAMP (135)	Controls timestamp checking of DBsets.
	0 Do not check
	1 Check (Default)
	2 Same as 1 and print diagnostics
QUADINT (141)	Specifies quadratic or linear interpolation for the line search method in nonlinear analysis (SOL 106, SOL 129, and SOL 153).
	0 Quadratic interpolation (Default)
	1 Linear interpolation
SCR300 (142)	Requests creation of SCR300 partition on SCRATCH DBset. <a href="#">INIT</a> FMS statement.
	1 Do not create SCR300 partition.
	2 Create SCR300 partition (Default).
LOCBULK (143)	LOCBULK=1 or 2 specifies that Bulk Data is being obtained via the DBLOCATE FMS statement. NASTRAN LOCBULK=2 is specified when no Bulk Data entries, except for PARAM entries, are to be deleted or added. All PARAM entries must be respecified. All other entries will be ignored and, if present, may increase CPU times in XSORT and IFP. With LOCBULK=2, the XSORT and IFP modules will not reprocess the Bulk Data Section stored in the SEMAP run. Also, GP1, TASN2, SEP1 and SEP1X modules will be skipped. 0 is the default, which assumes the RESTART FMS statement. This system cell is recognized only in SOLs 101 through 200.
(144)	RESTART FMS statement existence flag. Set to 1 if RESTART statement is present.
BFGS (145)	Selects strategies of BFGS updates for the arc-length methods in nonlinear analysis. Please see the <i>MSC Nastran Nonlinear Handbook</i> .
	0 Update $\Delta u_R$ and $\Delta u_p$ at every iteration with $\gamma^*$ . (Default)
	1 Update $\Delta u_R$ only with $\gamma$

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
	2 Update $\Delta u_R$ only with $\gamma^*$ .
FBSMEM (146)	Reserves part of Nastran open core memory for faster solution in the Lanczos method of eigenvalue extraction. Default = 0. When FBSMEM is less than or equal to 0, available memory is used to store the factor matrix for use during forward-backward substitution (FBS) operations (aka “factor caching”). System Information Message 4199 is printed in the F04 file when the factor matrix is cached. Set FBSMEM=1 to disable Lanczos factor caching.
UWM (147)	SYSTEM(147)=1 issues a User Warning Message for a DMAP parameter appearing on a CALL statement that has an inconsistent authorization in the called subDMAP. 0 is the default, which means no message is issued. A value of 1 also issues User Warning Message 1 during DMAP compilation if a module instruction is missing trailing commas for input and output data blocks.
DBVERCHK (148)	In general, databases are not compatible between major releases; therefore, a check is performed in MSC Nastran to ensure that the major version which created the database is the same as that being executed. Since specific data on the database may be compatible, SYSTEM(148) allows this check to be circumvented. However, circumventing the check may lead to problems later in the run.
	0 Check is performed (Default)
	1 Check is not performed
SCR300DEL (150)	Sets minimum number of blocks of SCR300 partition of SCRATCH DBset at which it is deleted. INIT FMS statement (Default = 100).
(151)	Requests spill or no spill of the SCR300 partition of SCRATCH DBset. INIT FMS statement (Default = 0).
DBLAMKD (155)	Differential stiffness formulation for CBEAM and CTETRA elements.
	0 Current formulation (Default)
	1 Pre-Version 67 formulation

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference		
(162)	<p>Reserves MSC Nastran executive first number for GINO for each of the tables DBTEMP, FCB, DBDICT and FIST. Default value is 6000 and is often controlled in DMAP just before and just after modules that deal with large input families. Usually a user will have no need to modify. However if the user gets the following message:</p> <p>*** SYSTEM FATAL MESSAGE 1267 (XSTORE)            NO MORE MEMORY SPACE AVAILABLE FOR EXECUTIVE TABLES.            USER ACTION: INCREASE DEFAULT NUMBER OF EXECUTIVE TABLES ENTRIES VIA SYSTEM CELL 162.  <b>SYSTEM(162) IS CURRENTLY SET TO 6000</b></p> <p>A DMAP workaround is of the form:            Just before the offending module            NP=GETSYS(NP,162) \$            NP1= value \$ value &gt; 6000            PUTSYS(NP1,162) \$            Just after offending module            PUTSYS(NP,162) \$</p>		
(166)	<p>Controls sparse symmetric decomposition. Sum the desired values (Default = 0).</p>		
	<table border="1"> <tr> <td data-bbox="601 998 689 1038">0</td> <td data-bbox="689 998 1325 1038">No action</td> </tr> </table>	0	No action
0	No action		
	<table border="1"> <tr> <td data-bbox="601 1038 689 1112">1</td> <td data-bbox="689 1038 1325 1112">If insufficient core is encountered, then switch to conventional decomposition and continue (Default).</td> </tr> </table>	1	If insufficient core is encountered, then switch to conventional decomposition and continue (Default).
1	If insufficient core is encountered, then switch to conventional decomposition and continue (Default).		
	<table border="1"> <tr> <td data-bbox="601 1112 689 1159">2</td> <td data-bbox="689 1112 1325 1159">Print diagnostics.</td> </tr> </table>	2	Print diagnostics.
2	Print diagnostics.		
	<table border="1"> <tr> <td data-bbox="601 1159 689 1293">4</td> <td data-bbox="689 1159 1325 1293">Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.</td> </tr> </table>	4	Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.
4	Do not issue fatal message if maximum ratios are exceeded. Although high maximum ratios may be printed, they will not cause job termination. This applies to the DCMP, DECOMP, REIGL, and LANCZOS modules.		
	<table border="1"> <tr> <td data-bbox="601 1293 689 1367">8</td> <td data-bbox="689 1293 1325 1367">Output a matrix containing the maximum ratio vector in the output slot for the upper factor.</td> </tr> </table>	8	Output a matrix containing the maximum ratio vector in the output slot for the upper factor.
8	Output a matrix containing the maximum ratio vector in the output slot for the upper factor.		
	<table border="1"> <tr> <td data-bbox="601 1367 689 1414">32</td> <td data-bbox="689 1367 1325 1414">Turn off internal matrix scaling in the READ module.</td> </tr> </table>	32	Turn off internal matrix scaling in the READ module.
32	Turn off internal matrix scaling in the READ module.		
	<table border="1"> <tr> <td data-bbox="601 1414 689 1461">64</td> <td data-bbox="689 1414 1325 1461">Turn off internal matrix balancing in the READ module.</td> </tr> </table>	64	Turn off internal matrix balancing in the READ module.
64	Turn off internal matrix balancing in the READ module.		
	<table border="1"> <tr> <td data-bbox="601 1461 689 1619">8192</td> <td data-bbox="689 1461 1325 1619">Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information.</td> </tr> </table>	8192	Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information.
8192	Use in-core sparse Cholesky factorization method. Only valid for the DCMP, SOLVE, and DECOMP modules. The in-core sparse CHOLESKY factorization code is derived from the TAUCS software package. See <a href="http://www.tau.ac.il/~stoledo/taucs/">http://www.tau.ac.il/~stoledo/taucs/</a> for more information.		

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
LDQRKD (170)	Selects the differential stiffness method for CQUAD4 and CTRIA3 elements:	
	0	Version 68, improved method (Default).
	1	Pre-Version 68 method
OLDQ4K (173)	Requests the pre-Version 68 CQUAD4 element stiffness formulation. No value is required after the keyword. Equivalent to SYSTEM(173)=1.	
	0	Default: Current QUAD4 formulation
	1	Requests pre-V68 QUAD4 formulation.
	2	Requests V68 - V70.5 QUAD4 formulation.
Q4TAPER (189)	Specifies the maximum allowable value of taper for CQUAD4 element. Taper is computed by connecting opposite grid points and computing the area of the enclosed triangles. Another way to think of taper is the ratio of the areas on the two sides of a diagonal (Real $\geq$ 0.0; Default = 0.5).	
Q4SKEW (190)	Specifies the minimum allowable value of skew for the CQUAD4 element. Skew is the angle measured in degrees between the lines that join opposite midsides (Real $\geq$ 0.0; Default = 30.0).	
TETRAAR (191)	Specifies the maximum allowable aspect ratio of the longest edge to the shortest altitude for the CTETRA element (Real $\geq$ 0.0; Default=100.0).	
SCRSAVE (196)	Lanczos high performance option: Controls reuse of scratch files in segment logic.	
	0	Do not reuse (Default)
	1	Reuse
MINFRONT (198)	Lanczos high performance option: minimum front size. (The default value is machine dependent).	
NSEGADD (200)	Number of segments in the element error table that is generated in adaptive analysis (Default = 2).	
CORDM (204)	Specifies the default value for CORDM field on the PSOLID entry (Integer $\geq$ -1; Default = 0).	
(205)	Rank to use for real symmetric sparse decomposition high rank update. Default is hardware dependent.	
DCMPSEQ (206)	Selects ordering method for sparse matrix decomposition.	
	0	Method selected automatically in symbolic factoring phase (Default).

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	1	Minimum degree ordering.
	2	Modified minimum degree ordering for indefinite matrices.
	3	No ordering (uses given sequence).
	4	Extreme ordering. (BEND)
	8	METIS ordering. Metis was developed by George Karpis and Vipin Kumar at the University of Minnesota. More information may be found at <a href="http://www.cs.umn.edu/~karypis/metis">http://www.cs.umn.edu/~karypis/metis</a> .
	9	Selects the better of METIS and MMD.
	10	Selects dof-based (rather than grid-based) modified minimum degree ordering.
	68	This option may reduce the number of nonzero factors in the sparse decomposition method.
	132	Similar to 68 but does not require the USET and SILS table as input. Uses extreme reordering.
	136	Same as 132 but performs METIS reordering.
	260	Selects dof-based (rather than grid-based) extreme ordering.
	264	Selects dof-based (rather than grid-based) METIS ordering.
	1024	Selects Minimum Degree reordering in Intel MKL Pardiso sparse solver.
	2048	Selects METIS reordering in Intel MKL Pardiso sparse solver.
	4096	Selects OpenMP (SMP) METIS reordering in Intel MKL Pardiso sparse solver.
USPARSE (209)	Unsymmetric sparse matrix method selection for the decomposition and forward-backward substitution operations.	
	0	Deactivate
	1	Select Nastran unsymmetric sparse factorization (Default).
	16	Select UMFPACK unsymmetric sparse factorization.
	64	Select LUSOL unsymmetric sparse factorization.
PUNCHTYPE (210)	Used to control punch formula.	
	0	“Old” punch, default in MSC Nastran 2001 and earlier versions.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
	1 "New" punch, default starting in MSC Nastran 2004 and uses the NDDL.
	2 Same as 1 except the line numbers are eliminated.
DISTORT (213)	Element distortion fatal termination override. Applies to the TETRA element.
	0 Terminate run (Default)
	1 Do not terminate run.
T3SKEW (218)	Allows the user to control the minimum vertex angle for CTRIA3 elements at which USER WARNING MESSAGE 5491 is issued. See the description of CTRIA3.
(219)	Rank to use for complex symmetric sparse decomposition high rank update (Default = 1).
(220)	Rank to use for real unsymmetric sparse decomposition high rank update (Default = 1).
(221)	Rank to use for complex unsymmetric sparse decomposition high rank update (Default = 1).
(253 - 262)	SYSTEM(252) to (262) have been set aside for user DMAPS. MSC will not use these values in its code in present or future versions. The SSSAlter library may use this range.
MAXSET (263)	Controls the default number of vectors in block or set for Lanczos Eigenvalue extraction. See EIGRL. The default is 7 for most machines but it is machine dependent.
QUARTICDLM (270)	A value of 1 selects the new quartic formulation of the doublet latic kernel (N5KQ), while 0 selects the original quadratic form (Default = 0).
(273)	A value of 1 selects the old Lanczos shift logic from Version 70 and previous systems (Default = 0).
DBCFACT (274)	Option to create an xdb file with a multi-key data format and is intended for "large" xdb files.
	0 no multi-key format (default)
	2 auto-select multi-key format
	4 multi-key format
(275)	Specifies the timeout for ISHELL in seconds. Values greater than 2,678,400 (31 days) will be set to 31 days.
SPLINE_METRICS (281)	Print additional spline quality information

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	=0	Don't print (default).
	=1	Print quality information for each user-generated spline.
	=2	Print quality information for each user-generated spline and each Nastran-generated center-to-corner point spline.
	Note that if SPLINE_METRICS is defined on the NASTRAN statement without a numeric value, then a value of 1 will be used.	
MINDEF (303)	Indefinite mass matrix check, the Default = 1 does not perform the check.	
	> 0	Check is not performed
	< 0	Epsilon is set to -1.E(MINDEF)
	0	MINDEF defaults to -6
MPERTURB (304)	Perturbation factor for indefinite mass matrix. The Default = 1 does not perturb the mass.	
	> 0	The mass is not perturbed
	< 0	The mass 1.E(MPERTURB) is added to the diagonal terms of the mass matrix
	0	MPERTURB defaults to -6. The perturbed mass matrix is used in the subsequent eigenvalue analysis
(309)	If set to 1, requests the pre-Version 70.7 CHEXA8 element stiffness formulation (Default = 0).	
OLDRBE3 (310)	If set to 1, requests the pre-Version 70.7 RBE3 formulation (Default = 0).	
TBCMAG (311)	Change the stiffness to 1.0E2 if using thermal conductivity in Btu/sec/in.F. See Bulk Data entry <a href="#">TEMPBC</a> for more information. The default stiffness is 1.0E10.	
INDEX (316)	Indexes and/or saves a minimum set of data blocks to the database needed to for postprocessing in Patran or the toolkit. This cell must be used with scratch=no on the nastran command. This cell has several options which are set by adding the following values:	
	1	Index IFP data blocks.
	2	Index OFP data blocks.
	4	Save above data blocks to the MASTER dbset.
	16	Save above data blocks to the DBALL dbset.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	For example, INDEX=7 will index the IFP and OFP data blocks and save them to the MASTER dbset. (Scratch=post is equivalent to scratch=no and INDEX=19.) But with INDEX>0 the resulting database is not restartable.	
XMSG (319)	If set to 1, gives extended error messages (Default = 0).	
OLDDAREA (320)	Controls the conversion of DAREA Bulk Data entries for grid and scalar points to equivalent FORCE/MOMENT/ SLOAD Bulk Data entries as appropriate.	
	0	Perform the conversion, but do not give details of the conversion (Default).
	N	Perform the conversion and give details of the first N such conversions.
	-1	Do not perform the conversion.
RSEQCONT (357)	If system cell IFPSTAR=NO, RSEQCONT default = 0	
	1	causes all continuation fields to be ignored and treated as blank. If set to 1, the continuation entries must immediately follow the parent
	2	causes fatal message to be issued if there are orphaned continuation entries.
	System cell (357) only works when IFPSTAR=NO, it is fixed to 1 when IFPSTAR=YES	
QLHOUL (359)	0	Use the user-requested eigensolution method.
	< > 0	When LAN is requested, switch to AHOU if the number of DOFs sent to the eigensolver is $\leq$ "nswitch", an input parameter to the READ module. This parameter has an MPL default of 20. It may be set to other values in the solution sequences, depending on the context. When HOU, MHOU, or AHOU is selected, switch to the new Householder-QL solution (Default = 1).
PRTPCOMP (361)	If set to 1, then the equivalent PSHELLs and MAT2s from PCOMPs or PCOMPGs are printed to the .f06 file provided that ECHO = NONE is not set. (Default = 0, suppresses this printout).	
STRICTUAI (363)	A value of 1 accepts strict UAI/Nastran Bulk Data entries (Default = 0).	
STPFLG (366)	Selects the SUBCASE or STEP layout when there are a number of SUBCASE commands and no STEP command in a Case Control file for SOL 400 (Default = 0).	



Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	0	Keep all SUBCASE commands in the Case Control file and insert a “STEP 1” for each SUBCASE.
	1	Convert all the SUBCASE IDs into STEP IDs, and then insert a “SUBCASE 1” before the first STEP.
QRMETH (370)	When the input consists of CQUADR/CTRIAR elements, QRMETH allows you to convert them to QUAD4/TRIA3 elements or when the input consists of QUAD4/TRIA3 elements to convert them to CQUADR/CTRIAR elements.	
	Some of the output requests for CQUAD4/CTRIA3 are not available for CQUADR/CTRIAR, for example, the CUBIC option on the STRESS Case Control command. In this case, the equivalent CQUADR/CTRIAR options are used.	
	0	Use the user input CQUADR/CTRIAR elements. (Default)
	3	Converts user input CQUADR/CTRIAR into CQUAD4/CTRIA3.
	5	Converts user input CQUAD4/CTRIA3 into CQUADR/CTRIAR.
PARAMCHK (372)	DMAP parameter initialization check.	
	0	Issue User Fatal Message for an input parameter that is not used in a type statement in the subDMAP argument (Default).
	1	Issue User Fatal Message for the initialized parameter.
TZEROMAX (373)	Controls time step adjustment in nonlinear transient analysis.	
	> 0	Maximum number of times to return to time zero.
	0	No initial time step adjustment (identical to V2001), default for SOL 400 with CGAP elements.
	< 0	No limit on DT adjustment.
	4	Default for SOL 400 without CGAP element or for SOL 129.
NOLIN (386)	(Real, Default = 1.0) Tolerance value for controlling adaptive time step bisection for NOLIN1 entries in SOL 129. Turned off for rotor dynamics. For NLRGAP see SYSTEM (431).	
	0.	Bisection is suppressed (same as Version 2001).
	.001	Increase accuracy slightly.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	.01	Increase accuracy a little more.
	1.0	Allow full adaptive time step bisection.
KRYLOV1 (387)	Fast direct frequency response option.	
	-1	Yes
	0	No (Default)
KRYLOV2 (388)	Options related to fast direct frequency response analysis. Selects subspace generation method.	
	1	Lanczos (Default)
	2	Arnoldi (Use for unsymmetric systems)
KRYLOV3 (389)	Options related to fast direct frequency response analysis. Defines exponent of relative accuracy.	
	-4	Error<1.0E-4 (Default)
	-6	Error<1.0E-6
KRYLOV4 (390)	Options related to fast direct frequency response analysis. Defines pole selection distance.	
	0	Next pole is next unconverged frequency (Default).
	2	2*next frequency distance.
	-2	1/2*next frequency distance.
KRYLOV5 (391)	Options related to fast direct frequency response analysis. Selects decomp/fbs tradeoff parameter determining Krylov method accuracy.	
	0	never terminate, do FBS iterations till accuracy is met
	1	terminate when FBS time > decomposition time (Default)
	2	terminate when FBS time > 2 * decomposition time
	-2	terminate when FBS time > (1/2) * decomposition time
BARMASS (398)	Allows the user to select the bar torsional mass moment of inertia. If set to 0, request the pre-MSC Nastran 2004 (Default = 0). If set to greater than 0, the torsional mass moment of inertia term is included in the mass matrix formulation of bar elements. For both values of COUPMASS, the torsional inertia is added. For COUPMASS = 1, the axial mass will be consistent rather than coupled.	
DPBLKTOL (402)	Specifies Bulk Data tolerance value for GRID, CORD2C, CORD2R, and CORD2S entries. (Default = 0). See / Control Input Stream section under Bulk Data Entries below	

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
	< 0	Do not remove duplicate entries
	0.0	Check specified Bulk Data entries for exact physical match and remove duplicates
	> 0	Then perform the DPBLKTOL=0.0 check and additionally GRID entry as duplicate if $\{ x1(i) - x2(i)  \leq DPBLKTOL ; i = 1, 2, 3 \text{ and } (cp1 \neq cp2 \text{ and } cp1 \cdot cp2 = 0) \text{ and } (cd1 \neq cd2 \text{ and } cd1 \cdot cd2 = 0) \text{ and } (ps1 = ps2) \text{ and } (seid1 = seid2)\}$ using entry with $cpi \neq 0$ and $cdi \neq 0$ if possible
OP2NEW (403)	Selects the additional version information in the OUTPUT2 file. (Default = 0)	
	0	Leave alone and unidentified, pre-2004 convention
	1	Add version major, minor, special to tape label and change IFP datablock locate code word 3
DEF_DENS (408)	Set DEFAULT value for the MODEL_CHECK Executive statement MAT_DENSITY=DEFAULT operation (Default = 0.0).	
DEF_TECO (410)	Set DEFAULT value for the MODEL_CHECK Executive statement MAT_TECO=DEFAULT operation (Default = 0.0).	
DEF_TEIJ (411)	Set DEFAULT value for the MODEL_CHECK Executive statement MAT_TEIJ=DEFAULT operation (Default = 0.0).	
DEF_DAMP (412)	Set DEFAULT value for the MODEL_CHECK Executive statement MAT_DAMP=DEFAULT operation (Default = 0.0).	
OPTCOD (413)	Specifies which optimization code to be used in SOL 200 (Default = 0; automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory).	
	<b>Note:</b> Options 1 and 2 are no longer used.	
	3	MSCADS
	4	IPOPT Optimizer
(414)	Only used if COUPMASS < 0. Default = 0 Coupled Mass for CBAR and CBEAM elements.	
	> 0	Lumped Mass matrix will contain translational components only for CBAR and CBEAM elements.
OLDTDMTH (428)	If nonzero, requests the pre-version 2005 r3 method for computing thermal expansion in CHEXA, CPENTA, and CTETRA elements.	

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
NONLRGAP (431)	(Real, Default = -1.0) Sets tolerance value for controlling adaptive time step bisection for NLRGAP entries in SOL 129 and SOL 400. Time step bisection will occur if the contact force magnitude changes more than the tolerance.	
	= 0.	(Same logic as Version 2005)
	< 0.	Tolerance = 1.E+9
	> 0.	Tolerance = 1000./SYSTEM(431)
ESLNRO (443)	Flag to invoke Nonlinear Response Optimization with the concept of Equivalent Static Loads	
	0	No ESLNRO (Default)
	1	Turn on ESLNRO
(444)	0	Selects the pre Nastran 2010 release of IFP for bulk data processing.
	1	Selects the MSC Nastran 2010 release Common Data Model for data processing, do not allow an integer in a field that the only possible format is real. (Default)
	9	Selects the MSC Nastran 2010 release Common Data Model for data processing, allow an integer in a field that the only possible format is real.
IFPSTAR	YES	Selects the MSC Nastran 2010 release Common Data Model for data processing, do not allow an integer in a field that the only possible format is real. (Default)
	NO	Select the pre MSC Nastran 2010 IFP for bulk data processing.
	On command line IFPSTAR=YES or NO is able to express the value of 1 or 0 only.	
MNLQ4C (445)	Allows or disallows corner stress calculations for CQUAD4 if material nonlinear (Default = 1).	
	0	Allows material nonlinear CQUAD4 corner stress calculations if plastic deformation has occurred, results may be totally incorrect.
	1	Disallows material nonlinear CQUAD4 corner stress calculations. If corner requested and material nonlinear sensed, corner is turned off and center only is computed.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference		
(446)	Controls 4 noded CTETRA fluid mass calculations. (Default=0)		
	0	To improve the modal results for fluid mass and remove the possibility of spurious fluid modes, a four point integration method is used.	
	1	A one point integration method is used to compute the fluid mass.	
PARCHILD(449)	Bitwise control of DMP operations in SOL 400 nonlinear analysis.		
	Bit	Value	Description
	1	Default	System determines automatically at job startup. When running the job on multiple hosts, the value is set to 0; otherwise (single host), it is set to 1. <b>User setting of PARCHILD is not recommended.</b>
		0	Child processes execute the entire analysis. This value is recommended for distributed execution.
		1	Child processes only participate in parallel computational tasks. Recommended for DMP running on a single host (i.e., non-distributed DMP execution).
	2	0	DMP linear equation solver disabled.
		1	DMP linear equation solver enabled (default).
	3	0	DMP nonlinear element matrix generation disabled.
		1	DMP nonlinear element matrix generation enabled (default).
	To change defaults for this system cell, use the DOMAINSOLVER command in the <i>MSC Nastran Quick Reference Guide</i> .		
(451)	If set nonzero, requests the method to calculate transverse shear correction for elements using MID4, or “Z0” on the PCOMP to offset a shell element. The default 0 uses a new method that avoids excessive transverse shear stiffness. This is expected to have small effect on most models, unless the shell is thick and the offset is large.		

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
DIFFS(614)	Flag to control differential stiffness computation (Default=0).	
	DIFFS = -2	Enforce differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.
	DIFFS = -1	Skip differential stiffness computation of RBE3 elements in linear buckling analysis regardless of the maximum number of element grids.
	DIFFS = 0	Compute differential stiffness matrix. For nonlinear elements with geometric nonlinear analysis in SOL 400, also compute follower force stiffness. For linear buckling analysis, differential stiffness calculations for RBE3 elements will be skipped when the maximum number of grids exceeds 10000.
DIFFS > 0	Defined via Bulk Data entry MDLPRM,NLDIFF to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis in SOL 400.	
IFPBUFF(624)	Specifies the size of IFPSTAR data base I/O transfers	
	Default value=1024 words The physical I/O size is IFPBUFF words. The maximum value of IFPBUFF is 65536 words.	
	Example: NASTRAN IFPBUFF=8192	
NONUIV (653)	Parameter to select the numeric compute kernel and pivoting methods in MSCLDL and MSCLU sparse direct solvers. Also see the bulk data entry <a href="#">MDLPRM, 2431</a> ,NONUIV.	
	0	Use the native Bunch-Kauffman threshold pivoting in MSCLDL, and the native threshold partial pivoting in MSCLU (Default).
	1	Use no numeric pivoting in MSCLDL and MSCLU. BLAS3 TRSMs are called to compute the pivot column update to improve performance. Ill-conditioned models may die of “singular matrix” during sparse factorization.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference					
	3	LAPACK SYTRFs with Bunch-Kaufman pivoting and GETRFs with partial pivoting are called to perform factorizations, and BLAS3 TRSMs are called to compute pivot column update to improve performance.				
GPU_MIN_RANK (655)	If the rank of the frontal matrix is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. The default value for GPU_MIN_RANK is 32.					
GPU_MIN_FRONT (656)	<p>The criteria for GPGPU execution during matrix factorization are the frontal matrix front size and the rank of the frontal matrix. Minimum dimensions are set via gpu_min_front and its companion parameter, gpu_min_rank. The value specified must be an integer greater or equal to 1.</p> <p>If the front size is smaller than value, the rank update of the front is processed on the CPU. Otherwise, the GPGPU device would be used for the rank update of the front. The default value for GPU_MIN_FRONT is 2048.</p>					
(662)	<p>Controls CBUSH internal load calculation for radial dependence. (Default=0)</p> <table border="1" data-bbox="595 956 1322 1170"> <tr> <td data-bbox="595 956 692 1064">0</td> <td data-bbox="692 956 1322 1064">Radial dependence of PBUSHT entries DOES NOT depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.</td> </tr> <tr> <td data-bbox="595 1064 692 1170">1</td> <td data-bbox="692 1064 1322 1170">Radial dependence of PBUSHT entries DOES depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.</td> </tr> </table>		0	Radial dependence of PBUSHT entries DOES NOT depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.	1	Radial dependence of PBUSHT entries DOES depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.
0	Radial dependence of PBUSHT entries DOES NOT depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.					
1	Radial dependence of PBUSHT entries DOES depend on the alignment of the CBUSH element when subject to nonlinear force deflection relations.					
SEGLOG(664)	<p>Control the logic to compute the default value of PENALT in segment-to-segment contact analysis. Before v2017, Default value is 0.</p> <table border="1" data-bbox="595 1244 1322 1519"> <tr> <td data-bbox="595 1244 692 1352">0</td> <td data-bbox="692 1244 1322 1352">Use the existing/old logic where the default value is based on the average elastic stiffness of the two contact bodies and the default error tolerance.</td> </tr> <tr> <td data-bbox="595 1352 692 1519">1</td> <td data-bbox="692 1352 1322 1519">Use new logic (Default) for PENALT to avoid a too large value of PENALT, where the default value is defined by the body with the softer material and the minimum edge length of the two contact bodies. This logic is often better for thin shell structures.</td> </tr> </table>		0	Use the existing/old logic where the default value is based on the average elastic stiffness of the two contact bodies and the default error tolerance.	1	Use new logic (Default) for PENALT to avoid a too large value of PENALT, where the default value is defined by the body with the softer material and the minimum edge length of the two contact bodies. This logic is often better for thin shell structures.
0	Use the existing/old logic where the default value is based on the average elastic stiffness of the two contact bodies and the default error tolerance.					
1	Use new logic (Default) for PENALT to avoid a too large value of PENALT, where the default value is defined by the body with the softer material and the minimum edge length of the two contact bodies. This logic is often better for thin shell structures.					

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
CNTBKCMP(666)	Control the logic to compute the contact at shell corners in node-to-segment contact analysis to ensure backward compatibility.	
	0	Use the current/new contact logic for shell corners (Default). This logic uses weighted average normal vector and adjusted thickness for a better representation of the sharp corner at both top and bottom of the shell.
	1	Use old contact logic for shell corners to ensure backward compatibility. This logic uses average thickness and average normal vector even at sharp corners.
(670)	Control for Automatic Contact Generation. Used when BCONTACT=AUTO is present in case control.	
	0	Default, Nastran run to completion after generating acg file.
	3	Nastran terminates after generating acg file.
LMFBKCMP(676)	Control the logic to compute the default value of Lagrange multiplier (LMFACT and PENFN)	
	0	(Default) Use new algorithm for LMFACT/PENFN, which are two order lower than previous value used before 2013.0 Version.
	1	Use old algorithm used before 2013.0 Version.
FFOBKCMP(677)	Control the logic to compute the unloading follower force.	
	0	(Default) Use new algorithm for unloading follower force, i.e., follower force in the previous loadcase will be unloaded by follower force method but not by linear interpolation.
	1	Use old algorithm for unloading follower force, i.e., by linear interpolation (before 2013.0 Version).
OLDLCNTR(678)	Ensure backward compatibility for linear connector elements.	
	0	Use existing (after V2014.1) algorithm with RBE3 relationships for linear connectors (Default).
	1	Use old algorithm (V2014.1 or earlier) for linear connectors to ensure backward compatibility.
	PARAM,OLDWELD,YES is equivalent to a value NASTRAN OLDLCNTR=1	



Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference													
(679)	Controls checking for validity of sparse matrix passed to Intel MKL Pardiso.													
	0	Disable matrix checking (Default)												
	1	Enable matrix checking for Column indices must be in increasing order within rows												
	2	Enable matrix validity checking for reusing previously computed permutation in SOL 400												
(684)	Control of matrix type for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso documentation</a>													
RDBOTH(695)	<p>Parameter to select Rayleigh damping approach for rotordynamics (compatibility with V2005) implementation, Integer. A cumulative sum can be provided in case multiple features are desired in the analysis. The parameter is allowed to take values of 1, 2, 4, 8, 16, or any combination of them except both 4 and 8 are included, such as 12, 13, 14 etc. Default value for this parameter is 0.</p> <table border="1" data-bbox="595 817 1322 1519"> <tbody> <tr> <td data-bbox="595 817 692 887">0</td> <td data-bbox="692 817 1322 887">Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)</td> </tr> <tr> <td data-bbox="595 887 692 1095">1</td> <td data-bbox="692 887 1322 1095">Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through “PARAM, ALPHA1” and “PARAM, ALPHA2” are applied to the complete model and Rayleigh damping specified through “ALPHAR1” and “ALPHAR2” in RSPINR/RSPINT are not used</td> </tr> <tr> <td data-bbox="595 1095 692 1138">2</td> <td data-bbox="692 1095 1322 1138">Ignore circulation effects in rotordynamic analysis.</td> </tr> <tr> <td data-bbox="595 1138 692 1277">4</td> <td data-bbox="692 1138 1322 1277">Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.</td> </tr> <tr> <td data-bbox="595 1277 692 1416">8</td> <td data-bbox="692 1277 1322 1416">Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.</td> </tr> <tr> <td data-bbox="595 1416 692 1519">16</td> <td data-bbox="692 1416 1322 1519">This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.</td> </tr> </tbody> </table>		0	Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)	1	Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through “PARAM, ALPHA1” and “PARAM, ALPHA2” are applied to the complete model and Rayleigh damping specified through “ALPHAR1” and “ALPHAR2” in RSPINR/RSPINT are not used	2	Ignore circulation effects in rotordynamic analysis.	4	Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.	8	Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.	16	This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.
0	Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)													
1	Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through “PARAM, ALPHA1” and “PARAM, ALPHA2” are applied to the complete model and Rayleigh damping specified through “ALPHAR1” and “ALPHAR2” in RSPINR/RSPINT are not used													
2	Ignore circulation effects in rotordynamic analysis.													
4	Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.													
8	Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.													
16	This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.													

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
S2SDEF(701)	Parameter to control backward compatibility of Seg-to-Seg contact default values.	
	0	Original Seg-to-Seg contact default values
	1	Using new Seg-to-Seg contact default values in and after 2016 version. (Default)
HDF5 (702)	Control NH5RDB database creation.	
	-1	Do not create NH5RDB database (Default).
	0	Create NH5RDB database without compression.
	1	Create NH5RDB database with compression.
	2	Create uncompressed NH5RDB database without input data.
OBEAMS (725)	Control Linear format output of Advanced Nonlinear Bar Element. Default: 0	
	0	New. Output of beam stress is separated into Bending and Axial stresses.
	1	Old. Output of beam stress combines bending and axial stresses together.
OLDCCONE (726)	The CCONEAX Force/Stress Recovery element coordinate system is inconsistent with the input RINGAX coordinate system relative to the RINGAX A/RINGAX B order. The CCONEAX Force/Stress Recovery has now been made consistent. For those who may have post processing that recognized the inconsistency and do not wish to change, the inconsistency may be restored with OLDCCONE.	
	0	Default: Force/Stress output consistent with RINGAX coordinate.
	1	Restore inconsistent RINGAX A/RINGAX B Force/Stress output.
METS2S (727)	Control Segment to Segment contact. Default: 0	
	0	Support SEGTOSEG only since 2017 BETA and later version
	1	Old. Support SEGSMALL, SEGLARGE, as well SEGTOSEG.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
DMP (728)	The number of Distributed Memory Processes used. This option is only useful for specific Solution Sequences as described in the <i>MSC Nastran HPC User's Guide</i>	
N2SDOF (729)	Control node freedom number of contact Default: 0	
	0	Depend on contact type: 3 for touching contact; 6 for glued contact
	1	Set the node freedom of contact to be 6 always
H5NORDOF(730)	To suppress output rotational components to NH5RDB database	
	0	Output both translational and rotational components to NH5RDB database (Default).
	1	Do not output rotational components to NH5DB database.  Only used when HDF5(702) is 0 or 1.
H5MTX (739)	Write matrix data in separate file	
	1	Write matrix data into a separate file
	0	Do not write matrix data into a separate file (Default)
H5MDL (740)	Write model input data in separate file	
	2	Write model input data only
	1	Write model input data into a separate file
	0	Do not write model input data into a separate file (Default)
	-1	Use OP2GM34 setting for GEOM3 and GEOM4 output
ACCSDSLZ(747)	SDL work area memory size of IFPStar, default=100000	
	> 100000, work area size in word	
H5GM34 (751)	Write GEOM3 and GEOM4 data in NH5RDB	
	-1	Use OP2GM34 setting (Default).
	1	Write GEOM3 and GEOM4 data in NH5RDB
	0	Do not write GEOM3 and GEOM4 data in NH5RDB
UDEFGRID(753)	Control whether having undefined grid points on ASET/ASET1 entries is allowed. Default=0.	
	0	Issue a User Warning Message and continue the simulation.
	1	Issue a User Fatal Message and terminate the simulation

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference
CNTBACK(758)	BIAS=0.0 on BCONPRG/BCTABLE will be used as 0.0 in analysis. Before 2018.2, BIAS=0.0 is treated as default, same as blank, which will turn to the value of BIAS on BCPARA and so the user has to put a small value to approximate absolute 0.0. Since 20190.0, segment to-segment initial stress free contact can adjust geometry to clear gap/penetration. Default=0.
	0   BIAS=0.0 on BCONPRG/BCTABLE is treated as 0.0 and will override BIAS on BCPARA. Segment-to-Segment Initial Stress Free: geometry adjustment as Node-to-Segment method, old method turn off.
	1   (backward compatible) BIAS=0.0 on BCONPRG/BCTABLE is treated as blank; BIAS on BCPARA will be used in analysis.
	2   Switch to Step Glue if Permanent Glue with LGDISP>1
	4   (back to 2018.2) Segment-to-Segment Contact Initial Stress Free: no geometry adjustment, old method turned on.
	8   Segment-to-Segment Contact Initial Stress Free: geometry adjustment plus old method turned on
	OP2IMPV(761)
	0   Use the old (till V2018.2) I8-I4 conversion method.
	1   Use the new conversion method
IGNBLN(767)	Control if ignore blank lines in an entry followed by continuation lines.
	0   (Default) Don't ignore and issue UFM when blank lines followed by continuations are found.
	1   Ignore those blank lines.
BLNMEM(768)	Memory size in bytes to store blank line numbers in input files.
	8000:   Default, 8000 bytes.
	>=800:   User specified size.
	<800:   Will be forcedly changed to 800.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
OPENFSI(776)	Type: Integer	
	OpenFSI interface option.	
	0: (Default) For implicit type, skip the putWettedNodeDisp method call when the current time is converged and no displacement update from last iteration.	
	1 Do not skip the putWettedNodeDisp call even there is no displacement update.	
	Refer to <i>MSC Nastran User Defined Services User's Guide</i> for details.	
H5INFO(789)	Write job run information in NH5RDB or not.	
	1 Yes (Default)	
	0 No	
STRNCUR(778)	In nonlinear <u>strain recovery</u> for the CQUAD4 and CTRIA3 elements, <b>especially for nonlinear composites with offsets and MDLPRM OFFDEF LROFF</b> , an addition final pass of updating midplane strain and curvature is made. In some cases, this results in strain curvature results significantly different than those used to obtain the force and stress results. STRNCUR allows the user to obtain the midplane strain and curvature, and hence the ply strains, that are identical to the midplane strain and curvature used in force and stress recovery operations.	
	STRNCUR=0	Default: geometric nonlinear midplane strain and curvature are updated for strain recovery and may not be identical to those used in force and stress recovery.
	STRNCUR=1	Geometric nonlinear midplane strain and curvature recovery values are the same values as used for element force and stress recovery when nonlinear strain recovery is requested for elements utilizing LROFF.
	STRNCUR=2	Geometric nonlinear midplane strain and curvature recovery values are the same values as used for element force and stress recovery when nonlinear strain recovery is requested.

Table 2-1 System Cell Summary (continued)

System Cell Name (Number)	Function and Reference	
CNT101(786)	Set Linear Contact (LINCNT=1) as default in SOL 101 only	
	0	(Default) Linear Contact off/on depends on LINCNT on BCPARA Note that Linear Contact is off (LINCNT=0) by default when no BCPARA entry present.
	1	Linear Contact is turned on in SOL 101, regardless of LINCNT on BCPARA.
H5XHH (790)	Write BHH, MHH and KHH matrices in NH5RDB	
	0	Do not write BHH, MHH and KHH in NH5RDB (Default)
	1	Write BHH, MHH and KHH in NH5RDB
DELTAU(800)	The CELAS1/3 force convention for linear elements is determined by the relationship $F=K(U1-U2)$ . For material nonlinear CELAS1/3 elements, initiated via the PELAST, TKNID option, the force convention is $F=K(U2-U1)$ .	
	0	(Default) Maintain the current PELAST; TKNID convention $F=K(U1-U2)$ with material nonlinear.
	1	Replace the current PELAST; TKNID convention with the linear convention $F=K(U1-U2)$ .
See Remark 7. of the PELAST entry.		

# 3

## File Management Statements

- Key to Descriptions
- The File Management Section (FMS)

## Key to Descriptions

**RESTART** Reuse Database From a Previous Run

A brief sentence about the function of the statement is given.

Requests that data stored in a previous run be used in the current run.

**Format:**

RESTART PROJECT = 'project-ID' VERSION = [version-ID] [KEEP | NOKEEP]

Describers in uppercase letters are keywords that must be specified as shown.

Describers in lower case are variables.

If the describers are stacked vertically, then only one may be specified.

**Describers:**

Brackets [] indicate that a choice of describers is optional.

Braces {} indicate that a choice of describers is mandatory.

The default describers are shaded.

Each of the describers is discussed briefly. Further details may be discussed under Remarks.

If the describer is in lower case, then it is a variable and the describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the describer must be specified by the user.

The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the describers.

**Remarks:**

- There may be only one RESTART statement in a run.
- A new version-ID is automatically assigned whenever a RESTART is performed.
- If project-ID or version-ID or both are specified and cannot be found a User Fatal Message will be issued.
- The RESTART statement is used in the following sequence: (101) RESTART, (102) PROJECT, (103) VERSION, (104) KEEP, (105) NOKEEP.
- If PROJECT is not specified, the last version under the current project-ID will be used. (See Example 2 below.)

**Examples:**

- RESTART VERSION=7  
Version number 7 will be retrieved for this run (version 8). At the end of the run version 7 will be deleted.
- PROJ='FENDER'  
RESTART  
The last version under project-ID FENDER will be used in the current run.



## The File Management Section (FMS)

The File Management Section (FMS) is primarily intended for the attachment and initialization of Database sets (DBsets) and FORTRAN files. The initialization of DBsets includes specification of their maximum size, member names, and physical filenames. The initialization of FORTRAN files includes the specification of their filenames, FORTRAN unit numbers, and FORTRAN attributes.

In most classes of problems that use MSC Nastran solution sequences (SOLs), no File Management statements are required because a default File Management Section is executed at the beginning of every run. The default File Management Section is described in the [Database Concepts in the MSC Nastran Reference Guide](#). If a restart is desired, then the RESTART statement is required. All other solutions may not be restarted. If the problem is large in terms of requiring significant amounts of memory or disk space, then the INIT, ASSIGN, and EXPAND statements may be required. If any FORTRAN files are required, then the ASSIGN statement is required; for example, the OUTPUT2 DMAP module. The ASSIGN statement is also required to assign databases for DBLOCATE, DBLOAD, and DBUNLOAD. Special database operations are performed by the DBLOCATE, DBLOAD, DBUNLOAD, DBCLEAN, ACQUIRE, DBDICT, DBFIX, DBSETDEL, DBUPDATE, and PROJECT statements.

## File Management Statement Summary

The following is a summary of all File Management statements:

\$	Comment statement.
ACQUIRE	Selects NDDL schema and MSC Nastran delivery database.
ASSIGN	Assigns physical files to DBset members or special FORTRAN files.
CONNECT	Group geometry data by evaluator and database.
DBCLEAN	Deletes selected database version(s) and/or projects.
DBDICT	Prints the database directory in user-defined format.
DBFIX	Identifies and optionally corrects errors found in the database.
DBLOAD	Loads a database previously unloaded by DBUNLOAD.
DBLOCATE	Obtains data blocks and parameters from databases.
DBSETDEL	Deletes DBsets.
DBUNLOAD	Unloads a database for compression, transfer, or archival storage.
DBUPDATE	Specifies the time between updates of the database directory.
ENDJOB	Terminates a job upon completion of FMS statements.
EXPAND	Concatenates additional DBset members to an existing DBset.
INCLUDE	Inserts an external file in the input file.
INIT	Creates a temporary or permanent DBset.
NASTRAN	Specifies values for system cells.
PROJ	Defines the current or default project identifier.

The FMS statements are executed in the following order regardless of their order of appearance in the input file:

NASTRAN, DEFINE  
RFINCLUDE, INCLUDE  
ASSIGN, INIT, EXPAND, DBUPDATE  
PROJECT  
DBCLEAN  
DBFIX  
DBDICT(1)  
DBSETDEL  
ACQUIRE  
RESTART  
DBLOCATE  
DBUNLOAD  
DBLOAD  
DBDIR (2), DBDICT(2)  
ENDJOB

If DBDICT is specified before any of the FMS statements DBSETDEL through DBLOAD, then the directory printout will reflect the processing of DBCLEAN and DBFIX only. If DBDICT is specified after DBSETDEL through DBLOAD, then the directory printout will reflect the processing of all statements in the FMS Section. We recommend that the DBDICT statements be specified last in the FMS Section. Multiple DBLOCATE, DBLOAD, or DBUNLOAD statements are processed in the order in which they appear. If the ENDJOB statement is specified, then only the File Management Section is processed and the Executive Control, Case Control, and Bulk Data Sections are ignored.

## File Management Statement Descriptions

File Management statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

### Description

A brief sentence about the function of the statement is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma as long as the comma is preceded by one or more spaces and no keyword is split across lines. For example:

```
DBLOCATE DATABLK= (KAA)      ,
        WHERE (PROJECT='FRONT BUMPER' AND      ,
        SEID>0 AND VERSION=4)      ,
        LOGI=MASTER3
```

However, if a filename is to be continued on the next line, no space must precede the comma, and the continuation line must have no leading spaces.

### Example

```
ASSIGN      SDB='/jw/johannes/Projects/secret/Aero/Tests/wing/,
Modes/wing_modal.MASTER'
```

Note that all quote marks shown under formats and examples are right-handed single quotation marks and must be entered as such. For example:

```
PROJ=' MYJOB'
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks are generally arranged in order of importance and indicate such things as the FMS statement's relationship to other commands, restrictions and recommendations on its use, and further descriptions of the describers.

### WHERE and CONVERT Clauses

The WHERE clause is used in the selection of items (data blocks and parameters) on the DBDICT, DBLOCATE, DBLOAD, and DBUNLOAD statements. The CONVERT clause modifies qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements.

The WHERE and CONVERT clauses specify values for PROJECT, VERSION, qualifiers, and DBSET. PROJECT specifies the project-ID that is originally defined on the PROJECT FMS statement at the time the project is created. VERSION specifies the desired version-ID under the project-ID. Qualifiers are used to uniquely identify items on the database with the same name. For example, data block KAA has SEID as one of its qualifiers, which is the superelement ID. An item may have more than one qualifier and the collection of all qualifiers assigned to an item is called a path. All data blocks and parameters with qualifiers are defined in the NDDL Sequence (NASTRAN Data Definition Language), see [MSC Nastran DMAP Programmer's Guide](#). Data blocks and parameters are defined on the DATBLK and PARAM NDDL statements. The DATBLK and PARAM statements specify the name of the data block, parameter, and also its pathname. The pathnames are defined on the PATH NDDL statement, which lists the qualifiers assigned to the path. Qualifiers are defined on the QUAL NDDL statement. DBSET specifies the desired DBset. The DBset of an item is specified after the LOCATION keyword on the DATBLK and PARAM NDDL statement.

The format of the WHERE clause is:

```
WHERE (where-expr)
```

where-expr is a logical expression that specifies the desired values of qualifiers, PROJECT, VERSION, and DBSET. If the result of the logical expression is TRUE for an item on the database then the item is selected. For example, WHERE(VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2.

A simple where-expr is a comparison using the following relational operators: =, >, <, ≤, ≥, >> or <<. For example, SEID>0 means if SEID is greater than zero, then the logical expression is true. Several simple where expressions may be joined into one where expression by the following logical operators: AND, OR, XOR, and EQV. The NOT operator may be used to negate a where expression. For example, NOT(SEID>0) is the same as SEID≤0. Arithmetic operations and DMAP functions may also be specified in the where expression (see the [MSC Nastran DMAP Programmer's Guide](#).)

If a qualifier in a where-expr is not a qualifier in the path of a specified item, then the where-expr is set to FALSE. If the where-expr does not contain a specification for all qualifiers in the path of an item, then the unspecified qualifiers will be wildcarded (i.e., quali=\*, all values will be selected.) The default values of qualifiers, PROJECT, VERSION, and DBSET are described under the statement in which the WHERE clause is specified.

Examples of the WHERE clause are:

1. Select all items in the database for all superelements except 10 and 30 from Version 1.  
WHERE (VERSION=1 AND SEID>=0 AND NOT (SEID=10 OR SEID=30))
2. Select all entries in database on DBSET=DBALL from all projects and versions.  
WHERE (PROJECT=PROJECT AND VERSION>0 AND DBSET='DBALL')

The CONVERT clause modifies project- and version-ID, DBset-name (see INIT statement), and qualifier values of items selected by the WHERE clause on the DBLOCATE and DBLOAD statements. It contains one or more assignment statements separated by semicolons. The format of CONVERT clause is:

```
CONVERT (PROJECT=project-expr; VERSION=version-expr; ,  
DBSET=DBset-expr; quali=qual-expr1[;...])
```

The PROJECT and VERSION statements modify the project-ID (see PROJECT FMS statement) and version-ID. The DBSET statement modifies the DBset-name. The value of quali will be replaced by qual-expri for selected items that have quali in their path. qual-expri is any valid expression (see [Expressions and Operators in the MSC Nastran DMAP Programmer's Guide](#)) containing constants or any qualifier name defined in the path of the item. If qual-expri contains names of qualifiers not in the path of the selected item, then a fatal message is issued. If project-expri and/or version-expri produces a project- or version-ID which does not exist, then one will be created. Also, all version-IDs less than version-expri that do not exist will be created; but they will be “empty.”

Examples of the CONVERT clause are:

1. Set qualifiers SEID, PEID, and SPC to constants 10, 20, 102 respectively.

```
CONVERT (SEID=10; PEID=20; SPC=102)
```

If more than one value of a qualifier is found for an item by the WHERE clause, then each value is processed in qual-expri to define the new qualifier value for each of the selected items. In the following example, if the original values of PEID were 1, 2, and 3; then the new values for the SEID qualifier will be 2, 4, and 6.

2. Set all values of qualifier SEID to be twice the value of the PEID qualifier:

```
CONVERT (SEID=2*PEID)
```

## File Management Statements

### \$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

\$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

## ACQUIRE Selects NDDL Schema

Selects the NDDL schema and MSC Nastran delivery database to be used for primary database creation.

Format:

$$\text{ACQUIRE NDDL} = \left\{ \begin{array}{l} \text{NDDL} \\ \text{nddl-name} \end{array} \right\}$$

Describer	Meaning
NDDL	MSC Nastran NDDL schema.
nddl-name	Name of a user NDDL schema specified on a COMPILE NDDL statement when the user NDDL was stored.

Remark:

This statement is used to specify the delivery database when the user wishes to create a solution sequence, yet use the subDMAP objects or NDDL schema or both from the MSC-supplied delivery database.

Example:

The following requests the MSC Nastran NDDL schema to be used when creating a new database.

```
ACQUIRE NDDL
SOL MYDMAP
COMPILE DMAP=MYDMAP, SOUOUT=USROBJ
.
.
LINK MYDMAP, SOLOUT=USROBJ
```

## ASSIGN

## Assigns Physical File

Assigns physical file names or other properties to DBset members or special FORTRAN files that are used by other FMS statements or DMAP modules. Also, assigns physical name and/or other properties to modal neutral files (.mnf) for MSC Nastran/ADAMS interface.

Format 1: Assign a DBset member name

$$\text{ASSIGN log-name} = \left[ \begin{array}{l} = \text{'filename1'} \\ = * \\ = '*\text{'} \end{array} \right] \quad [\text{TEMP}] \quad [\text{DELETE}] \quad [\text{SYS} = \text{'sys-spec'}]$$

Format 2: Assign a FORTRAN file

$$\text{ASSIGN logical-key} \left[ \begin{array}{l} = \text{'filename2'} \\ = * \\ = '*\text{'} \end{array} \right] \quad [\text{UNIT} = \text{u}]$$

$$\left[ [\text{STATUS} = ] \left\{ \begin{array}{l} \text{NEW} \\ \text{OLD} \\ \text{UNKNOWN} \end{array} \right\} \right]$$

$$\left[ [FORM = ] \left\{ \begin{array}{l} \text{FORMATTED} \\ \text{UNFORMATTED} | \text{UNFORMATTED}_{64} \\ \text{UNFORMATTED}_{32} \\ \text{UNFORMATTED}_{\text{MIXED}} \end{array} \right\} \right]$$

$$[\text{DEFER}] \left[ \begin{array}{l} \text{TEMP} \\ \text{DELZERO} \end{array} \right] \quad [\text{DELETE}] \quad [\text{SYS} = \text{'sys-spec'}]$$

$$[\text{IMPORT}]$$



Examples:

1. Assign the DBALL DBset:

```
ASSIGN DB1='filename of member DB1'
INIT DBALL LOGI=(DB1)
```

2. Assign FORTRAN file 12 to the OUTPUT4 module using the ASCII option:

```
ASSIGN OUTPUT4='filename of FORTRAN file'
UNIT=12, FORM=FORMATTED
```

3. Assign FORTRAN file to the OPCASE using the ASCII option:

```
ASSIGN OPCASE='Filename of FORTRAN file', STATUS=NEW
```

4. Define SYS parameters for the SCR300 DBset file using the default file name:

```
ASSIGN SCR300 SYS='...'
```

5. Set the default .op2 file format to UNFORMATTED\_64 and assign two .op2 files, one to unit 12 with the file name "test\_op2.12" and one to unit 35 with file name 'test\_op2.35' in ASCII mode.

```
ASSIGN OUTPUT2 UNFORMATTED_64
...
ASSIGN OUTPUT2='test_op2.12' UNIT=12
ASSIGN OUTPUT2='test_op2.35' UNIT=35 FORM=FORMATTED
```

6. Assign a Fortran unit to the Universal File (UF) containing the FRF information for a test FRF component to be used in an FRF Based Assembly (FBA) process.

```
ASSIGN UNVFILE='testcomp_unv' UNIT=25
```

Descriptor	Meaning
log-name	The name of a DBset member name. log-name may also be referenced on an INIT statement after the LOGICAL keyword.
filename1	The physical filename assigned to the DBset member. If the default filename (if there is one) is to be used, filename1 may be omitted or specified as * or '*'. See Remark 6.
logical-key	Specifies defaults for STATUS, UNIT, and FORM of FORTRAN files for other FMS statements, DMAP modules, punching, and plotting operations.
filename2	The physical file name assigned to the FORTRAN file. If the default filename is to be used, filename2 may be omitted or specified as * or '*'. See Remark 7.
UNIT=u	u is the FORTRAN unit number of the FORTRAN file. If this descriptor is omitted and if filename2 is omitted, this ASSIGN statement will update the defaults for subsequent ASSIGN statements for the same logical-key value. See Remark 7.
TEMP	Requests that the file associated with log-name or logical-key/UNIT be deleted at the end of the run. <b>Warning, using TEMP on OP2 or XDB files will result in the inability to postprocess the results.</b>
DELETE	Requests that the file associated with logical-key/UNIT, if it exists before the start of the run, be deleted.

Descriptor	Meaning
DELZERO	Requests that the file associated with logical-key/UNIT be deleted at the end of the run if it is zero-length; that is, if it does not contain any data.
STATUS	Specifies whether the FORTRAN file is being created (STATUS=NEW) or has been created prior to the run (STATUS=OLD). If its status is not known, then STATUS=UNKNOWN is specified.
FORM	Indicates whether the FORTRAN file is written in ASCII (FORM=FORMATTED) or binary (FORM=UNFORMATTED, UNFORMATTED_64, UNFORMATTED_32, UNFORMATTED_MIX) format. See Remark 10., 11., 12., 13. 18. and 21.
DEFER	Defers opening/creating the specified file. That is, the file will not be opened/created during MSC Nastran initialization. The file must be explicitly opened by the module or DMAP accessing the file using, for example, FORTIO, before it can be used.
sys-spec	System-specific, machine-dependent or application-specific controls. For DBset files, these control I/O performance. For FORTRAN files, only the PLOT and DBC Logical-Key Names use this field. For the DBC Logical-Key, these controls are for I/O performance just as for DBset files. For the PLOT Logical-Key, these controls are used for PostScript processing when FORM=FORMATTED is in effect and are ignored otherwise. See Remark 14.
RECL = l	The size of a block of input/output information specified in words. See Remark 15.
SIZE = s	The number of blocks allocated to the DBC database. See Remark 16.
IMPORT	Imports external ifpdb. When this is absent, IFPDB is for export.

#### Remarks:

1. The ASSIGN statement and its applications are discussed further in the [Database Concepts in the MSC Nastran Reference Guide](#).
2. The log-name or logical-key descriptor must be the first descriptor on the ASSIGN statement. All other descriptors may appear in any order. With the exception of log-name, logical-key, filename1, filename2, and sys-spec, descriptors and values longer than four characters may be abbreviated to four characters.
3. For FORTRAN files, the logical-key names and their default attributes are listed in [Table 3-1](#). If a logical-key name is identified as “Assignable YES”, then the defaults may be overridden on the ASSIGN statement.
4. Certain reserved names may not be used for log-names or logical-key names. These names are the logical names listed in [Table 3-1](#) that are identified as “Assignable NO”. This list includes: SEMTRN, LNKSWH, MESHFL, LOGFL, INPUT, PRINT, INCLD1, and CNTFL. If they are used, then a User Fatal Message is issued. Unit numbers 1 through 10, 14, 16, 18, 19 and 21 should not be assigned. Up to 4000 ASSIGNs are allowed. Some operating systems may have a smaller limit. To avoid that limit use “ulmit -n 4096”. Unit numbers 1234, 1235, 1236, 1133 to 2269 are not allowed.

Some operating systems may have their own limit on the number of open files as well. PUNCH and PLOT may be used, but are not recommended. Most keyword assignments can be specified as command line arguments and/or included in RC files. There are some exceptions such as solve=auto may be specified on the command line or the User RC files, but not in the system RC files (MSC\_BASE/conf/RCfile).

5. If one of the logical-key names indicated in the Remarks 3. and 4. is not specified on this statement, then it is assumed to be a DBset member name log-name as shown in Format 1.
6. If the same log-name is used on more than one DBset ASSIGN statement, the following rules apply:
  - a. If there is no current entry for the specified log-name, a new entry in the DBset tables will be created. If there is an existing entry for the specified log-name, the ASSIGN parameters will modify that entry instead of creating a new one.
  - b. If filename1 is omitted or is specified as \* or ‘\*’, the default file name or, if this is a second or subsequent ASSIGN statement for the same log-name, the previously specified file name (or default name if none was previously specified) will be used.
7. If the same logical-key is used on more than one FORTRAN file ASSIGN statement, the following rules apply:
  - a. If filename2 is omitted (or specified as \* or ‘\*’) and if the UNIT descriptor is omitted, the ASSIGN parameters will modify the system default entry for the logical-key, establishing the new defaults for any subsequent ASSIGN entry for the logical-key. Note, however, that any entries previously created with the same logical-key will not be modified by the new parameters specified on this ASSIGN statement.
  - b. If the value specified by the UNIT descriptor matches the value for an entry created by a previous ASSIGN statement with a UNIT descriptor, then:
    - If the logical-key values are different, a UFM will be generated; and
    - If the logical-key values are the same, the previous entry will be updated instead of having a new entry created.
  - c. If the value specified by the UNIT descriptor does not match the value for an entry created by a previous ASSIGN statement with a UNIT descriptor, then a new entry will be created in the FORTRAN unit tables.
  - d. If the file name is omitted or specified as \* or ‘\*’, the default file name or, if this is a second or subsequent ASSIGN statement for the same logical-key/UNIT combination, a previously specified file name (or default name if none was previously specified) will be used.
8. If you are using IFPSTAR (default), you can use below command to import IFPDAT file created by the initial run if the file is not located in the same directory as MASTER and DBALL files. ASSIGN IFPDB=first\_run\_directory/run1.IFPDAT IMPORT.
9. STATUS, UNIT, and FORM are ignored if assigning a log-name (DBset member name).
10. FORM=FORMATTED must be specified for a unit when:
  - ASCII output is desired from the OUTPUT4 DMAP modules that processes the unit. See the [MSC Nastran DMAP Programmer’s Guide](#).

- `FORMAT=NEUTRAL` is selected on the `DBUNLOAD` and `DBLOAD` FMS statements that process the unit. See the [Database Concepts in the MSC Nastran Reference Guide](#).
  - The neutral file format is desired for the `OUTPUT2` module.
  - PostScript output is desired for `PLOT` requests when “`PLOTTER NAST`” is in effect.
11. For the `DBUNLOAD` FMS statement and the `OUTPUT2` and `OUTPUT4` modules, binary format may be requested using `FORM=UNFORMATTED`, `FORM=UNFORMATTED_64`, `FORM=UNFORMATTED_32` and `FORM=UNFORMATTED_MIX`. The output formats with these keywords are as the following:

FORM	Table (bit)	Matrix (bit)	Old keyword
UNFORMATTED	64	64	UNFORMATTED
UNFORMATTED_64	64	64	LITTLEENDIAN64
UNFORMATTED_32	32	32	N/A
UNFORMATTED_MIX	32	64	LITTLEENDIAN

12. The `FORM=` descriptor is ignored for the `DBLOAD` FMS statement and `INPUTT2` and `INPUTT4` modules. MSC Nastran determines the actual file format when it accesses the specified file. If the `FORM=` descriptor is specified on an `ASSIGN` statement for these logical-keys, the syntax of the descriptor will be validated but will otherwise be ignored. However,
- For non-native binary files the `INPUTT2` modules can only process data blocks with an `NDDL` description. (See the [MSC Nastran DMAP Programmer's Guide](#) under the `DATABLK` statement.) An `NDDL` description is required for `TYPE=TABLE` and none is required for `TYPE=MATRIX`. The data block must be processed with `FORM=UNFORMATTED` if `TYPE=UNSTRUCTURED`, `KDICT`, or `KELM`.
  - Although formatted files are machine independent, if the file in unformatted `DBLOAD` can only process input files in native binary format.
13. For the `DBUNLOAD` FMS statement and `OUTPUT2` module, if `FORM` is other than `UNFORMATTED` (or equivalent, e.g., `UNFORMATTED_32` on a Linux or Windows platform), then only data blocks with an `NDDL` description are processed. (See the [MSC Nastran DMAP Programmer's Guide](#) under the `DATABLK` statement.) An `NDDL` description is required for `TYPE=TABLE` and none is required for `TYPE=MATRIX`. The data block must be processed with `FORM=UNFORMATTED` if `TYPE=UNSTRUCTURED`, `KDICT`, or `KELM`.
14. See the for further information on `sys-spec` controls and on machine-dependent aspects of the `ASSIGN` statement. Also, if there are `SYS` specifications on more than one `ASSIGN` statement specifying the same log-name or logical-key/`UNIT` combination, the second and subsequent specifications will appended to the current `SYS` specification with a comma separator.
15. Currently the `RECL` keyword is used by the `DBC` module and has a default minimum of 1024 words. The maximum allowed is 65536 words and is used to increase the database capacity.

16. The SIZE keyword is used by the DBC module and has a default of 16777215. The maximum allowed is 2147483647 and is used to increase the database capacity. Patran releases before 2001 should use the defaults for RECL and SIZE or database verification failures will occur.
17. logical-key name MNF does not utilize UNIT or FORM.
18. For logical-key DBC, if the .xdb file is new, the desired binary format may be specified in the same way as for the OUTPUT2 and OUTPUT4 modules, as described in Remark 11., except that FORM=FORMATTED is not valid. If the .xdb file is not new, the FORM= descriptor is ignored and MSC Nastran determines the format of the existing .xdb file. MSC Nastran can read and or update an .xdb file in any valid format. Note, on long-word systems (mode=i8) the effect of 64-bit output files is that ADAMS(MNF), Patran, and SimXpert may fail to process the results.
19. SOL700 reserves unit numbers 87-92 for its internal use. When using SOL700 users should not use these unit number to avoid conflicts.
20. The total length of any line in an ASSIGN statement must not exceed 72 characters. Long file names may be split across multiple lines with commas at the end of line and next to be without spaces. For example, the file: `ASSIGN SCR300='E:\brhikesh_scr300\testing\delimiter\issue\scratch.SCR300' DELETE` should be specified with the following input: `ASSIGN SCR300='E:\brhikesh_scr300\testing\delimiter\issue\scratch.SCR300' DELETE.`

**Table 3-1 FORTRAN Files and Their Default Attributes**

Logical Key Name	Physical Name	Unit No.	Form	Status	Assignable	Open	Access	Description/ Application
SEMTRN	sdir/data.f01	1	FORMATTED	NEW	NO	YES	SEQ.	Input Data Copy Unit
LNKSWH	sdir/data.f02	2	UNFORMATTED	NEW	NO	YES	SEQ.	Link Switch Unit
MESHFL	sdir/data.f03	3	FORMATTED	NEW	NO	YES	SEQ.	Input Data Copy Unit
LOGFI	out.f04	4	FORMATTED	NEW	NO	YES	SEQ.	Execution Summary Unit
INPUT	data.dat	5	FORMATTED	OLD	NO	YES	SEQ.	Input File Unit
PRINT	out.f06	6	FORMATTED	NEW	NO	YES	SEQ.	Main Print Output Unit
PUNCH	out.pch	7	FORMATTED	NEW	YES	YES	SEQ.	Default Punch Output Unit
	authorize.dat	8	FORMATTED	OLD	NO	YES	SEQ.	Authorization File
INCLD1					NO			Available for Use
CNTFL					NO			Available for Use
INPUTT2	REQ	REQ		OLD	YES	NO	SEQ.	INPUTT2 Unit
OUTPUT2+	out.op2	12	UNFORMATTED*	NEW	YES	YES	SEQ.	OUTPUT2 Unit
INPUTT4	REQ	REQ		OLD	YES	NO	SEQ.	INPUTT4 Unit
OUTPUT4	REQ	REQ	UNFORMATTED*	NEW	YES	NO	SEQ.	OUTPUT4 Unit
PLOT	out.plt	14	UNFORMATTED++	NEW	YES	YES	SEQ.	Plotter Output Unit
BULKECHO	out.becho	18	FORMATTED	NEW	YES	YES	SEQ.	Bulk EchoUnit
OUTPUT2F	out	19	UNFORMATTED	NEW	YES		SEQ.	Named OUTPUT2 Pattern
OPCASE	REQ	22	FORMATED	NEW	YES		SEQ.	Available for Use
TOPDES	out.des	21	FORMATTED	NEW	YES	YES	SEQ.	Topology Optimization

Table 3-1 FORTRAN Files and Their Default Attributes (continued)

Logical Key Name	Physical Name	Unit No.	Form	Status	Assignable	Open	Access	Description/ Application
AESO	<i>out.AESO</i>	23	FORMATTED	NEW	YES	YES	SEQ.	Optimization
DBC	<i>out.xdb</i>	40	UNFORMATTED	NEW	YES	YES	DIRECT	Database Converter Unit
DBUNLOAD	REQ	50	UNFORMATTED*	NEW	YES	NO	SEQ.	DBUNLOAD FMS statement
DBLOAD	REQ	51		OLD	YES	NO	SEQ.	DBLOAD FMS statement
MNF	<i>out.mnf</i>	none	none	NEW	YES	NO	SEQ.	Interface for ADAMS/Flex
A502LU								Available for Use
DBMIG								Available for Use
USERFILE	REQ	REQ	REQ	REQ	YES	NO	SEQ.	Any User-Defined File
UNVFILE	REQ	REQ	FORMATTED	OLD	YES	NO	SEQ.	Universal File (UF) Unit
ADFFILE	<i>out.afu</i>	none	none	NEW	YES	NO	SEQ.	Associated Data File Unit
IFPDB	<i>out.IFPDB</i>	no	no	NEW	YES	NO	SEQ.	Export or import IFPDAT file
ACG	<i>out.acg</i>	no	no	NEW	YES	NO	SEQ.	Automatic Contact Generation file
HDF5	<i>out.h5</i>	no	no	NEW	YES	NO	N/A	Nastran HDF5 result database
HDF5IN	<i>pre_run.h5</i>	REQ	no	OLD	YES	NO	SEQ.	Read displacement or eigenvector datasets from MSC Nastran hdf5 result database
IMPFIN	<i>example.impf</i>	REQ	no	OLD	YES	NO	N/A	Read imperfection file

where:

- Logical Key Name                      Specifies the logical-key NAME used on the ASSIGN statement.
- Physical Name                         Specifies the default name used to open the file; i.e., the default filename2 name.  
“REQ” means that this parameter is required in the ASSIGN statement from the user.
- Unit No.                                 Specifies the default FORTRAN unit number used by MSC Nastran.  
“REQ” means that this parameter is required in the ASSIGN statement from the user.
- Form                                      Specifies the default FORM used when the file is opened.
- Status                                    Specifies the default STATUS used when the file is opened.  
“REQ” means that this parameter is required in the ASSIGN statement from the user.

Assignable	If “YES”, the user may assign a physical file to this logical name. If “NO”, the unit (if any) and logical name are reserved by MSC Nastran.
Open	If “YES”, the file is opened by default. If “NO”, the file must be explicitly opened.
Access	If “SEQ”, the file is opened for sequential access. If “DIRECT”, the file is opened for direct access.
sdir	The scratch directory specified using the “sdirectory” keyword.
data	The name of the input data file with all directory and extensions removed.
out	The directory and file prefix specified using the “out” keyword or taken by default.

- Notes:**
- + The actual logical-key name for this is “.op2”. If you use “OUTPUT2” (even though this is still the logical-key name put out by Patran) you will get a User Fatal Message from MSC Nastran.
  - \* FORMATTED is required for neutral-format OUTPUT2 files and ASCII-format OUTPUT4 files.
  - ++ If FORM=FORMATTED is specified, the default extension is changed to “.ps” and, if PLOTTER NAST is requested (PLOTTER SC is not supported in PostScript mode), the plot data will be generated in PostScript format directly instead of having to use the PLOTPS or MSCPLOTPS utility programs. Any desired plotting options may be specified as keywords in the SYS= field of the ASSIGN statement or using the SYSFIELD command-line option, specifying PLOT (keyword=value, . . .). The valid keywords are the same as those that can be specified for the PLOTPS or MSCPLOTPS utility programs except that the “begin”, “debug”, “dump”, “end”, “format” and “output” keywords are not allowed. PostScript mode can be made the default by specifying the following ASSIGN statement in an RC file:

```
ASSIGN PLOT=*, FORM=FORMATTED
```

This ASSIGN statement may also include a SYS= specification if any special PostScript keywords are to be used for every plot.

21. Refer to Remarks 10., 11., 12., 13. and 18. above. For UNFORMATTED, the default is 64-bit machine precision. The Linux and Windows platforms, currently supported by MSC Nastran, are binary compatible. FORMATTED involves converting the binary numbers into a compressed ASCII representation, which will result in some roundoff. In a numerically stable problem, the roundoff should not cause problems. However, if the model is not numerically stable or the user is performing operations which require numeric precision, then the roundoff may come into play. See, also, IEEE 754-2008, published in August 2008 for roundoff criterion.





## CONNECT

## User Defined Service and Group Evaluator Data

Defines the newly introduced Simulation Component Architecture (SCA) User Defined Service (UDS) into MSC Nastran via a new CONNECT SERVICE statement. Bulk Data entries such as NLRSD and MATUDS refer to this new service.

External geometric or beam cross section entities, external design responses, and external splines are still defined through the old grouping also described below. These entities should belong to the same evaluator-class (set of routines that process them), and in the case of geometric data, should reside on the same database.

1. A group of external beam cross section entities. These entities should belong to the same evaluator-class (a set of routines that process them). The PBARL and PBEAML entries relate to the BEAMEVAL type.
2. A group of external spline entities. The SPLINEX Bulk Data entry relates to the SPLINEX type.
3. A group of external design response entities. The DRESP3 Bulk Data entry relates to the DRESP3 type.

### UDS Format:

```
CONNECT SERVICE <service_identifier> <service_name>
```

### Old Group Format:

```
CONNECT [
  GEOMEVAL
  BEAMEVAL
  DRESP3
  SPLINEX
] group evaluator 'path' 'data'
```

### UDS Examples:

```
CONNECT SERVICE mysub 'SCA.MDSolver.Util.Ums'
```

Creates a service identifier “mysub” which points to the “SCA.MDSolver.Util.Ums” service.

```
CONNECT SERVICE MYSUB 'ExtServ.Nlrsfd'
```

In this case, the user is requesting the NLRSD Bulk Data entry with the GRPNAME MYSUB obtain its characteristics from the externally connected service defined in the CDL file under the component keyword. The user must create an external shared object library (dynamic link library on Windows) using the SCA Scons utility and configure it to function with the MSC Nastran executable.

### Old Group Examples:

```
CONNECT BEAMEVAL HOIST, NEWBEAMS
```

In this case, the user is requesting that all calculations on PBARL and PBEAML Bulk Data entries that are grouped as HOIST use the NEWBEAMS evaluator. In this case, the user must supply the NEWBEAMS beam cross section evaluator library, and configure it to function with the Nastran executable program.

CONNECT DRESP3 TAILWING, EXTRESP

In this case, the user is requesting that all calculations on DRESP3 Bulk Data entries, that are grouped as TAILWING use the EXTRESP evaluator. Thus, the user must create the EXTRESP external response server program, and configure it to function with the MSC Nastran executable.

CONNECT SPLINEX SPLNGRP EXTSPLN

In this case, the user is requesting that all calculations on SPLINEX Bulk Data entries, that are grouped as SPLNGRP, use the EXTPLN evaluator. Thus, the user must create the EXTSPLN external spline server program and configure it to function with the Nastran executable.

Describer (UDS Format)	Meaning
group	Group name referenced by the GROUP field on the NLRSD and MATUDS Bulk Data entries.
service_identifier	service_identifier is a name tag (8 characters long) which will be used to identify the service implementation to be used for a specific bulk data entry.
service_name	service_name is the name of service

Describer (Old Group Format)	Meaning
group	Group name referenced by the GROUP field on DRESP3, PBARL, PBEAML, and SPLINEX Bulk Data entries.
evaluator	Identifies the particular class of evaluator to which the geometric, beam cross section, external response, or external spline entities belong. Entities belonging to one evaluator-class are handled by the same set of routines (either MSC-provided or user-provided). For geometry, two classes of evaluators are provided internally with MSC Nastran. They are MSCRPC (rational parametric cubic) and MSCEQN (generic equation). For beam cross sections, the class MSCBML (MSC Beam Library) is provided internally. Users may develop custom evaluator libraries for geometry, beam cross sections, external responses, or external splines and configure them for use with MSC Nastran. See Remarks 4, 5, and 7.
path	Optional pathname or filename used by evaluator. Path must be enclosed by single quotation marks if it contains lowercase characters.
data	Optional character string passed to the evaluator. Data must be enclosed by single quotation marks if it contains lowercase characters or embedded blanks.

### Remarks (UDS Format):

1. The process of enabling User Defined Services in MSC Nastran consists of the following four steps:
  - a. Creating the desired implementation for the User Defined Services in the form of dynamic-link libraries,
  - b. Defining the location of the user defined service, service catalogue, service resource directory.
  - c. Specifying the proper commands in the model to load the service,
  - d. Identifying the elements that use the user supplied implementation.
2. In order to create a dynamic-link library suitable for usage with MSC Nastran, a build environment is delivered to assist the user in building the library. Please refer to the *Simulation Component Architecture Guide* and the *User Defined Service Guide* which describe the features and capabilities of the build system.

### Remarks (Old Group Format):

1. CONNECT requests:
  - An external data base or evaluator, or
  - A user-defined grouping for beam cross section data defined by PBARL and PBEAML entries.
2. Two reserved group names, MSCGRP0 and MSCGRP1, have been predefined for geometric entities. The group MSCGRP0 corresponds to the MSCRPC (rational parametric cubic) evaluator and the group MSCGRP1 corresponds to the MSCEQN (Generic Equation) evaluator.
3. A single reserved group name, MSCBML0, has been predefined for beam cross section entities. It may be used in the PBARL and PBEAML entries without being defined explicitly by means of a CONNECT FMS statement. It corresponds to the MSCBML (MSC Beam-Library) evaluator.
4. Custom geometric evaluator libraries developed by users should comply with the *MSC Nastran Geometry Evaluator Developer's Guide*.
5. Custom beam cross section evaluator libraries developed by users should comply with the guidelines in the *MSC Nastran V69 Release Guide*, Section 3.1, Beam Cross-Section Library, and Appendix C: Adding Your Own Beam Cross-Section Library.
6. Custom responses developed by users should comply with the procedures and guidelines in "Support of External Response in SOL 200" on page 55 of the *MSC Nastran 2004 Release Guide*.
7. Once developed, an evaluator may be configured as:
  - Internal, where the evaluator routines are linked with the rest of the Nastran object modules to comprise the Nastran executable program; or
  - External, where the evaluator routines are linked with an MSC-provided server program to constitute an independent geometry server.

## **DBCLEAN**      Deletes Database Versions and/or Projects

Deletes one or more versions and/or projects from the database.

Format:

DBCLEAN VERSION = {version-ID,\*} [PROJECT={project-ID',\*}]

Describer	Meaning
version-ID	Version identifier of the database to be deleted.
*	Wildcard. All versions or projects to be deleted.
project-ID	Project identifier of the project to be deleted. (See the FMS statement, <a href="#">PROJ, 103.</a> )

Remarks:

1. There may be up to ten DBCLEAN statements in the FMS Section.
2. If no project-ID is given, the current project-ID is assumed.

Example:

```
DBCLEAN VERS = 7 PROJ = 'OUTER WING - LEFT'
```

The preceding example would delete from the database all data blocks and parameters stored under Version 7 of the project identified as OUTER WING - LEFT.

## DBDICT

## Prints Database Directory Tables

DBDICT prints the following database directory tables:

- Data blocks described by an NDDL DATABLK statement.
- Parameters described by an NDDL PARAM statement.
- All unique paths (KEYs) and their qualifier values.
- Qualifiers and their current values.
- Data blocks not described by an NDDL DATABLK statement.
- Parameters not described by an NDDL PARAM statement.
- Project and version information.

## Basic Format:

The basic format of DBDICT specifies which tables to print and prints all items (data blocks and parameters) found in the directory. Also, the attributes (colnames) to be printed, and the print format, are predefined. Note that more than one table may be specified on the same DBDICT statement.

DBDICT [DATABLK PARAM PROJVERS QUALCURR QUALIFIERS]

## Examples:

```
DBDICT
DBDICT PARAM PROJVERS
```

## Full Format:

The full format permits the selection of items by name and/or by the WHERE describer. The full format also permits the attributes to be printed using the SELECT describer. In addition, the print format can be specified with the SORT, FORMAT, and LABEL describers. Note that the full format only allows the specification of a single table on a DBDICT statement.

$$\text{DBDICT} \left[ \begin{array}{l} \left( \left[ \begin{array}{l} \text{DATABLK} \\ \text{DATABLK(LOCAL)} \end{array} \right] = \left[ \begin{array}{l} * \\ \text{(datablk-list)} \end{array} \right] \right) \\ \left( \left[ \begin{array}{l} \text{PARAM} \\ \text{PARAM(LOCAL)} \end{array} \right] = \left[ \begin{array}{l} * \\ \text{(param-list)} \end{array} \right] \right) \\ \text{PROJVERS} \\ \text{QUALCURR} \\ \text{QUALIFIERS} \end{array} \right] \text{WHERE}(\text{where-expr}),$$

SELECT(colname[- 'col-label']. . . ),

FORMAT (FWIDTH = w [.d] DWIDTH = w [.d] AWIDTH = a IWIDTH = i,  
LWIDTH = k COLSPACE = c VALUE = w,  
colname = col-width, . . .),

SORT (colname =  $\begin{bmatrix} A \\ D \end{bmatrix}$ , ...),

LABEL ('page - title'  $\begin{bmatrix} \text{RIGHT} \\ \text{CENTER} \\ \text{LEFT} \end{bmatrix}$ )

Describer	Meaning
DATABLK	Print the data blocks. datablk-list specifies a list of NDDL-defined data blocks separated by commas. If LOCAL is specified, the non-NDDL-defined data blocks are printed.
PARAM	Print the parameter table. param-list specifies a list of parameters separated by commas. If LOCAL is specified, the non-NDDL-defined parameters are printed.
PROJVERS	Print the project-version table.
QUALIFIERS	Print the qualifier table.
QUALCURR	Print the current values of the qualifiers. SORT is ignored.
where-expr	Logical expression that specifies the desired values of colnames described below. For example, WHERE(VERSION=4 AND SEID < > 2 AND SEID > 0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for a further description. The default for VERSION is the last version, and PROJECT is the current project. The default for qual is *, which is all qualifier values found on the database. See also Remark 12.
SELECT	Specifies a list of column names to be printed. The order of the specified colnames will be printed from left to right. If colname is not specified, then all columns will be printed.
colname	Column name. Colname specifies a particular attribute of the database item such as data block name (NAME), creation date (CDATE), number of blocks (SIZE), or qualifier name (SEID, SPC, etc.). The allowable colnames are given in the Remarks.
col-label	The label to printed above the column identified by colname. The default for col-label is the colname. col-label may not be specified for the following colnames: QUALSET, QUALALL, and TRAILER.

Describer	Meaning
FWIDTH=w.d	Specifies the default width for single-precision real numbers in real and complex qualifiers (Integers: w>0 and d>0; Default=12.5).
DWIDTH=w.d	Specifies the default width for double-precision real numbers in real and complex qualifiers (Integers: w>0 and d>0; Default=17.10).
AWIDTH=a	Specifies the default width for character string qualifiers. Character strings are printed with enclosing single quotation marks, even if the string is blank (Integer>0; Default=8).
IWIDTH=i	Specifies the default width for integer qualifiers (Integer>0; see Remarks for defaults).
LWIDTH=k	Specifies the default width for logical qualifiers. Logical values are printed as either "T" for TRUE or "F" for FALSE (Integer>0; Default=1).
COLSPACE=c	Specifies the default number of spaces between columns (Integer>0; see Remarks for defaults).
VALUE=w	Specifies the default width for parameter values. The values are printed as character strings with left justification (Integer>0; Default=40)
col-width	The print width of the data under colname or qual-name. For real numbers, specify w.d where w is the width of the field and d is the number of digits in the mantissa. For integers and character strings, specify w, where w is the width of the field. col-width may not be specified for colnames QUALSET, QUALALL, and TRAILER.
SORT	Specifies how the rows are sorted. The sort is performed in order according to each colname specified in the list. A "D" following the colname causes the sort to be in descending order. An "A" following the colname causes the sort to be in ascending order. Colnames QUALSET, QUALALL, and TRAILER may not be specified under SORT. Each colname specified in SORT must be separated by commas.
page-title	A title to be printed on each page of the directory output.
RIGHT, CENTER, LEFT	Print justification of the page title.

## Remarks:

1. DBDICT prints seven different tables according to a default or a user-defined format. The tables are:

Table 1 DBDICT Tables

Describer	Description	Default Page-Title	See Remark
DATABLK	Data blocks described by a NDDL DATABLK statement.	NDDL DATABLOCKS	2.
PARAM	Parameters described by a NDDL PARAM statement.	NDDL PARAMETERS	3.
QUALCURR	Current qualifiers and their values.	CURRENT QUALIFIERS	4.
QUALIFIERS	Qualifiers and their values for each key number.	QUALIFIERS	5.
DATABLK(LOCAL)	Data blocks not described by a NDDL DATABLK statement.	LOCAL DATABLOCKS	6.
PARAM(LOCAL)	Parameters not described by a NDDL PARAM statement.	LOCAL PARAMETERS	7.
PROJVERS	Project-Version.	PROJECT-VERSION	8.

If DBDICT is specified without any describers, then the NDDL Data blocks Table will be printed. See Remark 2.

DATABLK(LOCAL) and PARAM(LOCAL) produce no output, and QUALCURR produces the default values specified on the NDDL QUAL statement.

The defaults and allowable colnames for SELECT, FORMAT, SORT, and LABEL depend on the table. The defaults are described in the following remarks and tables.

2. The default print of the NDDL Data Blocks Table is obtained by

```
DBDICT
```

or

```
DBDICT DATABLK
```

and is equivalent to

```
DBDICT DATABLK ,
  SELECT (NAME, DATABASE, DBSET, PROJ, VERS, CDATE, CTIME,
    SIZE, KEY, PURGED=' PU', EQUIVD=' EQ',
    POINTER=' FILE', QUALSET) ,
  FORMAT (NAME=8, DBSET=8, CDATE=6, CTIME=6, SIZE=5,
    KEY=4 , PURGED=4, EQUIVD=4, POINTER=8,
    IWIDTH=5, COLSPACE=1) ,
  SORT (PROJ=A, VERS=A, DBSET=A, NAME=A) ,
  LABEL (' NDDL DATABLOCKS' CENTER)
```



and looks like:

```

***** DICTIONARY PRINT *****
EXECUTION OF DMAP STATEMENT NUMBER      20
MODULE NAME = DBDICT      , SUBDMAP SEKRRS      , OSCAR RECORD NUMBER      16

NAME      DATABASE DBSET  PROJ VERS  CDATE      NDDL DATABLOCKS
          CTIME  SIZE  KEY  PU  EQ  FILE  SEID  PEID  LOAD  SPC  MPC  METH
-----
AGG      MASTER  DBALL  1    1  930805  72340  0    326  1  0  132484  0  0
AXIC     MASTER  DBALL  1    1  930805  72336  0    315  1  0  65764
BGPDT5   MASTER  DBALL  1    1  930805  72338  1    324  0  2  131332  0
BGPDTX   MASTER  DBALL  1    1  930805  72338  1    324  0  1  131332  0
BJJ      MASTER  DBALL  1    1  930805  72341  0    332  1  0  132612  0
BULK     MASTER  DBALL  1    1  930805  72336  2    315  0  0  65700
CASECC   MASTER  DBALL  1    1  930805  72336  1    316  0  2  67428
    
```

Figure 3-1 DBDICT DATABLE Example

Table 2 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 2 DBDICT DATABLE Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement.
PROJ	4	PROJ NO	Project number associated with PROJECT.
VERS	4	VERSION	Version number.
CDATE	6	CDATE	Creation date.
CTIME	6	CTIME	Creation time.
NAME	8	NAME	Parameter name.
DATABASE	8	DATABASE	MASTER DBset name.
DBSET	8	DBSET	DBset name.
RDATE	6	RDATE	Revision date.
RTIME	6	RTIME	Revision time.
SIZE	5	SIZE	Number of blocks.
qual-name	See Note.	qualifier name	Qualifier name.
KEY	4	KEY	Key number.
TRLi	8	TRLi	i-th word in the trailer.
TRAILER	8	TRLi	All 10 trailer words.
EXTNAME	8	EXTNAME	Extended name.
EQUIVD	4	EQ	Equivalenced flag.



Table 2 DBDICT DATBLK Colnames (continued)

colname	Default col-width	Default col-label	Description
PURGED	4	PU	Purged flag.
EQFLAG	4	EF	Scratch equivalenced flag.
SCRFLAG	4	SF	Scratch DBSET flag.
POINTER	8	POINTER	Directory pointer.
DBENTRY	8	DBENTRY	Database entry pointer.
FEQCHAIN	8	FEQCHAIN	Forward equivalence chain.
BEQCHAIN	8	BEQCHAIN	Backward equivalence chain.
DBDIR20	9	DBDIR(20)	Directory word 20.
QUALALL	See Note.	qualifier name	All qualifiers.
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers.

**Note:** Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, AWIDTH=8, and FWIDTH=12.5.

3. The default print of the NDDL Parameter Table is obtained by

```
DBDICT PARAM
```

and is equivalent to

```
DBDICT PARAM,
  SELECT (NAME, DATABASE, DBSET, PROJ, VERS, CDATE, CTIME,
    KEY, VALUE, QUALSET) ,
  FORMAT (NAME=8, DATABASE=8, DBSET=8, CDATE=6, CTIME=6,
    KEY=4, VALUE=40, IWIDTH=5, COLSPACE=1) ,
  SORT (PROJ=A, VERS=A, DBSET=A, NAME=A) ,
  LABEL ('NDDL PARAMETERS' CENTER)
```

and looks like:

```

***** DICTIONARY PRINT *****
EXECUTION OF DMAP STATEMENT NUMBER      21
MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER      17

NAME      DATABASE  DBSET  PROJ VERS  CDATE  CTIME  NDDL PARAMETERS
          KEY VALUE                                SEID  PEID  LOAD  SPC  MPC  METH
-----
ACOUSTIC MASTER  MASTER  1    1    930805  72338  323 0          0    0
ALTRED   MASTER  MASTER  1    1    930805  72338  319 NO         0
BCHNG   MASTER  MASTER  1    1    930805  72337  325 FALSE      0
DBALLX  MASTER  MASTER  1    1    930805  72336  318 DBALL     -1   -1
EPSBIG  MASTER  MASTER  1    1    930805  72339  323 1.000000E+12 0    0
ERROR   MASTER  MASTER  1    1    930805  72338  319 -1          0    0
FIXEDB  MASTER  MASTER  1    1    930805  72338  323 0          0    0

```

Figure 3-2 DBDICT PARAM Example

Table 3 gives the allowable colnames along with a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 3 DBDICT PARAM Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement.
PROJ	5	PROJ	Project number associated with PROJECT.
VERS	4	VERS	Version number.
CDATE	6	CDATE	Creation date.
CTIME	6	CTIME	Creation time.
NAME	8	NAME	Parameter name.
DATABASE	8	DATABASE	MASTER DBset name.
DBSET	8	DBSET	DBset name.
RDATE	6	RDATE	Revision date.
RTIME	6	RTIME	Revision time.
POINTER	8	POINTER	Directory pointer.
VALUE	40	VALUE	Parameter value.
KEY	4	KEY	Key number.
qual-name	See Note.	qualifier name	Qualifier name.
QUALALL	See Note.	qualifier name	All qualifiers.
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers.





Table 5 DBDICT QUALCURR Colnames

colname	Default col-width	Default col-label	Description
qual-name	See Note.	qualifier name	Qualifier name.
QUALALL	See Note.	qualifier name	All qualifiers.
QUALSET	See Note.	qualifier name	Predefined subset of all qualifiers.

**Note:** Default widths for qualifiers are DWIDTH=17.10, IWIDTH=5, LWIDTH=1, and FWIDTH=12.5. AWIDTH defaults to the length specified on the QUAL statement in the NDDL sequence.

6. The default print of the Local Data Block Table is obtained by

```
DBDICT DATABLK (LOCAL)
```

and is equivalent to

```
DBDICT DATABLK (LOCAL) ,
  SELECT (NAME, SUBDMAP, SIZE='BLOCKS', PURGED='PU',
    EQUIVD='EQ', POINTER, TRL1, TRL2, TRL3, TRL4,
    TRL5, TRL6, TRL7) ,
  FORMAT (NAME=8, SUBDMAP=8, IWIDTH=8, COLSPACE=2) ,
  SORT (NAME=A) LABEL ('LOCAL DATABLOCKS' CENTER)
```

and looks like:

```

***** DICTIONARY PRINT *****
EXECUTION OF DMAP STATEMENT NUMBER      23
MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER      19
LOCAL DATABLOCKS
NAME  SUBDMAP  BLOCKS  PU  EQ  POINTER  TRL1  TRL2  TRL3  TRL4  TRL5  TRL6  TRL7
-----
CASEW PHASE1DR      1    0  0  131780    201    4    0   308    0    0    0

```

Figure 3-5 DBDICT DATABLK(LOCAL) Example

TRL<sub>i</sub> specifies the data block trailer word *i* where  $1 \leq i \leq 10$ . TRAILER selects all 10 data block trailer words.

Table 6 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 6 DBDICT DATBLK(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name.
SUBDMAP	8	SUBDMAP	SubDMAP name.
SIZE	8	BLOCKS	Number of blocks.
EQUIVD	8	EQ	Equivalenced flag.
PURGED	8	PU	Scratch flag.
POINTER	8	POINTER	Directory pointer.
TRLi	8	TRLi	i-th word in the trailer.
TRAILER	8	TRLi	All 10 trailer words.
EXTNAME	8	EXTNAME	Extended name.

7. The default print of the local parameter table is obtained by

```
DBDICT PARAM(LOCAL)
```

and is equivalent to

```
DBDICT PARAM(LOCAL) SELECT (NAME, SUBDMAP, VALUE),
    FORMAT(COLSPACE=4, VALUE=40, AWIDTH=8),
    SORT(NAME=A) LABEL(' LOCAL PARAMETERS' CENTER)
```

and looks like:

```

* * * * D I C T I O N A R Y   P R I N T   * * * *
EXECUTION OF DMAP STATEMENT NUMBER      24
  MODULE NAME = DBDICT   , SUBDMAP SEKRRS   , OSCAR RECORD NUMBER      20
                                LOCAL PARAMETERS
NAME      SUBDMAP      VALUE
-----
AERO      SESTATIC     FALSE
AERO      PHASE1DR     FALSE
ALTRED    SESTATIC     NO
ALTRED    PHASE1DR     NO
ALWAYS    PHASE1DR     -1
ALWAYS    PHASE1C     -1
ALWAYS    SEKRRS      -1
ALWAYS    SESTATIC     -1
APP       PHASE1DR     STATICS
APP       PHASE1C     STATICS
APP       SESTATIC     STATICS
APRCH     SESTATIC
ASING     PHASE1DR     0
ASING     SEKRRS      0
ASING     PHASE1C     0
ASING     SESTATIC     0

```

Figure 3-6 DBDICT PARAM(LOCAL) Example

Table 7 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 7 DBDICT PARAM(LOCAL) Colnames

colname	Default col-width	Default col-label	Description
NAME	8	NAME	Parameter name.
SUBDMAP	8	SUBDMAP	SubDMAP name.
VALUE	40	VALUE	Parameter name.

8. The default print of Project-Version Table is obtained by

DBDICT PROJVERS

and is equivalent to

```
DBDICT PROJVERS ,
  SELECT (PROJECT=' PROJECT NAME' , PROJ=' PROJ NO.' ,
    VERS=' VERSION' , DELFLG=' DELETED' ,
    CDATE=' CREATION DATE' CTIME=' CREATION
    TIME' ) ,
  FORMAT (PROJECT=40 , PROJ=10 , VERS=10 , DELFLG=7 ,
    COLSPACE=1 , CDATE=13 , CTIME=13) ,
  LABEL (' PROJECT-VERSION' , CENTER) ,
  SORT (PROJ=A , VERS=A)
```

and looks like:

```
***** DICTIONARY PRINT *****
EXECUTION OF DMAP STATEMENT NUMBER      19
MODULE NAME = DBDICT , SUBDMAP SEKRRS , OSCAR RECORD NUMBER      15
PROJECT-VERSION
PROJECT NAME          PROJ NO.   VERSION DELETED  CREATION DATE  CREATION TIME
-----
'LEFT FENDER          '          1           1           930805           72319
```

Figure 3-7 DBDICT PROJVERS Example

Table 8 gives the allowable colnames and a description that may be specified in the FORMAT, SELECT, and SORT descriptors.

Table 8 DBDICT PROJVERS Colnames

colname	Default col-width	Default col-label	Description
PROJECT	40	PROJECT NAME	Project name defined by PROJECT statement.
PROJ	10	PROJ NO	Project number associated with PROJECT.



Table 8 DBDICT PROJVERS Colnames

colname	Default col-width	Default col-label	Description
VERS	10	VERSION	Version number.
DELFLG	7	DELETED	Flag indicating whether this project/version has been deleted by the RESTART NOKEEP or DBCLEAN statements.
CDATE	13	CREATION DATE	Creation date.
CTIME	13	CREATION TIME	Creation time.

CDATE is printed as YYMMDD where YY, MM, and DD are the year, month, and date, respectively. CTIME is HHMMSS where HH, MM, and SS are the hour, minute, and second, respectively.

9. If a parameter or qualifier value is defined to be character string, then the value will be printed with enclosing single quotation marks. Blank strings will also be printed with single quotation marks.
10. If a given qualifier is not in the path of a given data block or parameter, then blank spaces will be printed.
11. A line will wrap if additional columns need to be printed and not enough space is available on the output (assumed to be 132). The first column of each additional line is to be indented by the width of the first column printed for the entry.
12. The where-expr has the following rules:
  - If the where-expr specifies a colname that is not assigned to the data block or parameter, then no directory information will be printed for that data block or parameter. For example, given that SPC is not a qualifier for KGG, the following DBDICT statement will produce no output:
 

```
DBDICT DATABLK=KGG WHERE (SPC=10)
```
  - If the where-expr does not specify a colname that is assigned to the data block (or parameter), then the qualifier is wildcarded. For example, given that SEID is a qualifier for KAA, the following DBDICT statements are equivalent:
 

```
DBDICT DATABLK=KAA
DBDICT DATABLK=KAA WHERE (SEID = *)
```
13. A colname specified in the where-expr must be specified in the SELECT clause if the SELECT clause is also specified.

#### Examples:

1. Print the project version table with a title.

```
DBDICT PROJVERS SORT (PROJ, VERSION) LABEL (' PROJECT
VERSION TABLE' LEFT)
```

2. Print a directory of all data blocks qualified with PEID=10 or SEID=10. Print columns for the NAME and DBSET, and the qualifiers SPC, MPC, and LOAD.

```
DBDICT DATABLK SELECT (NAME, SPC, MPC, LOAD, DBSET, SIZE,
```

```
SEID, PEID) ,  
SORT (NAME, SIZE=D) WHERE ( SEID=10 OR PEID=10)
```

**DBDIR**

## Prints Database Directory Tables

Obsolete. See the DBDICT statement.

## DBFIX Database Directory Error Detection

Detects and optionally corrects errors in the database directory.

```
DBFIX { LIST } { CORRECT }
      { NOLIST } { NOCORRECT }
```

Example:

```
DBFIX LIST, NOCORRECT
```

The preceding example requests a printout of the directory pointers and any errors, but not the corrections.

Describer	Meaning
LIST	Requests a debug listing of the database directory pointers.
NOLIST	Suppresses a debug listing of the database directory.
CORRECT	Corrects the database if any errors are found.
NOCORRECT	Suppresses the correction of the database.

Remarks:

1. It is recommended that a backup copy of the database be made before this statement is used, since corrections of the database are achieved through the deletion of data. Data blocks and parameters are deleted from the database if they have (1) incorrect paths (different than listed in the NDDL); (2) incorrect names (two or more names that are not equivalenced and reference the same data), or (3) incorrect directory pointers.
2. NOLIST does not suppress the listing of any corrections made to the database.

## DBLOAD Loads a Database from a FORTRAN File

Recovers data blocks or parameters from a database created by the DBUNLOAD statement.

Format:

$$\text{DBLOAD} \left[ \text{DATABLK} = \left[ \begin{array}{c} * \\ \text{(datablk-list)} \end{array} \right] \text{PARAM} = \left[ \begin{array}{c} * \\ \text{(param-list)} \end{array} \right] \text{WHERE}(\text{where-expr}) ,$$

$$\text{CONVERT}(\text{convert-expr}) \text{UNIT} = \text{unit} \text{FORMAT} \left\{ \begin{array}{l} \text{BINARY} \\ \text{NEUTRAL} \end{array} \right\} \left\{ \begin{array}{l} \text{OVRWRT} \\ \text{NOOVRWRT} \end{array} \right\}$$

Example:

1. Load the database stored in ASCII format on FORTRAN unit 12.

```
DBLOAD      UNIT=12  FORMAT=NEUTRAL
ASSIGN      DBLOAD='physical file name of unloaded database'
            UNIT=12  FORMATTED
```

2. Load version 1 of KAA under project FRONT BUMPER and store it on the primary database under version 5 and project BUMPER. Overwrite duplicates found on the primary database.

```
DBLOAD      DATABLK=(KAA)  WHERE (PROJECT='FRONT BUMPER'
            AND, SEID=10 AND VERSION=1)  CONVERT (VERSION=5; ,
            PROJECT='BUMPER')  OVRWRT
ASSIGN      DBLOAD='physical file name of unloaded database'
```

Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is * which selects all data blocks. The loaded data block may be renamed in the primary database by specifying a slash after the old name, followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The loaded parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSETS is to be renamed to LUSET, then PARAM=(LUSETS/LUSET) is specified.

Describer	Meaning
where-expr	<p>A logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example, WHERE(VERSION=4 AND SEID&lt;&gt;2 AND SEID&gt;0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for more information on WHERE and CONVERT clauses.</p> <p>The default for VERSION is * for all versions; PROJECT is * for all projects; and DBSET is* for all DBsets. The default for qual is *, which is all qualifier values found on the loaded database. See also Remark 8.</p>
convert-expr	<p>Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is:</p> <p>PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expr[;..]</p> <p>For example, CONVERT (SEID=100 + SEID; SPC=102). See the beginning of this section for more information on WHERE and CONVERT clauses.</p> <p>The default action for VERSION and PROJECT is to use the same version IDs and project IDs; i.e., CONVERT(PROJECT=PROJECT; VERSION=VERSION). But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. The default action for qualifiers and DBSET is to use the same values as long as they are defined in both databases. If not, see Remark 8.</p>
unit	<p>Specifies the FORTRAN unit number of the database to be loaded. The unit must be specified on an ASSIGN statement that references the physical filename of the loaded database. The default is 51.</p>
OVRWRT NOOVRWRT	<p>By default, if duplicate data blocks or parameters exist on the loaded and primary databases, then a fatal message is issued. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as a data block or parameter on the primary database.</p>
NEUTRAL BINARY	<p>The database to be loaded may be in BINARY or NEUTRAL format. BINARY indicates the database to be loaded is in binary or FORTRAN unformatted format. NEUTRAL indicates the database to be loaded is in ASCII format. The default is BINARY.</p>

#### Remarks:

1. The DBLOAD statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATBLK keyword is specified and PARAM is not specified, then only data blocks may be loaded. If the PARAM keyword is specified and DATBLK is not specified, then only parameters may be loaded. If neither DATBLK nor PARAM is specified, then all data blocks and parameters may be loaded.

3. The DB keyword is equivalent to DATABLK, and the PARM keyword is equivalent to PARAM.
4. The database to be loaded is attached as read-only. In other words, items can only be fetched and not stored on this database.
5. If more than one DBLOAD statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOAD statements, then the last duplicate will be used.
6. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If the database to be loaded and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
  - If a qualifier in the NDDL of the database to be loaded is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double-precision, complex, or character, then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not loaded.
  - If a DBset-name in the NDDL of the database to be loaded is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the database to be loaded.
9. Data blocks that are equivalenced on the database to be loaded remain equivalenced as long as they are loaded in the same DBLOAD statement or in consecutive DBLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.
10. It is not possible to restart from a database created by DBLOAD in the same run.
11. SOL 190 (or DBTRANS) is also required with DBLOAD if:
  - The database to be loaded has a different BUFFSIZE.
  - The database to be loaded is in neutral format or is being transferred between different machine types.

See also the [Database Concepts](#) in the *MSC Nastran Reference Guide*.

## DBLOCATE Attaches Secondary Databases

Obtains data blocks or parameters from prior versions of the primary database, or other databases. DBLOCATE may also be used to compress the primary database and to migrate databases created in prior MSC Nastran versions.

Format:

```
DBLOCATE [ DATABLK= [ * ] (datablk-list) PARAM= [ * ] (param-list) WHERE(where-expr) ,
```

```
CONVERT(convert-expr) LOGICAL = dbname [ OVRWRT ] [ NOOVRWRT ] COPY ]
```

Example:

1. Locate in version 4 of MASTER3 all data blocks named KAA for all superelements with IDs greater than 0.

```
DBLOCATE      DATABLK=(KAA) WHERE (PROJECT='FRONT BUMPER' , AND
              SEID>0 AND VERSION=4) LOGI=MASTER3
ASSIGN        MASTER3='physical file name of master DBset'
```

2. Copy all data blocks and parameters from the last version of MASTER3 to the primary database. For all items with the qualifier SEID, change the SEID to twice the old ID number.

```
DBLOCATE      CONVERT (SEID=2 *SEID) COPY LOGI=MASTER3
ASSIGN        MASTER3='physical file name of master DBset'
```

3. Compress a database with multiple versions. All versions under the current project-ID (see PROJ statement) will be copied from the database OLDDDB to NEWDB.

```
ASSIGN        MASTER3=' physical filename of new master DBset'
ASSIGN        OLDDDB='physical filename of old master DBset'
DBLOCATE      LOGI=OLDDDB COPY WHERE (VERSION=*) ,
              CONVERT (VERSION=VERSION ; PROJECT=PROJECT)
```



Describer	Meaning
datablk-list	Specifies a list of data blocks separated by commas. The default is *, which selects all data blocks. The located data block may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if KLL is to be renamed to KLL1, then DATABLK=(KLL/KLL1) is specified.
param-list	Specifies a list of parameters separated by commas. The default is *, which selects all parameters. The located parameter may be renamed in the primary database by specifying a slash after the old name followed by the new name. For example, if LUSSETS is to be renamed to LUSET, then PARAM=(LUSSETS/LUSET) is specified.
where-expr	A logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example, WHERE(VERSION= 4 AND SEID <> 2 AND SEID > 0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section for more information on WHERE and CONVERT clauses.  The default for VERSION is the last version-ID and PROJECT is the current project-ID. The default for qual is *, which is all qualifier values found on the located database. See also Remark 9.
convert-expr	Modifies the values for PROJECT, VERSION, DBSET, and qualifiers selected by the where-expr. The format of convert-expr is:  PROJECT=project-expr; VERSION=version-expr; DBSET=DBset-name; quali=qual-expr1[;...]  For example, CONVERT (SEID=100+SEID; SPC=102)  See the beginning of this section for more information on WHERE and CONVERT clauses.  The default action for VERSION and PROJECT is to convert to the current version-ID and current project-ID. But if either PROJECT or VERSION is specified in the convert-expr, then both must be specified. See Example 3. The default action for qualifiers and DBSET is to use the same values as long both databases have the same NDDL scheme. If not, see Remark 9.
dbname	Specifies the logical name of the master directory DBset of the located database. dbname must be specified on an ASSIGN statement, which references the physical file name. By default, the located database is also the primary database. (If dbname is specified for the primary database, then dbname must be MASTER.)
OVRWRT NOOVRWRT	By default, duplicate data blocks or parameters on the located database will take precedence over those on the primary database. A duplicate means that a data block or parameter has not only the same name but also the same qualifier values, PROJECT, VERSION, and DBSET as the data block or parameter on the primary database. If NOOVRWRT is specified, then a fatal message is issued.
COPY	Requests that the located data blocks or parameters be copied to the primary database.

**Remarks:**

1. The DBLOCATE statement and its applications are discussed further in the [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATABLE keyword is specified and PARAM is not specified, then only data blocks may be located. If the PARAM keyword is specified and DATABLE is not specified, then only parameters may be located. If neither DATABLE nor PARAM is specified, then all data blocks and parameters may be located.
3. The DB keyword is equivalent to DATABLE, and the PARM keyword is equivalent to PARAM.
4. If more than one DBLOCATE statement is specified, then they will be processed in the order in which they appear. If a duplicate data block or parameter is found on two or more different DBLOCATE statements, then the last duplicate will be used.
5. If the located database is not the primary database, then it is attached for read-only purposes. In other words, items can only be fetched and not stored on the located database.
6. If the RESTART FMS statement is also specified, then located data blocks and parameters are treated as if they exist in the restart version. In other words, restart equivalences will be made on located items at the beginning of the run and can be subsequently broken as a result of regeneration and/or NDDL dependencies.
7. If a data block or parameter is being renamed, then the new name must be defined in the NDDL of the primary database.
8. If LOGICAL refers to the primary database and one version is to be copied to another, then the items are equivalenced.
9. If the located database and the primary database have different NDDL schemes and CONVERT is not used, then the following is performed:
  - If a qualifier in the NDDL of the located database is not in the NDDL of the primary database, then all of its values are converted to the null value corresponding to its type. For example, if the qualifier is integer, real, double precision, complex or character then the value is converted to 0, 0., 0.D0, (0.,0.), or blank, respectively. If this conversion results in a duplicate data block(s) or parameter(s), then a User Warning Message is printed and the duplicates are not located.
  - If a dbset-name in the NDDL of the located database is not in the NDDL of the primary database, then its values will be converted to the PARAM default value in the NDDL of the located database.

## DBSETDEL Deletes a DBset

Deletes a DBset, all of its members, and associated physical files.

### Format:

```
DBSETDEL dbsetnamei
```

### Example:

Delete DBset DBUP20 from the database.

```
DBSETDEL DBUP20
```

Describer	Meaning
dbsetnamei	Specifies the name(s) of DBset(s) to be deleted. The DBset names MASTER, OBJSCR, or SCRATCH may not be specified.

### Remarks:

1. The DBSETDEL statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If dbsetnamei does not exist, then no action is taken.
3. After a DBset has been deleted with this statement, it may be recreated with the INIT statement in a subsequent run.

## DBUNLOAD

Unloads a Database to a FORTRAN File

Stores data blocks or parameters from the primary database onto a FORTRAN file in a binary or neutral format, for purposes of database compression or database transfer between different computers.

Format:

$$\text{DBUNLOAD} \left[ \text{DATBLK} = \left[ \begin{array}{c} * \\ \text{(datblk-list)} \end{array} \right] \text{PARAM} = \left[ \begin{array}{c} * \\ \text{(param-list)} \end{array} \right] \text{WHERE}(\text{where-expr}) \right]$$

$$\text{UNIT} = \text{unit} \text{FORMAT} = \left\{ \begin{array}{l} \text{BINARY} \\ \text{NEUTRAL} \end{array} \right\} \left\{ \begin{array}{l} \text{REWIND} \\ \text{NOREWIND} \end{array} \right\}$$

Example:

1. Unload the database in ASCII format onto FORTRAN unit 12.

```
DBUNLOAD      UNIT=12  FORMAT=NEUTRAL
ASSIGN        DBUNLOAD='physical file name of FORTRAN unit 12' ,
              UNIT=12  FORMATTED
```

2. Unload version 1 of KAA under project FRONT BUMPER.

```
DBUNLOAD      DATBLK=(KAA) WHERE(PROJECT='FRONT BUMPER' ,AND
              SEID=10 AND VERSION=1)
ASSIGN        DBUNLOAD=' physical file name of FORTRAN unit 50'
```

Describer	Meaning
datblk-list	Specifies a list of data blocks separated by commas. The default is *, which selects all data blocks.
param-list	Specifies a list parameters separated by commas. The default is *, which selects all parameters.
where-expr	Logical expression that specifies the desired values of qualifiers PROJECT, VERSION, and DBSET. For example, WHERE (VERSION=4 AND SEID<>2 AND SEID>0) selects all items under version 4 for all values of SEID greater than 0 except 2. See the beginning of this section on WHERE and CONVERT Clauses.

Describer	Meaning
	The default for VERSION is * for all versions; PROJECT is* for all projects; and DBSET is * for all DBsets. The default for qual is*, which is all qualifier values found on the primary database.
unit	Specifies the FORTRAN unit number to unload the database. The unit must be specified on an ASSIGN statement, which references its physical filename. The default is 50.
NEUTRAL BINARY	The database may be unloaded in BINARY or NEUTRAL format. BINARY indicates the database is to be unloaded in binary or FORTRAN unformatted. NEUTRAL indicates the database is to be unloaded in ASCII format. The default is BINARY.
NOREWIND REWIND	By default, if DBUNLOAD is executed more than once for the same unit, then the unit is not rewound. REWIND requests that the unit be rewound prior to unloading.

Remarks:

1. The DBUNLOAD statement and its applications are discussed further in [Database Concepts in the MSC Nastran Reference Guide](#).
2. If the DATABLE keyword is specified and PARAM is not specified, then only data blocks may be unloaded. If the PARAM keyword is specified and DATABLE is not specified, then only parameters may be unloaded. If neither DATABLE nor PARAM is specified, then all data blocks and parameters may be unloaded.
3. The DB keyword is equivalent to DATABLE, and the PARM keyword is equivalent to PARAM.
4. If more than one DBUNLOAD statement is specified, then they will be processed in the order in which they appear.
5. If NEUTRAL is specified, then the FORMATTED keyword must be specified on the corresponding ASSIGN statement.
6. If NEUTRAL is specified, then only data blocks with an NDDL description are unloaded. (See the [MSC Nastran DMAP Programmer's Guide](#) under the DATABLE statement.) An NDDL description is required for TYPE=TABLE and none is required for TYPE=MATRIX. The data block must be unloaded in BINARY if TYPE=UNSTRUCTURED, KDICT, or KELM.
7. Data blocks that are equivalenced on the primary database remain equivalenced as long as they are unloaded in the same DBUNLOAD statement or in consecutive DBUNLOAD statements with the same unit number. Otherwise, a separate copy for the secondary data block is produced.

**DBUPDATE** Specifies Database Directory Update Interval

Specifies the maximum length of CPU time between database directory updates to the MASTER DBset. This statement is intended to be used if the INIT MASTER(RAM=r) option is specified.

**Format:**

DBUPDATE [=] update-time

**Example:**

DBUPDATE = 5.5

The preceding example would call for a database directory update at the end of a DMAP module execution after five and one-half minutes of CPU time have elapsed from the last update.

Describer	Meaning
update-time	CPU time interval in minutes (real or integer) between database directory updates.

**Remarks:**

1. The difference in CPU time from the last update is checked after the execution of each DMAP instruction. The database directory is updated if this difference is greater than update-time. Update-time and CPU time is accurate to the nearest whole second only.
2. If update-time < 0, then database directory updates are only performed at the end of the run.
3. Defaults for update-time are machine dependent and may be found in the *MSC Nastran Installation and Operations Guide*.
4. Periodic updates of the directory tables to MASTER DBset increases the integrity of the database during system crashes (for example, crashes due to insufficient time or space).
5. Directory updates are performed automatically at various points in the execution of the DMAP in addition to those specified by DBUPDATE. An asterisk appears after the word "BEGN" in the executive summary table whenever an update occurs. See the [Output Description](#) in the *MSC Nastran Reference Guide*. These updates occur whenever a permanent data block, parameter DMAP equivalence, or restart equivalence is broken. Updates also occur upon deletions. Additions to the database do not automatically cause a directory update to take place.
6. This statement is in effect only when INIT MASTER(RAM=r) is being used. INIT MASTER(S) and INIT MASTER(NORAM) disable periodic and automatic updates.
7. Update-time may also be changed with the DMAP instruction PUTSYS(update-time, 128) or the NASTRAN SYSTEM(128)=update-time statement. (The update-time must be a real, single-precision value specified in minutes.)

## DEFINE Parameter Definition

Assigns user-defined keywords (or cellnames) to a NASTRAN system cell. (See the NASTRAN statement for a description of “cellname”). In addition, the DEFINE statement provides a mechanism to set default values for system cells.

### Format:

DEFINE keyword [ =expression ] [ LOCATION=SYSTEM(i) ] [ TYPE=type ]

Describer	Meaning
keyword	User-defined name, 1 through 24 characters in length. The first character must be alphabetic. The following characters can be used for keywords: A through Z, _, and 0 through 9. Any other characters are invalid.
expression	Expression produces a single value from a set of constant and/or variable parameters separated by operators. The value is assigned to the “keyword” and is also used to set the value for the NASTRAN system cell specified by “LOCATION”. TYPE determines both the type of the result and the type conversions that will be applied to the constants and variables within the expression--mixed mode expressions are allowed (see Remark 6.). The parentheses can be used to change the order of precedence. Operations within parentheses are performed first, with the usual order of precedence being maintained within the parentheses. The variable parameters within the expression must be keywords previously defined on a DEFINE statement. The following operations are allowed:

Parameter Type	Operator	Operation
Integer or Real	+	Addition
	-	Subtraction
	*	Multiplication
	/	Division
Logical	+	Bit-wise OR
Logical	-	Bit clear. For example, the result of $a-b$ is equal to the value of $a$ with the bits associated with $b$ set to 0.

Describer	Meaning
SYSTEM(I) type	Specifies the NASTRAN system cell number to be associated with the keyword. The type of expression result, and the type of conversions that will be applied to the constants and variables within the expression. Allowable data types are as follows:

Description	Type
Integer (default)	I
Real	R
Logical	LOGICAL

#### Remarks:

1. If TYPE, LOCATION, and EXPRESSION are omitted, the default data type is Integer and the default value is zero.
2. If EXPRESSION is omitted, an internal default will be assigned to the keyword/cellname based on the LOCATION (See [The NASTRAN Statement](#) for a list of internal default values).
3. A DEFINE statement that specifies a LOCATION is actually setting the default for a NASTRAN system cell, I and therefore it is not necessary to also set the system cell value on a subsequent NASTRAN statement unless the user wishes to override the previous DEFINE statement setting. Also, since more than one DEFINE statement may be present for the same “keyword”, the last specification takes precedence. “Keywords” referenced on a NASTRAN statement, or in an expression on the DEFINE statement, are automatically substituted by the last specification of the “keyword” prior to the current statement being processed.
4. DEFINE statements may also be specified in runtime configuration (RC) files. See the .
5. System cells may also be set with the NASTRAN statement. In addition, they may be set or values returned with the DMAP PUTSYS and GETSYS functions and the PARAM module. See the [MSC Nastran DMAP Programmer's Guide](#).
6. Each operand within the expression will be converted to the result type prior to the arithmetic operation. For example: the statement “DEFINE JJ=2.5 + 3.6 TYPE=I” would result in 2.5 and 3.6 being converted to 2 and 3, respectively, and the result of 5 would be assigned to JJ.

#### Examples:

1. Change the default value for block size:  
`DEFINE BUFFSIZE=4097 LOCATION=SYSTEM(1)`
2. Set the sparse matrix selection to forward-backward substitution only:  
`DEFINE SPARSE=16 LOCATION=SYSTEM(126)`



3. Define the system cell keyword and default value for the maximum output line count and then reset it to another value on a NASTRAN statement. Note: The DEFINE statement would typically be placed in an RC file and the NASTRAN statement would be placed in the File Management Section whenever the user wants to override the DEFINE statement default setting.

```
DEFINE      MAXLINES=999999999  LOCATION=SYSTEM (9)
NASTRAN     MAXLINES=100000
```

4. Define system cells that behave like “toggles,” turning some feature on or off:

```
DEFINE      MESH=2  LOCATION= (31)
DEFINE      NOMESH=0  LOCATION= (31)
NASTRAN     MESH
```

**Note:** Since each subsequent DEFINE statement redefines the default value, the second DEFINE of system cell location 31 sets the default value to 0. A NASTRAN statement can then be inserted in the input file to reset the MESH system cell back to a value of 2. This same technique can be used with any system cell where the user wishes to refer to the system cell keyword and have the system cell set to a previous DEFINE statement default.

5. Invalid usage of the DEFINE and NASTRAN statement:

```
DEFINE      BUFFSIZE=4097
NASTRAN     BUFFSIZE=2048
```

Valid usage:

```
DEFINE      BUFFSIZE=4097  LOCATION=SYSTEM (1)
NASTRAN     BUFFSIZE=2048
```

## **ENDJOB** Terminates Job

Terminates the job at a user-specified location in the FMS Section.

Format:

```
ENDJOB
```

Example:

```
DBDICT  
ENDJOB
```

Remark:

ENDJOB is normally used after a DBDICT statement, or after database initialization.

**EXPAND****Concatenates New DBset Members**

Concatenates additional DBset members on an existing permanent DBset previously defined with an INIT statement.

**Format:**

```
EXPAND dbset-name LOGICAL=( log-namei [(max-sizei)]...)
```

**Example:**

```
ASSIGN          DBMEM02='physical file name'
EXPAND          DBALL LOGICAL=(DBMEM02)
```

This would create and add the DBset member DBMEM02 to the existing DBset DBALL.

Describer	Meaning
dbset-name	The name of a DBset previously defined with an INIT statement.
log-namei	Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement which refers to the physical file name of the DBset member.

Describer	Meaning
max-sizei	Specifies the maximum size in blocks, words or bytes of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the following unit keywords:

Unit Keyword	Storage Unit
W	Words
B	Bytes
KW, K	Kilowords (1024 words)
KB	Kilobytes (1024 bytes)
MW, M	Megawords (1024 <sup>2</sup> words)
MB	Megabytes (1024 <sup>2</sup> bytes)
GW, G	Gigawords (1024 <sup>3</sup> words)
GB	Gigabytes (1024 <sup>3</sup> bytes)
TW, T	Terawords (1024 <sup>4</sup> words)
TB	Terabytes (1024 <sup>4</sup> bytes)

For example, 100MB = 100 megabytes, 1.5GB = 1.2 gigabytes = 1536 megabytes, and 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFFSIZE.

**Remark:**

1. On all computers with dynamic file allocation, the physical filename of a DBset member may be specified on an ASSIGN statement:

ASSIGN log-name='physical filename'

If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in [Database Concepts in the MSC Nastran Reference Guide](#).

## INCLUDE Inserts External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

Format:

```
INCLUDE 'filename'
```

Example:

The following INCLUDE statement is used to obtain the bulk data from another file called MYBULK.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
INCLUDE 'MYBULK.DATA'
ENDDATA
```

Descriptor	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks ( ' ).

Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file:

```
/dir123/dir456/dir789/filename.dat
```

may be included with the following input:

```
INCLUDE '/dir123
/dir456
/dir789/filename.dat'
```

3. See the for more examples.

**INIT**                      Creates a DBset

Creates a temporary or permanent DBset. For the SCRATCH and MASTER DBsets, all or some of their space may be allocated to real memory.

Format 1: Initialize any DBset except MASTER and SCRATCH:

```
INIT DBset-name [LOGICAL=(log-namei(max-sizei),...) BUFFSIZE=b CLUSTER=c]
```

Format 2: Initialize the MASTER DBset:

```
INIT MASTER [ [ ( RAM = r , S ) LOGICAL = (log-name(max-sizei), ...),
               ( NORAM )
             ]
            BUFFSIZE = b CLUSTER = c ]
```

Format 3: Initialize the SCRATCH DBset:

```
INIT SCRATCH [ [ ( MEM = m ) LOGICAL = (log-name(max-sizei), ...),
                ( NOMEM )
              ]
             SCR300 = (log-namei(max-sizei),...) BUFFSIZE = b CLUSTER=c ]
```

Example:

1. Modify the default allocation of the DBALL DBset to 50000 blocks:  
INIT DBALL LOGI=(DBALL(50000))
2. Do not allocate any real memory for the MASTER and SCRATCH DBsets:

```
INIT          MASTER (NORAM)
INIT          SCRATCH (NOMEM)
```

3. Create a new DBset called DBUP with two members DBUP1 and DBUP2:

```
INIT          DBUP LOGI=(DBUP1, DBUP2)
ASSIGN        DBUP1 ='physical filename 1'
ASSIGN        DBUP2 ='physical filename 2'
```

Describer	Meaning
-----------	---------

dbset-name  
MASTER  
SCRATCH

The name of a temporary or permanent DBset.

log-namei

Specifies the logical name of a DBset member. log-namei may also be referenced on an ASSIGN statement, which refers to the physical file name of the DBset member. If no log-namei is specified, then the DBset will have one member and the log-name will be the same as the DBset-name. A maximum of twenty log-names may be specified. For the SCRATCH DBset, see also Remark 8. SCR300 is a special keyword that indicates that the log-names are members reserved for DMAP module internal scratch files.

max-sizei

Specifies the maximum size, in blocks, words, or bytes, of a DBset member. For storage units specified in words or bytes, the size must be followed by one of the following unit keywords:

Unit Keyword	Storage Unit
W	Words
B	Bytes
KW, K	Kilowords (1024 words)
KB	Kilobytes (1024 bytes)
MW, M	Megawords (1024 <sup>2</sup> words)
MB	Megabytes (1024 <sup>2</sup> bytes)
GW, G	Gigawords (1024 <sup>3</sup> words)
GB	Gigabytes (1024 <sup>3</sup> bytes)
TW, T	Terawords (1024 <sup>4</sup> words)
TB	Terabytes (1024 <sup>4</sup> bytes)

For example, 100MB = 100 megabytes, 1.5GB = 1.5 gigabytes = 1536 megabytes, and 2.5M = 2.5 megawords = 2560 kilowords. The size of a block in words is defined by BUFFSIZE. The default for DBALL and SCRATCH may be found in the and ranges from 250,000 blocks to 4,000,000 blocks.

RAM  
NORAM

RAM=r requests that r words of real memory are to be allocated for the MASTER DBset. See the [nast20214 Command](#) in the *MSC Nastran Reference Guide*. The default is RAM or RAM=30000. NORAM or RAM=0 specifies that no real memory is to be allocated.

Describer	Meaning
S	If the primary database is being created in the run, this option requests that all DBsets in the primary database will be automatically deleted at the end of the run. INIT MASTER(S) is equivalent to specifying <code>scr=yes</code> on the “nastran” statement. See <a href="#">Executing MSC Nastran, 1</a> . If the run is a restart, then this option is ignored.
MEM NOMEM	MEM=m specifies that m blocks of real memory are to be allocated for the SCRATCH DBset. See <a href="#">The NASTRAN Statement (Optional)</a> (p. 13) in the <i>MSC Nastran Reference Guide</i> . The default m is machine dependent and may be found in the . NOMEM or MEM=0 requests that no real memory is to be allocated.
BUFSIZE	BUFSIZE=b specifies the number of words per block in the DBset and will override the value specified by the BUFSIZE keyword on the NASTRAN statement. The default for b is obtained from the NASTRAN BUFSIZE statement. See the .
CLUSTER	CLUSTER=c specifies the number of blocks per cluster in the DBset. The default is 1, and any other value is not recommended.

#### Remarks:

1. The INIT statement and its applications are further discussed in [Database Concepts in the MSC Nastran Reference Guide](#).
2. Four DBsets are predefined and automatically allocated by the program. Their DBset-names are MASTER, DBALL, SCRATCH, and OBJSCR, and they are described in [Database Concepts in the MSC Nastran Reference Guide](#).
3. On all computers with dynamic file allocation, the physical filename of a DBset member may specified on an ASSIGN statement:
 

```
ASSIGN log-name='physical filename'
```

 If an ASSIGN statement is not specified for the member, then a name is automatically assigned. The naming convention is described in [Database Concepts in the MSC Nastran Reference Guide](#).
4. It is recommended that there be sufficient physical space to hold a DBset member should it reach its maximum size (max-sizei). The max-sizei may be converted to words by multiplying by b. A summary of space usage and allocation is printed at the end of the execution summary table.
5. In restart runs, the INIT statement is ignored for preexisting permanent DBsets. The INIT statement is intended to be specified only in the run in which the DBset is being created. If more DBset members need to be added to the DBset, then the EXPAND statement is used.
6. If RAM or RAM=r is specified and the run terminates because the computer operating system detects insufficient space or time, or the computer halts due to a power outage or operator interruption, then it may not be possible to restart from the database. See the DBUPDATE FMS statement.



7. BUFFSIZE=b and CLUSTER=c must satisfy the following inequality:

$$b \leq \frac{64000}{c} + 5$$

8. By default, the SCRATCH DBset is divided into two partitions: LOGICAL and SCR300. The LOGICAL partition, log-names after the LOGICAL keyword, are reserved for DMAP scratch data blocks, and the SCR300 partition for DMAP module internal scratch files.
- The maximum total number of log-names for LOGICAL and SCR300 is 20. For example, if LOGICAL has 8 log-names, then SCR300 can have no more than 12 log-names.
  - If NASTRAN SYSTEM(142)=1 is specified, then the SCR300 partition is not created and internal scratch files, along with DMAP scratch data blocks, will reside on the LOGICAL partition. The default is SYSTEM(142)=2.
  - If NASTRAN SYSTEM(151)=1 is specified and the LOGICAL partition has reached its maximum size, then the SCR300 partition will be used. The default is SYSTEM(151)=0.
  - By default, the space specified for the SCR300 partition is released to the operating system after each DMAP module is executed if the module used more than 100 blocks for internal scratch files. If 100 blocks is not a desirable threshold, then it may be changed by specifying NASTRAN SYSTEM(150)=t, where t is the number of blocks for the threshold.
9. BUFFSIZE=b is predefined for DBset-names MSCOBJ, OBJSCR, and USROBJ and may not be changed by BUFFSIZE on this statement or if the NASTRAN BUFFSIZE=bmax statement (see [The NASTRAN Statement \(Optional\)](#) (p. 13) in the *MSC Nastran Reference Guide*). The default for b is recommended for all except very large problems. bmax must reflect the maximum of b specified for all DBsets attached to the run, including the delivery database. See for the defaults of b and bmax.
10. If INIT MASTER(RAM=r) and INIT SCRATCH(MEM=m) are specified, then BUFFSIZE for these DBsets must be the same. If not, a warning message is issued, and the BUFFSIZE for the SCRATCH DBset is reset to that of the MASTER DBset.
11. Only one INIT statement per dbset-name may be specified in the File Management Section.

## MEMLIST Specify Datablocks Eligible for SMEM

Specifies a list of scratch datablocks that may reside in scratch memory (SMEM).

Format:

MEMLIST DATABLK = (DBname1, DBname2, ..., DBnamei)

Example:

MEMLIST DATABLK = (KOO, MOO, KQQ, MQQ)

If generated, datablocks KOO, MOO, KQQ, and MQQ will reside in scratch memory. All other datablocks will be excluded from scratch memory.

Describer	Meaning
DBnamei	Name of a Nastran datablock.

Remarks:

1. Only NDDL and local scratch datablocks may be included in MEMLIST specification.
2. Datablocks specified will reside in SMEM on a first-come, first-served basis.
3. Datablocks not specified by this statement will not reside in SMEM.
4. Database directories for the SCRATCH DBset reside in SMEM and are not affected by any MEMLIST specification.
5. Continuation lines are allowed.
6. Multiple MEMLIST statements are honored.
7. Scratch I/O activity is reported in the .f04 file by including DIAG 42 in the Executive Control Section.

**PROJ** Defines Database Project-Identifier

Defines the current or default project identifier, project-ID.

Format:

PROJ [=] 'project-ID'

Examples:

1. PROJ = 'MY JOB'
2. The following project-ID will be truncated to 40 characters:  
 PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B RUN'  
 and all subsequent restarts must have the statement.  
 PROJ 'CAR MODEL 1999 BODY FRAME SYM - PROTYP B'

Describer	Meaning
project-ID	Project identifier. Must be enclosed in single quotes. (Character string, maximum of 40 characters; Default=blank)

Remarks:

1. There may be only one PROJECT statement in the File Management Section. The PROJECT statement must be specified before all DBCLEAN, DBDICT, RESTART, DBLOCATE, and DBLOAD statements where project-ID is not specified by the user.
2. This statement is optional and specifies that all data blocks and parameters to be stored on or accessed from the database in the current run shall also be identified by project-ID. Therefore, in subsequent runs that may access this data through other FMS statements such as RESTART, the project-ID must be specified.
3. Project-ID is the default on DBCLEAN, DBDICT, and RESTART FMS statements and in the WHERE and CONVERT clause of the DBLOCATE statement.
4. Leading blanks and trailing blanks enclosed within the single quotes are ignored. All other blanks are considered part of the project-ID.
5. Project-ID is saved with only the first 40 characters specified.

## RESTART Reuses Database From a Previous Run

Requests that data stored in a previous run be used in the current run.

Format:

```
RESTART [PROJECT='project' VERSION = [version-ID] [KEEP | LAST | NOKEEP] LOGICAL = dbname]
```

Examples:

1. RESTART VERSION=7

Version number 7 will be retrieved for this run (version 8). At the end of the run, version 7 will be deleted.

2. PROJ='FENDER'  
RESTART

The last version under project-ID FENDER will be used in the current run.

3. ASSIGN RUN1='run1.MASTER'  
RESTART LOGICAL=RUN1

The run1.MASTER and its associated database will be used (read only) for restart purposes.

Describer	Meaning
project-ID	Project identifier. See description of the PROJ FMS statement. Must be enclosed in single right-hand quotation marks (') (Character string, maximum of 40 characters; default is the project-ID specified on the PROJ FMS statement).
version-ID	Version number (Integer > 0).
LAST	Specifies the last version under project-ID.
KEEP	Data stored under VERSION will remain on the database after the run is completed.
NOKEEP	Data stored under VERSION will be deleted from the database after the run is completed.
dbname	Specifies the logical name of an existing MASTER (master directory) DBset to be used for restart purposes. This MASTER and its associated database will be opened in a read-only mode to perform the restart; any new data will be written to the database for the current run. For read-only restart (NOKEEP), the logical name MASTER should not be used and UFM 10626 will be issued.

Remarks:

1. There may only be one RESTART statement in the File Management Section.

2. A new version-ID is automatically assigned whenever a restart is performed.
3. If project-ID or version-ID or both are specified and cannot be found, then a fatal message will be issued.
4. The RESTART statement is required to perform restarts in solution sequences 4, and 101 through 200 and SOL 400.
5. If PROJECT is not specified, then the run will restart from the project-ID specified on the PROJ statement (See Example 2).
6. Databases created in one version typically cannot be directly restarted into a different version. Restrictions are typically documented in the current release guide; however, a DBLOCATE type restart might work.
7. Restarts do not work with DMP. Restarts do not work with ACMS for versions released before 2018.
8. If NASTRAN IFPSTAR=YES (default) and scr=no then the Bulk Data images will be stored on the IFPDAT file; for example, if the input file name is run1 then run1.IFPDAT will be created along with run1.MASTER and run1.DBALL. Please note the following when performing a restart.
  - a. If run1.IFPDAT, run1.MASTER and run1.DBALL are in the same folder then only the MASTER file needs to be specified on an ASSIGN statement. For example, ASSI cold='run1.MASTER' RESTART LOGI=cold
  - b. If run1.IFPDAT is in a different folder than that of run1.MASTER and run1.DBALL (or IFPDAT has been renamed in the same folder) then both the MASTER and IFPDAT must be specified on ASSIGN statements and the IMPORT keyword must be specified on the ASSIGN IFPDB statement:  
 ASSI cold='run1.MASTER'  
 RESTART LOGI=cold  
 ASSI IFPDB='other\_folder/run1.IFPDAT' IMPORT
9. For SOL 400 one must also use [NLRESTART \(Case\)](#).
10. The database is not suitable for restart if any of the following was specified in the cold start. If a restart is attempted then the program will issue User Fatal Message 9061.
  - a. EXTSEOUT(DMIGDB or MATDB or MATRIXDB) without defining DBEXT.
  - b. scr=post
  - c. scr=no and "NASTRAN SYSTEM(316)>0".
11. Restarts of SOL 112 fatigue analyses also require case control FATIGUE(STROUT=-1). Please see [Ap. A: Restarts](#) in *MSC.Nastran Embedded Fatigue User's Guide* for restarts for fatigue analyses.



# 4

## Executive Control Statements

- Key to Descriptions
- Executive Control Statement Descriptions

## Key to Descriptions

Braces { } indicate that a choice of descriptors is mandatory.

A brief sentence about the function of the statement is given.

Brackets [ ] indicate that a choice of descriptors is optional.

**LINK** Link a Main SubDMAP. The default descriptors are shaded.

Links a main subDMAP to form a solution sequence.

**Format:**

```
LINK {subDMAP-name} [SOLO = SOLOUT-DBset EXEOUT = exeout-DBset
INCLUDE = incl-DBset [MAP], SOLNAME = newname]
```

Describers in uppercase letters are keywords that must be specified as shown.

Describers in lower case are variables.

If the descriptors are stacked vertically, then only one may be specified.

**Descriptors:**

*n* The solution number of the member description for the list of valid numbers. (Integer > 0).

*subDMAP-name* The name of a main subDMAP. See the *MD Nastran DMAP Programmer's Guide*.

*solout-DBset* The name of a DBset solution sequence map.

*exeout-DBset* The name of an alternate only the OSCAR map.

*incl-DBset* The name of a DBset. See Remark 2.

*newname* A new name assigned to the OSCAR. The default is

**MAP** Each of the descriptors is discussed briefly. Further details may be discussed under Remarks. Give the name of all the subDMAPs in a sequence.

**NOMAP** Suppresses printing of the link map.

**Remarks:**

- All DBsets specified on this statement must have the same BUFSIZE. See the INIT FMS statement.
- SubDMAP objects are created with the COMPILE statement in the current run or obtained from previous runs. The following order:
  - Objects created with the COMPILE statement in the current run.
  - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.

The remarks are generally arranged in order of importance and indicate such things as the statement's relationship to other statements, restrictions and recommendations on its use, and further details regarding the descriptors.



## Executive Control Section

This section describes the Executive Control statements. These statements select a solution sequence and various diagnostics.

Most Executive Control statements are order independent. The exceptions are the COMPILE, COMPILER, ALTER, ENDALTER, and LINK statements. If used, the LINK statement must appear after all COMPILE statements. The COMPILER statement (or equivalent DIAGs) must appear before all COMPILE statements. The COMPILER statement also sets the defaults for subsequent COMPILE statements.

## Executive Control Statement Summary

The Executive Control statements are summarized as follows:

ALTER	Specifies deletion and/or insertion of the DMAP statements that follow.
APP	Specifies an approach in a solution sequence.
CEND	Designates the end of the Executive Control statements.
COMPILE	Requests compilation of specified subDMAPs or the NDDL file.
COMPILER	Specifies DMAP compilation diagnostics.
DIAG	Requests diagnostic output or modifies operational parameters.
DOMAINSOLVER	Selects domain decomposition solution methods.
ECHO	Controls the echo of Executive Control statements.
ENDALTER	Designates the end of a DMAP sequence headed by an ALTER.
GEOMCHECK	Specifies tolerance values and options for optional finite element geometry tests.
ID	Specifies a comment.
LINK	Requests the link of a main subDMAP.
MALTER	Inserts and/or deletes DMAP statements in solution sequences
MODEL_CHECK	Specifies model checkout run options.
SOL	Requests execution of a solution sequence or DMAP program.
SOL 600,ID	Creates Marc input and optionally executes Marc from SOL 600
SOL 700,ID	Executes MSC Nastran Explicit Nonlinear (SOL 700)
SPARSEsolver	Specifies various options used in sparse solution if equations operations.
TIME	Sets the maximum allowable execution time.

## Executive Control Statement Descriptions

Executive Control statements may be abbreviated down to the first four characters as long as the abbreviation is unique relative to all other statements. Each statement is described as follows:

### Description

A brief sentence about the function of the statement is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the statement line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
    COMPiLE  SEDRCVR  SOUIN=MSCSOU,  
    NOREF    NOLIST
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), its allowable range, and its default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks in the remarks section are generally arranged in order of importance and indicate such things as the Executive Control statement's relationship to other statements, restrictions and recommendations on its use, and further descriptions of the describers.

## \$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

### Format:

\$ followed by any characters out to column 80.

### Example:

```
$ TEST FIXTURE-THIRD MODE
```

### Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the bulk data.

## ALTER Inserts and/or Deletes DMAP Statements

Inserts and/or deletes DMAP statements in a subDMAP.

### Format:

```
ALTER k1 [,k2]
```

or

```
ALTER 'string1' [(occurrence,offset)] ,['string2' [(occurrence,offset) ] ]
```

or

```
ALTER k1 , ['string2' [(occurrence,offset) ] ]
```

or

```
ALTER 'string1' [(occurrence,offset)] , [k2]
```

### Examples:

1. The following alter will insert a MATPRN DMAP statement after the first occurrence of the string 'SDR2' in subDMAP DSASTAT:

```
SOL 101
COMPILE DSASTAT $
ALTER 'SDR2' $
MATPRN OESDS1//$
CEND
```

2. The following alter will delete the second occurrence of the OUTPUT4 DMAP statement in subDMAP DSASTAT and replace it with a MATPRN DMAP statement:

```
SOL 101
COMPILE DSASTAT $
ALTER 'OUTPUT4' (2) , 'OUTPUT4' (2) $
$ OR
$ ALTER 'OUTPUT4' (2) , '' $
MATPRN OESDS1//$
CEND
```

Describer	Meaning
k1	If k2 or 'string2' is not specified, the subsequent DMAP statements will be inserted after either the statement number k1 or the 'string1', [(occurrence,offset)] reference point.
k1, k2	DMAP statements numbered k1 through k2 will be deleted and may be replaced with subsequent DMAP statements.
'string1'	if 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.

Describer	Meaning
'string1','string2'	DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequent DMAP statements.
occurrence	This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP (Integer > 0; Default = 1).
offset	This flag indicates the offset from the reference DMAP statement. Depending on the sign the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement (Integer; Default = 0).

**Remarks:**

1. The ALTER statement must be used in conjunction with the COMPILE Executive Control statement. Note: ALTER statements cannot be used in conjunction with an MALTER statement, and therefore should never immediately follow this statement.
2. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.
3. The ALTERs can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
  - K2 or 'string2'(occurrence, offset) references must refer to a DMAP statement number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single ALTER statement.
  - K1 or 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another ALTER that references the same subDMAP.
4. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement; i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string—all blanks and comments either embedded or immediately preceding the DMAP statement, will be retained. However, comments are ignored for the following type of alter:
 

```
alter ^^ *gp0'
```
5. Within a SUBDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP—not at the current position of the last string match.
6. The special characters (metacharacters) used for string searching are described in Remark 9. The characters <, >, and \$, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string
 

```
IF (DDRMM >=-1)
```

the command is

```
ALTER 'IF (DDRMM \>=-1)' $
```
7. The ALTER statement must not exceed 72 characters (no continuations are allowed).



8. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (''). For example, the alter statement

```
ALTER 'string1' (r1,01),''
```

is equivalent to

```
ALTER 'string1' (r1,01),'string1' (r1,01)
```

The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

As another example, the alter statement

```
ALTER 'string1' (r1,01),'' (r2,02)
```

is equivalent to

```
ALTER 'string1' (r1,01),'string1' (r2,02)
```

9. Metacharacters\*:

- . Matches any single character except newline.
  - \* Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".\*" means "match any number of characters".
  - [...] Matches any one of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character not in the list.
  - ^ or ! Requires that the following regular expression be found at the beginning of the line. Note that these metacharacters may lead to User Fatal Message 802 if the preceding line is a comment.
  - \$ Requires that the preceding regular expression be found at the end of the line.
  - \ Treats the following special character as an ordinary character. For example, "\." stands for a period and "\\*" for an asterisk. Also, to search for a tic ('), the search string must be single quotes.
  - ' Marks the beginning and end of a pattern to be matched.
- Note: Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ ->! and [ ] -><>) if portability is required. However, all the preceding characters are recognized by MSC Nastran.

10. If a string-based alter uses the "!" in the expression (find occurrence at the beginning of line), it is possible MSC Nastran will fail with User Fatal Message 802.

## APP Specifies Solution Sequence Approach

Selects heat transfer analysis in the linear static solution sequence SOL 101, or a coupled analysis combining heat transfer and structural analysis in SOL 153.

Format:

APP approach

Example:

The following requests a heat transfer rather than a structural analysis in SOL 101.

```
SOL 101
APP HEAT
```

Describer	Meaning
approach	Specifies one of the following:
	HEAT                      Indicates that heat transfer is to be performed in SOL 101.
	COUPLED                Indicates that a coupled analysis combining heat transfer and structural analysis is to be performed in SOL 153.

Remarks:

1. The APP statement is optional.
2. The APP HEAT statement applies only to linear static SOL 101. The APP HEAT statement is not required in SOLs 153 and 159, or in SOL 101 if PARAM,HEATSTAT,YES is specified.
3. The NASTRAN HEAT=1 statement is an alternate specification of APP HEAT. See [Executing MSC Nastran, 1](#).

**CEND**                      End of Executive Control Section Delimiter

Designates the end of the Executive Control Section.

Format:

CEND

Remark:

1. CEND is an optional statement. If CEND is not specified, then the program will automatically insert one.



## COMPILE Compiles DMAP Statements

Requests the compilation of a subDMAP, subDMAP alter, or NDDL sequence.

Format 1: Compiles a subDMAP or subDMAP alter sequence

COMPILE 

SUBDMAP
DMAP

 subDMAP-name [SOUIN = souin-DBset SOUOUT = souout-DBset,

OBJOUT = objout-DBset 

LIST
NOLIST

REF
NOREF

DECK
NODECK

 ]

Format 2: Compiles an NDDL sequence

COMPILE NDDL = nddl-name { { SOUIN = souin-dbset } { SOUOUT = souout-dbset } { LIST } { NOLIST } { REF } { NOREF } { DECK } { NODECK } }

Examples:

1. The following compiles an alter in subDMAP PHASEIDR:

```
COMPILE PHASE1DR
ALTER 'CALL PHASE1A'
CEND
```

2. The following compiles a subDMAP called MYDMAP. (SUBDMAP and END are DMAP statements; see the [MSC Nastran DMAP Programmer's Guide](#)):

```
COMPILE MYDMAP LIST REF
SUBDMAP MYDMAP $
.
.
.
END $
CEND
```

3. The following obtains a listing of the NDDL:

```
ACQUIRE NDDL
COMPILE NDDL=NDDL LIST
CEND
```

Describer	Meaning
subDMAP-name	The name of a subDMAP sequence. SubDMAP-name must be 1 to 8 alphanumeric characters in length and the first character must be alphabetic. The keywords DMAP and SUBDMAP are optional and do not have to be specified.
nddl-name	The name of an NDDL sequence (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The keyword NDDL must be specified.
souin-DBset	The name of a DBset from which the subDMAP or NDDL source statements will be retrieved (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is MSCSOU if the next statement is not a subDMAP statement.
souout-DBset	The name of a DBset on which the subDMAP or NDDL source statements will be stored (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is the SCRATCH DBset.
objout-DBset	The name of a DBset on which the subDMAP object code will be stored (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic). The default is the OBJSCR DBset.
LIST,NOLIST	LIST requests a compiled listing of the subDMAP or NDDL sequence. NOLIST suppresses the listing. NOLIST is the default.
REF,NOREF	REF requests a compiled cross reference of the subDMAP or NDDL sequence. NOREF suppresses the cross reference. NOREF is the default.
DECK,NODECK	DECK requests the subDMAP or NDDL source statements to be written to the PUNCH file. NODECK suppresses the writing to the PUNCH file. NODECK is the default.

#### Remarks:

1. SubDMAP names for MSC Nastran solution sequences are given in the SOL statement description. The “COMPILER LIST REF” statement may be used to determine the appropriate subDMAP-name.
2. If a subDMAP is being compiled and SOUIN=souin-DBset is specified, then an ALTER Executive Control statement, or an INCLUDE statement which contains an ALTER statement as the first non-comment line, must appear immediately after this statement. If not, then the SUBDMAP DMAP statement must appear immediately after this statement. See the [MSC Nastran DMAP Programmer's Guide](#).
3. Starting in MSC/Nastran Version 69, DBsets USRSOU and USROBJ were no longer automatically created. They must be initialized by the INIT FMS statement and then may be specified for souin-dbset (or souout-dbset) and objout-dbset, respectively. They may be used to store the subDMAP source statements and object code on the primary database for reexecution in a subsequent run. Consider the following example:

In the first run, the following COMPILER statement compiles and stores a subDMAP called MYDMAP:

```
COMPILE MYDMAP SOUOUT=USRSOU OBJOUT=USROBJ
SUBDMAP MYDMAP $
.
.
END $
CEND
```

In the second run, the SOL statement is used to execute the MYDMAP stored in the previous run. The LINK statement is required to retrieve the object code from the USROBJ DBset:

```
SOL MYDMAP
LINK MYDMAP INCL=USROBJ
CEND
```

In the third run, the COMPILE statement is used to alter MYDMAP and execute:

```
SOL MYDMAP
COMPILE MYDMAP SOUIN=USRSOU
ALTER...
.
.
.
CEND
```

4. If SOUOUT or OBJOUT is specified and a subDMAP with the same name as subDMAP-name already exists on the database, then its source statements or object code will be replaced.
5. A COMPILE statement is required for each subDMAP to be compiled. If two or more COMPILE statements reference the same subDMAP name, then only the last is used in the linking of the object code. If the COMPILE statement is being used only to alter a subDMAP and two or more COMPILE statements reference the same subDMAP name, then the multiple alters are assembled and the subDMAP is compiled only once.
6. Only one COMPILE statement for an NDDL sequence may be specified in the input file.
  - SOUIN=souin-DBset requests only a compilation of the NDDL sequence stored on souin-DBset for purposes of obtaining a listing or a cross reference, and it cannot be modified with the ALTER statement. See Remark 3. COMPILE NDDL=NDDL SOUIN=MSCSOU LIST requests a listing of the MSC Nastran NDDL sequence. The ACQUIRE FMS statement or the SOL statement must be specified in order to attach the corresponding delivery database.
  - To alter the MSC Nastran NDDL sequences, the entire modified NDDL sequence is included after the COMPILE statement, and SOUIN=souin-DBset is not specified.
  - SOUOUT=souout-DBset requests the storage of the NDDL source statements on the souout-DBset, and may not be specified with SOUIN=souin-DBset.
7. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, and NODECK. In other words, if LIST or NOLIST, REF or NOREF, or DECK or NODECK is not specified, then the corresponding option on the COMPILER statement will be used. In the following example, REF on the COMPILER statement will override the default of NOREF on the COMPILE statement:

```
COMPILER REF
```

COMPILE MYDMAP

8. MSCSOU and MSCOBJ, specified with SOUOUT and OBJOUT, are special DBsets similar to USRSOU and USROBJ except that they are used in the creation or modification of a delivery database. For an example application, see the .

## COMPILER DMAP Compiler Output Options

Requests compilation of a DMAP sequence and/or overrides defaults on the COMPILE statement.

Format:

```
COMPILER[=] [ LIST ] [ DECK ] [ REF ] [ GO ] [ SORT ]
             [ NOLIST ] [ NODECK ] [ NOREF ] [ NOGO ] [ NOSORT ]
```

Example:

```
COMPILER=LIST
```

Describer	Meaning
LIST, NOLIST	LIST requests the compilation listing of the solution sequence. NOLIST suppresses the listing.
DECK, NODECK	DECK requests that the DMAP source statements of the solution sequence be written to the PUNCH file. NODECK suppresses the DECK option.
REF, NOREF	REF requests a compilation cross reference. NOREF suppresses a compilation cross reference.
GO, NOGO	GO requests the execution of the solution sequence following compilation. NOGO requests termination following compilation.
SORT, NOSORT	SORT compiles subDMAPs in alphabetical order. NOSORT compiles subDMAPs in calling sequence order.

Remarks:

1. REF is equivalent to DIAG 4. LIST is equivalent to DIAG 14. DECK is equivalent to DIAG 17.
2. NOGO is an alternative to NOEXE on the SOL statement.
3. This statement provides a means of obtaining a compilation or source listing, or both, of a complete solution sequence, including all the component subDMAPs.
4. See the COMPILE statement to compile a single subDMAP.
5. This statement also requests the automatic link of the solution sequence. Therefore, all objects must be created in the current run or obtained from the DBset such as USROBJ. See the COMPILE statement for how to create and store objects.
6. The COMPILER statement may be used to override the defaults of NOLIST, NOREF, NODECK on the COMPILE entry when they are not explicitly specified. However, COMPILER LIST produces a list of the entire solution sequence. If a listing of only specific subdmeps is desired, then COMPILER LIST should not be specified and the LIST request should be made on the COMPILE entry.

```
COMPILER REF
COMPILER MYDMAP
```

**DIAG****Requests Diagnostic Output**

Requests diagnostic output or special options.

Format:

DIAG [=] k1[k2, ..., kn]

Examples:

DIAG 8, 53

or

DIAG 8  
DIAG 53

Describer	Meaning
ki	A list separated by commas and/or spaces of desired diagnostics.

Remarks:

1. The DIAG statement is optional.
2. Multiple DIAG statements are allowed.
3. The following table lists the possible values for ki and their corresponding actions:

k=1	Dumps memory when a nonpreface fatal message is generated.
k=2	Prints database directory information before and after each DMAP statement. Prints bufferpooling information.
k=3	Prints “DATABASE USAGE STATISTICS” after execution of each functional module. This message is the same as the output that appears after the run terminates. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .
k=4	Prints cross-reference tables for compiled sequences. Equivalent to the COMPILER REF statement.
k=5	Prints the BEGIN time on the operator’s console for each functional module. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .
k=6	Prints the END time for each functional module in the log file or day file, and on the operator’s console. Modules that consume less time than the threshold set by SYSTEM(20) do not create a message. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .
k=7	Prints eigenvalue extraction diagnostics for the complex determinate method.
k=8	Prints matrix trailers as the matrices are generated in the execution summary table. See the <a href="#">Output Description in the MSC Nastran Reference Guide</a> .

- k=9 Prints a message in the .f04 file when EQUIV and EQUIVX perform a successful equivalence; in other words, both the input and output exists.
- k=10 Selects an alternate option for averaging nonlinear loading (NOLINi Bulk Data entry) in linear transient analysis. Replaces  $N_{n+1}$  with  $(N_{n+1} + N_n + N_{n-1})/3$ .
- k=11 DBLOAD, DBUNLOAD, and DBLOCATE diagnostics.
- k=12 Prints eigenvalue extraction diagnostics for complex inverse power and complex Lanczos methods.
- k=13 Prints the open core length (the value of REAL). See the [Output Description in the MSC Nastran Reference Guide](#).
- k=14 Prints solution sequence. Equivalent to the COMPILER LIST statement.
- k=15 Prints table trailers.
- k=16 Traces real inverse power eigenvalue extraction operations
- k=17 Punches solution sequences. Equivalent to the COMPILER DECK statement.
- k=18 In aeroelastic analysis, prints internal grid points specified on SET2 Bulk Data entries.
- k=19 Prints data for MPYAD and FBS method selection in the execution summary table.
- k=20 Similar to DIAG 2 except the output appears in the execution summary table and has a briefer and more user-friendly format. However, the .f04 file will be quite large if DIAG 20 is specified with an MSC Nastran solution sequence. A DMAP alter with DIAGON(20) and DIAGOFF(20) is recommended. DIAG 20 also prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics. See the [MSC Nastran DMAP Programmer's Guide](#).
- k=21 Prints diagnostics of DBDICT and DBENTRY table.
- k=22 EQUIV and EQUIVX module diagnostics.
- k=23 Not used.
- k=24 Prints files that are left open at the end of a module execution. Also prints DBVIEW diagnostics.
- k=25 Outputs internal plot diagnostics.
- k=27 Prints Input File Processor (IFP) table. See the *MSC Nastran Programmer's Manual*, Section 4.5.9.
- k=28 Punches the link specification table (XBSBD). The Bulk Data and Case Control Sections are ignored, and no analysis is performed.
- k=29 Process link specification table update. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.

- k=30 In link 1, punches the XSEMii data (i.e., sets ii via DIAG 1 through 15). The Bulk Data and Case Control Sections are ignored, and no analysis is performed. After link 1, this turns on BUG output. Used also by MATPRN module. See also Remark 5 on the [TSTEP, 3282](#) Bulk Data entry.
- k=31 Prints link specification table and module properties list (MPL) data. The Bulk Data and Case Control Sections are ignored, and no analysis is performed.
- k=32 Prints diagnostics for XSTORE and PVA expansion.
- k=33 Not used.
- k=34 Turns off plot line optimization.
- k=35 Prints diagnostics for 2-D slideline contact analysis in SOLs 106 and 129.
- k=37 Disables the superelement congruence test option and ignores User Fatal Messages 4277 and 4278. A better alternative is available with PARAM,CONFAC. See [Parameters, 793](#).
- k=38 Prints material angles for CQUAD4, CQUAD8, CTRIA3, and CTRIA6 elements. The angle is printed only for elements that use the MCID option of the connection entry.
- k=39 Traces module FA1 operations and aerodynamic splining in SOLs 145 and 146.
- k=41 Traces GINO OPEN/CLOSE operations.
- k=42 Prints output on .f04 file the usage statistic for datablock defined in the FMS command, MEMLIST.
- k=43 Print output for Buffer Pool Diagnostics.
- k=44 Prints a mini-dump for fatal errors and suppresses user message exit.
- k=45 Prints the same database directory information as DIAG 2 except that it prints only after each DMAP statement.
- k=46 Used by MSC Nastran development for GINO printout.
- k=47 Prints DBMGR, DBFETCH, and DBSTORE subDMAP diagnostics.
- k=48 Used by MSC Nastran development for GINO printout.
- k=49 DIAG 49 is obsolete and should not be used. The utility f04rprt should be used to summarize the .f04 execution summary instead.
- k=50 Traces the nonlinear solution in SOLs 106, 129, 153, and 159. Prints subcase status; echoes NLPARM, NLPCI, and entry fields; and prints initial arc-length. Prints iteration summary only in SOLs 129, and 159.  
In static aeroelastic analysis (SOL 144), prints transformation information associated with the generation of the DJX matrix in the ADG module and intermediate solutions information in the ASG module.
- k=51 Prints intermediate displacement, load error vectors, and additional iteration information helpful in debugging in SOLs 106, 129, 153, and 159.
- k=52 Disables the printing of errors at each time step in SOLs 129 and 159.



- k=53 MESSAGE module output will also be printed in the execution summary table. See the [Output Description in the MSC Nastran Reference Guide](#).
- k=54 Linker debug print.
- k=55 Performance timing.
- k=56 Extended print of execution summary table (prints all DMAP statements and RESTART deletions). See the [Output Description in the MSC Nastran Reference Guide](#).
- k=57 Executive table (XDIRLD) performance timing and last-time-used (LTU) diagnostics.
- k=58 Data block deletion debug and timing constants echo.
- k=59 Buffpool debug printout.
- k=60 Prints diagnostics for data block cleanup at the end of each module execution in subroutines DBCLN, DBEADD, and DBERPL.
- k=61 GINO block allocator diagnostics.
- k=62 GINO block manager diagnostics.
- k=63 Prints each item checked by the RESTART module and its NDDL description.
- k=64 Requests upward compatibility DMAP conversion from Version 65 only. Ignored in Version 70.5 and later systems.

## DOMAINSOLVER

## Domain Decomposition Solution Method

Selects domain decomposition solution methods.

Format:

$$DOMAINSOLVER \begin{bmatrix} STAT \\ MODES \\ FREQ \\ ACMS \\ DSA \\ NLSOLV \end{bmatrix} \left[ \left[ PARTOPT = \begin{bmatrix} DOF \\ GRID \\ FREQ \\ ELEM \end{bmatrix} \right] \right] NUMDOM = int$$

$$UPFACT = real, PRINT = \begin{bmatrix} YES \\ NO \end{bmatrix}, NCLUST = int, CLUSTSZ = int, COMPMETH = \begin{bmatrix} SUPER \\ GRID \\ NONE \end{bmatrix},$$

$$MODEL = \begin{bmatrix} FLUID \\ STRUCT \\ BOTH \end{bmatrix}, RUNOPT = \begin{bmatrix} PARCHILD \\ MULTIPAR \end{bmatrix}, PARTMETH = \begin{bmatrix} BEND \\ MSCMLV \\ METISG \\ METISO \end{bmatrix}, TIPSIZE = int,$$

$$VERSION = \begin{bmatrix} NEW \\ OLD \end{bmatrix}, SCHED = \begin{bmatrix} STATIC \\ DYNAMIC \end{bmatrix}, GRPSIZ = \begin{bmatrix} 0 \\ N \\ -1 \end{bmatrix}$$

MINDOF=int, XFACT=real, XFACTX=real

Examples:

```
DOMAINSOLVER STAT (PARTOPT=DOF)
DOMAINSOLVER ACMS (UPFACT=3.0, NUMDOM=128)
DOMAINSOLVER DSA
DOMAINSOLVER STAT, NLSOLV
DOMAINSOLVER ACMS (PARTMETH=BEND, NUMDOM=128)
DOMAINSOLVER ACMS (PARTMETH=METISO, TIPSIZE=500)
DOMAINSOLVER STAT (PARTMETH=MSCMLV)
DOMAINSOLVER ACMS (VERSION=NEW) - default
DOMAINSOLVER ACMS (VERSION=OLD)
DOMAINSOLVER ACMS (MODEL=STRUCT, UPFACT=5.0)
```

The keywords function as follows:

STAT	Linear statics.
MODES	Normal modes.
FREQ	Frequency response.
ACMS	Automated component modal synthesis.
DSA	Design sensitivity analysis. (See Remark 2.)
NLSOLV	Nonlinear analysis

The descriptions of the parameters are as follows:

PARTOPT	Partitioning option. Selects which domain is to be decomposed.
DOF	Degree of freedom domain.
GRID	Grid point (geometric) domain.
FREQ	Frequency domain.
ELEM	Finite element domain.

The default is dependent upon solution sequence. See [Table 4-1](#) for further descriptions.

Note that ACMS (VERSION=NEW) requires PARTOPT=DOF. If it is not set by the user, ACMS will use it automatically. However, if the user specifies a different value for PARTOPT, for example, ACMS (VERSION=NEW, PARTOPT=GRID), then MSC Nastran will execute old ACMS rather than new ACMS.

NUMDOM (ACMS)	Selects the number of domains as follows: If NUMDOM = 0 or 1, then the model will not be split. Default depends on the model size and the value of PARTOPT.
------------------	--

NUMDOM will be ignored for ACMS (VERSION=NEW); instead of the number of domains, new ACMS uses the size of the tip domains being specified via TIPSIZE (see below).

NUMDOM (STAT)	Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (equal to the number of processors used for the run). The model will be divided into NUMDOM domains in either the geometric (grid-based) or DOF domains, depending on the value of PARTOPT.
NUMDOM (MODES)	Default = <i>dmp</i> ; if NUMDOM has any other value, it will be reset to <i>dmp</i> (equal to the number of processors used for the run). The model will be divided into NUMDOM domains in either the frequency, geometric (grid-based) or DOF domains, depending on the value of PARTOPT.
NUMDOM (FREQ)	Default = <i>dmp</i> ; if NUMDOM has any other value, it will automatically be set to <i>dmp</i> (equal to the number of processors used for the run). The frequency range will be divided into NUMDOM regions which are then solved independently.

UPFACT (ACMS)	By default, the frequency range used for upstream component modes is two times larger than the desired range on the EIGR/L entry. To modify this factor, specify the UPFACT parameter (Real; Default=2.0 for STRUCT and 5.0 for FLUID).
PRINT (ACMS)	Controls intermediate print of upstream and data recovery processing in .f06 and .f04 files. Default='NO'. If PRINT=NO and an error occurs upstream, the intermediate output is placed in a separate output file named "jid.acms_out" for examination.
NCLUST (MODES)	Specifies the number of frequency segments for hierarchic parallel Lanczos. The frequency range is divided into NLCUST segments and, if PARTOPT=DOF, the stiffness and mass matrices are partitioned into dmp/NCLUST matrix domains. If PARTOPT = GRID, or if PARTOPT is not specified, the model geometry is partitioned into dmp/NCLUST domains.
CLUSTSZ (MODES)	Specifies the number of matrix or geometric domains for hierarchic parallel Lanczos. If PARTOPT=DOF, the stiffness and mass matrices are partitioned into CLUSTSZ matrix domains. If PARTOPT = GRID, or if PARTOPT is not specified, the model geometry is partitioned into CLUSTSZ domains. In either case, the frequency range is divided into dmp/CLUSTSZ frequency segments.
MODEL (ACMS)	Default=BOTH. Model specification to which ACMS parameters are applied. MODEL can be STRUCT or FLUID, or BOTH. For MODEL=BOTH (default), the specified options apply to both structure and fluid models
MINDOF (ACMS)	Default for STRUCT is 65. Default for FLUID is 250,000. MINDOF is the minimum number of degrees of freedom below which ACMS will not execute. If the number of DOF in the structural model is less than MINDOF (MODEL=STRUCT), a Fatal error is issued. If the number of fluid DOF is less than MINDOF. (MODEL=FLUID), a Warning message is issued, and the fluid eigensolution is computed with Lanczos. PARAM,MDMINDOF applies only to the structural model and will override this DOMAINSOLVER entry. PARAM,MDMINDOF will be deprecated in a future release.
XFACT (ACMS)	Default=0.5 for both STRUCT and FLUID. XFACT is used in the ACMS1 module to determine a threshold for reorthogonalization of component eigenvectors. Valid values are from 0.1 to 1.0. This option should be used only for very special conditions.
XFACTX (ACMS)	Default=1.0 for both STRUCT and FLUID. XFACTX is used in the ACMS1 module to define the maximum frequency range for internal reorthogonalization operations. Valid values are from 0.1 to 1.0. This options should be used only for very special conditions.
COMP METH	Specifies the compression method used before domain decomposition. COMP METH is considered only for keywords STAT, MODES and ACMS with PARTOPT = DOF, for all other settings of the keyword and parameter PARTOPT, COMP METH will be ignored. The following are valid options for COMP METH:
SUPER	Specifies the SUPER compression method. (Default)

	GRID	Specifies the GRID compression method.
	NONE	Specifies that no compression should be done.
RUNOPT		Specifies the DMP run option. The following are valid options for RUNOPT:
	PARCHILD	Specifies "parent-child" DMP execution mode. Only the Parent DMP process executes the full Nastran solution. This is optimal for single-host DMP execution.
	MULTIPAR	Specifies "multiple-parent" DMP execution mode. All DMP processes execute the full Nastran solution. This is optimal for multiple-host DMP execution.
		Nastran detects single- and multiple-host DMP execution and sets the default run option accordingly.
PARTMETH		Partitioning option (character string). PARTMETH selects which method is used for domain decomposition.
	BEND	Specifies the Extreme/BEND partitioning method
	MSCMLV	Specifies the MSCMLV partitioning method
	METISG	Specifies the METIS partitioning method based on number of tip domains requested (see NUMDOM above)
	METISO	Specifies the METIS partitioning method based on the desired tip size (see TIPSIZE below). Available for ACMS only.
		<b>Note:</b> NUMDOM (see above) will be ignored for PARTMETH=METISO.
		<b>Note:</b> ACMS (VERSION=NEW) requires PARTMETH=METISO. If this is not set by the user, ACMS will use it automatically. If the user sets a different value for PARTMETH when ACMS (VERSION=NEW) is also specified, then the MSC Nastran will automatically re-set PARTMETH to METISO and run with new ACMS.
TIPSIZE		Specifies a desired number of nodes in the tip domains (integer). ACMS only. This is used only when PARTMETH=METISO; otherwise, it is ignored.
	Default:	1000 (ACMS Version=OLD); 200 (ACMS Version=NEW)
VERSION		Selects new ACMS (MSC Nastran Version 2017 +) or old ACMS (prior to MSC Nastran Version 2017). The default value of VERSION is 'NEW'.
SCHED		For ACMS parallel scheduling with VERSION=NEW. Default is DYNAMIC, which is generally more efficient. Select STATIC for static parallel scheduling. See Remark 7.

GRPSIZ Specifies parallel group size for ACMS Dynamic scheduling. Integer:

- 0 Default
- N Set parallel group size to “N” SMP threads.
- 1 Use single parallel group; number of threads in the group is NSMP (Number of SMP).

Table 4-1 Analysis Types vs Supported Partitioning Methods

Solution Sequence	DMP Method	Partitioning Methods Available				
		GRID	DOF	FREQ	DOF+FREQ	ELEM
101	STAT	*	*			
103	MODES	*	*	*		
	ACMS(OLD)	*	*			
	ACMS(NEW)		*			
108	FREQ			*		
110	MODES			*		
	ACMS		*			
111	MODES		*	*		
	ACMS(OLD)	*	*	*	*	
	ACMS(NEW)		*			
	FREQ			*		
112	MODES		*			
	ACMS		*			
200	MODES		*	*	*	
	ACMS		*	*	*	
	FREQ			*		
	DSA					
400	STAT		*			
	NLSOLV					*
	ACMS		*			

\* - means "supported".

The DOMAINSOLVER command is optional. If “dmp=” is specified on the command without a DOMAINSOLVER command in the Executive Control Section, the actions shown in [Table 4-2](#) will result based on solution sequence.

Table 4-2      **DOMAINSOLVER Defaults with “dmp=” on command line**

Solution Sequence	Default DOMAINSOLVER Options	
	DMP Method	Partitioning Option
101	STAT	DOF
103	MODES	DOF
108	FREQ	FREQ
110	MODES	FREQ
111	MODES	FREQ
	FREQ	FREQ
112	MODES	FREQ
200	MODES	DOF
	FREQ	FREQ
400	STAT	DOF
	NLSOLV	ELEM

Note that DOMAINSOLVER ACMS is not the default for the relevant solution sequences, but it is the recommended option for larger models for SOL 103, 111, 112, and 200 to obtain the best performance.

**Remarks:**

1. Grid Point Weight Generator output selected by PARAM,GRDPNT or the WEIGHTCHECK Case Control command is not available when PARTOPT=GRID.
2. In SOL 200, design sensitivity calculations may be performed in a distributed parallel environment in SOL 200 with the DSA keyword. It is a coarse parallel implementation that divides the sensitivity task across a number of processors so that each processes a subset of the total number of design variables. Following the sensitivity analysis and before optimization, the separate sensitivity data are appended into a global sensitivity set. Also,
  - a. The dmp=n keyword must be specified on the Nastran submittal command where n is the number of available processors.
  - b. The ACMS keyword may also be specified or modified along with DSA. For example,  
DOMAINSOLVER DSA, ACMS
3. In SOL 400, the STAT method is used for DMP parallelism of matrix factorization and the NLSOLV method is used for DMP parallelism of the NLEMG process. The NLSOLV method is intended for nonlinear analysis using advanced nonlinear methods. To only use the STAT method or to change its option to use, for example, a different compression method, specify the following DOMAINSOLVER command:

DOMAINSOLVER STAT(COMPMETH=GRID)



Note that if STAT is changed as specified above, then the DMP parallelism for NLEMG will be turned off. If you want to change the STAT as described above and also keep NLSOLV for NLEMG parallelism, then use the following DOMAINSOLVER command:

```
DOMAINSOLVER NLSOLV,STAT(COMPETH=GRID)
```

Note that this can equivalently be written as follows:

```
DOMAINSOLVER NLSOLV
```

```
DOMAINSOLVER STAT(COMPETH=GRID)
```

Lastly, to only use the NLSOLV method for NLEMG parallelism, specify the following option:

```
DOMAINSOLVER NLSOLV
```

4. In SOL 400, the DOMAINSOLVER NLSOLV (RUNOPT=MULTIPAR) is not supported in combination with Intel MKL Pardiso Solver option, i.e., SPARSE\_SOLVER NLSOLV(FACTMETH=PRDLDL).
5. In SOL 111, the MODES or ACMS option is used for DMP parallelism of the modal calculations and the FREQ option is used for the DMP parallelism of the frequency response calculations. The DOMAINSOLVER FREQ is always turned on for SOL 111. To use the ACMS option for the modal calculation, use the following DOMAINSOLVER command:

```
DOMAINSOLVER ACMS
```

Furthermore, if the MODES option needs to be changed for SOL 111, then the correct approach is to include the following DOMAINSOLVER option:

```
DOMAINSOLVER MODES(PARTOPT=GRID).
```

6. Residual vectors are not available when PARTOPT=GRID.
7. Dynamic parallel scheduling is available for ACMS and VERSION=NEW. Dynamic scheduling provides better parallel speedup for larger numbers of threads. For NSMP less than or equal to 2, static scheduling is used regardless of the setting of SCHED.

## ECHO

### Controls Printed Echo

Controls the echo (printout) of the Executive Control Section.

#### Formats:

ECHOOFF  
ECHOON

#### Remarks:

1. The ECHO statement is optional.
2. ECHOOFF suppresses the echo of subsequent Executive Control statements. ECHOON reactivates the echo after an ECHOOFF statement.

**ENDALTER**      End of DMAP Alter

Designates the end of an alter.

Format:

ENDALTER

Remark:

1. The ENDALTER statement is required when using an alter unless the alter package ends with a CEND, COMPILE, or LINK statement.

## GEOMCHECK

### Specifies Geometry Check Options

Specifies tolerance values and options for optional finite element geometry tests.

Format:

GEOMCHECK test\_keyword [= tol\_value], [MSGLIMIT = n], [MSGTYPE = INFORM],  
FATAL  
WARN

[SUMMARY], [ADVNLLELM], [NONE]

Examples:

1. Set the tolerance for the CQUAD4 element skew angle test to 15.0 degrees and limit messages to 50:  
 GEOMCHECK Q4\_SKEW=15.0,MSGLIMIT=50
2. Limit messages to 500 for each element type:  
 GEOMCHECK MSGLIMIT=500
3. Set the message type to fatal for CQUAD4 element taper tests:  
 GEOMCHECK Q4\_TAPER,MSGTYPE=FATAL
4. Request summary table output only using default tolerance values:  
 GEOMCHECK SUMMARY
5. Request advanced and conventional element geometry check output using default tolerance values:  
 GEOMCHECK ADVNLLELM

Describer	Meaning
test_keyword	A keyword associated with the particular element geometry test. See Remark 2. for a list of acceptable selections.
tol_value	Tolerance value to be used for the specified test. See Remark 2. for default values of the test tolerances.
n	The minimum number of messages that will be produced. The default is 100 messages for each element type. See Remark 3.
FATAL	Geometry tests that exceed tolerance values produce fatal messages. See Remark 4.
INFORM	Geometry tests that exceed tolerance values produce informative messages. See Remark 4.
WARN	Geometry tests that exceed tolerance values produce warning messages. See Remark 4.
SUMMARY	A summary table of the geometry tests performed is produced. No individual element information messages are output.
ADVNLLELM	Geometry check performed on Advanced and conventional elements. If no ADVNLLELM, geometry check only performed on conventional element.
NONE	None of the optional element geometry tests will be performed.

Remarks:

1. The GEOMCHECK statement controls the number and severity of certain informational and warning messages produced by element matrix generation geometry checking operations. Controls are currently available for the CQUAD4, CQUADR, CTRIA3, CTRIAR, CHEXA, CPENTA, CTETRA, CPYRAM, CBAR, and CBEAM elements only. Multiple GEOMCHECK statement may be present. Continuations are acceptable.
2. The following table summarizes the acceptable specifications for test\_keyword.

Name	Value Type	Default	Comment
Q4_SKEW	Real $\geq 0.0$	30.0	Skew angle in degrees.
Q4_TAPER	Real $\geq 0.0$	0.50	Taper ratio.
Q4_WARP	Real $\geq 0.0$	0.05	Surface warping factor.
Q4_IAMIN	Real $\geq 0.0$	30.0	Minimum interior angle in degrees.
Q4_IAMAX	Real $\geq 0.0$	150.0	Maximum interior angle in degrees.
T3_SKEW	Real $\geq 0.0$	10.0	Skew angle in degrees.
T3_IAMAX	Real $\geq 0.0$	160.0	Maximum interior angle in degrees.
TET_AR	Real $\geq 0.0$	100.0	Longest edge to shortest edge aspect ratio.
TET_EPLR	Real $\geq 0.0$	0.50	Edge point length ratio.
TET_EPIA	Real $\geq 0.0$	150.0	Edge point included angle in degrees.
TET_DETJ	Real	0.0	J  minimum value.
TET_DETG	Real	0.0	J  minimum value at vertex point.
HEX_AR	Real $\geq 0.0$	100.0	Longest edge to shortest edge aspect ratio.
HEX_EPLR	Real $\geq 0.0$	0.50	Edge point length ratio.
HEX_EPIA	Real $\geq 0.0$	150.0	Edge Point Included Angle in degrees.
HEX_DETJ	Real	0.0	J  minimum value.
HEX_WARP	Real $\geq 0.0$	0.707	Face warp coefficient.
PEN_AR	Real $\geq 0.0$	100.0	Longest edge to shortest edge aspect ratio.
PEN_EPLR	Real $\geq 0.0$	0.50	Edge point length ratio.
PEN_EPIA	Real $\geq 0.0$	150.0	Edge point included angle in degrees.
PEN_DETJ	Real	0.0	J  minimum value.
PEN_WARP	Real $\geq 0.0$	0.707	Quadrilateral face warp coefficient.
PYR_AR	Real $\geq 0.0$	100.0	Longest edge to shortest edge aspect ratio.
PYR_EPLR	Real $\geq 0.0$	0.50	Edge point length ratio.
PYR_EPIA	Real $\geq 0.0$	150.0	Edge point included angle in degrees.

Name	Value Type	Default	Comment
PYR_DETJ	Real	0.0	J  minimum value.
PYR_WARP	Real $\geq 0.0$	0.707	Quadrilateral face warp coefficient.
BEAM_OFF	Real $\geq 0.0$	0.15	CBEAM element offset length ratio.
BAR_OFF	Real $\geq 0.0$	0.15	CBAR element offset length ratio.

where:

- Test\_keyword names starting with the characters Q4 are applicable to CQUAD4 and CQUADR elements. Test\_keyword names starting with the characters T3 are applicable to CTRIA3 and CTRIAR elements. Test\_keyword names starting with the characters TET\_ are applicable to CTETRA elements. Test\_keyword names starting with the characters HEX\_ are applicable to CHEXA elements. Test\_keyword names starting with the characters PEN\_ are applicable to CPENTA elements. Test\_keyword names starting with the characters PYR\_ are applicable to CPYRAM elements.
- Skew angle for the quadrilateral element is defined to be the angle between the lines that join midpoints of the opposite sides of the quadrilateral. Skew angle for the triangular element is defined to be the smallest angle at any of the three vertices.
- Interior angles are defined to be the angles formed by the edges that meet at the corner node of an element. There are four for quadrilateral shapes and three for triangular shapes.
- Taper ratio for the quadrilateral element is defined to be the absolute value of [the ratio of the area of the triangle formed at each corner grid point to one half the area of the quadrilateral minus 1.0]. The largest of the four ratios is compared against the tolerance value. Note that as the ratio approaches 0.0, the shape approaches a rectangle.
- Surface warping factor for a quadrilateral is defined to be the distance of the corner points of the element to the mean plane of the grid points divided by the average of the element diagonal lengths. For flat elements (such that all of the grid points lie in a plane), this factor is zero.
- The edge point length ratio and edge point included angle tests are only performed for the solid elements when edge node points exist. The length ratio test evaluates the relative position of the edge node point along a straight line connecting the two vertex nodes of that edge. Ideally, the edge point should be located on this line at a point midway between the two end points. The default tolerance allows the edge node to be positioned anywhere between the two quarter points on this line. In addition, the angle between the lines joining the edge node and the end points is determined. If the angle is less than the tolerance (default is  $150^\circ$ ), then the interior angle test is considered violated and a diagnostic message will be generated if appropriate.
- The face warp coefficient test tolerance is the cosine of the angle formed between the normal vectors located at diagonally opposite corner points on each face surface. This value is 1.0 for a face where all four corners lie in a plane. The default tolerance allows angles of up to  $45^\circ$  before a message is generated.

3. A single line of output summarizing the results of all tests for an element will be output if any of the geometry tests exceeds the test tolerance. Only the first *n* of these messages will be produced. A summary of the test results indicating the number of tolerances exceeded, as well as the element producing the worst violation, is also output. If the SUMMARY keyword has been specified, only the summary table is produced and none of the single line element messages will be output.
4. When SUMMARY is not specified, each geometry test that exceeds the tolerance will be identified in the single line output summary by an indicator based on the specification for MSGTYPE. For the FATAL option, the indicator is "FAIL"; for the INFORM option, it is "xxxx"; for the WARN option, it is "WARN". If the FATAL option is specified and any test fails, the run is terminated.

**ID**                      **Comment**

Specifies a comment.

Format:

ID [=] i1, i2

Describer	Meaning
i1, i2	Character strings (1 to 8 characters in length and the first character must be alphabetic).

Remark:

1. The ID statement is optional and not used by the program.



## INCLUDE Inserts External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

### Format:

INCLUDE 'filename'

### Example:

The following INCLUDE statement is used to obtain the bulk data from another file called MYEXEC.DATA:

```
SOL 101
INCLUDE 'MYEXEC.DATA'
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
BEGIN BULK
ENDDATA
```

Descriptor	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (').

### Remarks:

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file

```
/dir123/dir456/dir789/filename.dat
```

may be included with the following input:

```
INCLUDE '/dir123
           /dir456
           /dir789/filename.dat'
```

3. See the for more examples.

## LINK Links a Main SubDMAP

Links a main subDMAP to form a solution sequence.

Format:

LINK  $\left\{ \begin{array}{c} n \\ \text{subDMAP-name} \end{array} \right\} [\text{SOLOUT} = \text{solout-DBset} \text{ EXECOUT} - \text{exeout-DBset},$

INCLUDE - incl-DBset  $\left[ \begin{array}{c} \text{MAP} \\ \text{NOMAP} \end{array} \right] \text{SOLNAME} = \text{newname} ]$

Examples:

1. LINK STATICS

Links the STATICS main subDMAP. The program links any subDMAPs compiled in this run, with any other subDMAP objects called in STATICS and stored on the MSCOBJ DBset.

2. LINK MYDMAP, SOLNAM=STATICS, SOLOUT=USROBJ, NOMAP, INCLUDE=USROBJ

Links MYDMAP and renames the solution sequence executable to STATICS. The executable will be saved on the USROBJ DBset. The order of search for subDMAP objects is:

- Compiled subDMAP in this run.
- USROBJ DBset.

Describer	Meaning
n	The solution number of the main subDMAP. See the SOL statement description for the list of valid numbers (Integer > 0).
subDMAP-name	The name of a main subDMAP. See the <a href="#">MSC Nastran DMAP Programmer's Guide</a> (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).
solout-DBset	The name of a DBset where the solution sequence executable and the link table of the solution sequence may be stored. See Remark 6. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).
exeout-DBset	The name of an alternate DBset different than solout-DBset where only the solution sequence executable may be stored. See Remark 6. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).
incl-DBset	The name of a DBset where other subDMAP objects are obtained. See Remark 2. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).

Describer	Meaning
newname	A new name which is referenced by the SOL statement. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic; default is subDMAP-name.)
MAP	Prints the link map. A link map will give the name of all the subDMAPs that make up the solution sequence.
NOMAP	Suppresses printing of the link map.

#### Remarks:

1. All DBsets specified on this statement must have the same BUFFSIZE. See the [INIT, 98 FMS](#) statement.
2. SubDMAP objects are created with the COMPILE statement either in the current run or obtained from previous runs. The LINK statement collects objects in the following order:
  - Objects created with the COMPILE statement in the current run.
  - Objects residing on the DBset-name specified by the INCLUDE keyword. The default is MSCOBJ.
3. Upon successful linking of a subDMAP, the subDMAP may be executed with the SOL statement.
4. The LINK statement must appear after all the COMPILE packages, if any. A compile package begins with the COMPILE statement and is delimited by the ENDALTER, CEND, LINK, or another COMPILE statement.
5. The link table is necessary for COMPILER (or DIAG 4, 14, 17) Executive Control statement requests and the automatic link process.
6. EXEOUT is useful in building delivery databases where executables are not to be saved. EXEOUT will be defaulted to the same DBset as specified by SOLOUT.

## MALTER Inserts and/or Deletes DMAP Statements in Solution Sequences

Inserts or deletes DMAP statements by allowing a global “string” search across all subDMAPs within the current solution sequence.

Format:

MALTER ‘string1’[(occurrence,offset)] , [‘string2’[(occurrence,offset)] ]

or

MALTER ‘string1’[(occurrence,offset)] , [k2]

Examples:

1. The following MALTER will insert a MATPRN DMAP statement to print the KJJ matrix for each superelement.

```
SOL 101
MALTER 'MALTER:AFTER SUPERELEMENT STIFFNESS .* GENERATION'
MESSAGE //' SEID=' /SEID $
MATPRN KJJZ/ $
```

2. The following MALTER will add a user DMAP after the PREFACE modules in SOL 100 (USERDMAP).

```
SOL 101
MALTER 'AFTER CALL PREFACE'
.
.
.
```

Descriptor	Meaning
'string1'	If 'string2' or k2 is not specified, the subsequent DMAP statements will be inserted after the first occurrence of 'string1'.
'string1','string2'	DMAP statements beginning with the first occurrence of 'string1' through DMAP statements containing the first occurrence of 'string2' will be deleted and may be replaced with subsequent DMAP statements.
k2	If k2 is specified, it is applied to the subDMAP in which 'string1' was found (Integer > 0).
occurrence	This flag indicates which occurrence of the preceding string is to be used, starting at the beginning of the subDMAP (Integer > 0; Default=1).
offset	This flag indicates the offset from the referenced DMAP statement. Depending on the sign, the specific DMAP statement may be above (-offset) or below (+offset) the referenced DMAP statement (Integer; Default = 0).

## Remarks:

1. If an MALTER statement is used in conjunction with the ALTER statement, then the MALTER should be placed above the COMPILE statements. Failure to place the MALTER in this position may cause ALTER statements to be ignored.
2. The MALTER statement can reference the DMAP statements in any order within a subDMAP. Two restrictions on ordering are:
  - K2 or 'string2'(occurrence,offset) references must refer to a DMAP line number that is greater than or equal to the k1 or 'string1'(occurrence,offset) reference within a single MALTER statement.
  - 'string1' and k2 or 'string2' cannot overlap DMAP line positions with another MALTER that references the same subDMAP.
3. The 'string1' or 'string2' used as a search pattern will apply to one complete DMAP statement; i.e., a multiline DMAP statement will be searched for a pattern match as if each 72 character line of the DMAP statement were concatenated together into one string; all blanks and comments (either embedded or immediately preceding the DMAP statement) will be retained.
4. The special characters used for string searching are described in Remark 9. The characters <, >, and \$, which are common DMAP characters, are also special metacharacters. If they are to be used in the search string as regular characters, then they must be preceded by a backward slash (\). For example, to find the string

```
IF (DDRMM >=-1)
```

the command is

```
ALTER 'IF (DDRMM \>=-1)' $
```

5. 'string2' (r2,02) can be defaulted to 'string1' (r1,01) by using a null string (''). For example, the alter statement

```
MALTER 'string1' (r1,01) , ''
```

is equivalent to

```
MALTER 'string1' (r1,01) , 'string1' (r1,01)
```

The defaults for (r2,02) using the null string can be overridden by specifying (r2,02).

As another example, the alter statement

```
MALTER 'string1' (r1,01) , '' (r2,02)
```

is equivalent to

```
MALTER 'string1' (r1,01) , 'string1' (r2,02)
```

6. The existing COMPILE statement options, such as LIST, XREF, SOUIN, etc., cannot be directly specified on the new MALTER statement. They are obtained as follows:
  - If a COMPILE statement exists for the subDMAP referenced by the MALTER, then options from this COMPILE statement will be used.

Otherwise, they will be taken from the COMPILER statement, with the exception that the LIST, and SORT option is always on.

7. The MALTER string search order is as follows:
  - All COMPILE statement references that are part of the existing solution sequence (i.e., SOL=) are searched first.
  - Then, all remaining subDMAPs in the solution sequence are searched in ascending alphabetical order.
  - Within a subDMAP, both 'string1' and 'string2' will be used to search for a pattern match starting at the beginning of the subDMAP (not at the current position of the last string match).
8. The MALTER statement must not exceed 72 characters (no continuations are allowed).
9. Metacharacters:

.	Matches any <i>single</i> character except <i>newline</i> .
*	Matches any number (including zero) of the single character (including a character specified by a regular expression) that immediately precedes it. For example, since "." (dot) means any character, ".*" means "match any number of characters."
[...] or < >	Matches any one of the characters enclosed between the brackets. For example, "[AB]" matches either "A" or "B". A range of consecutive characters can be specified by separating the first and last characters in the range with a hyphen. For example "[A-Z]" will match any uppercase letter from A to Z and "[0-9]" will match any digit from 0 to 9. Some metacharacters lose special meaning inside brackets. A circumflex (^) as the first character in the bracket tries to match any one character not in the list.
^ or ! or .	Requires that the following regular expression be found at the beginning of the line.
\$	Requires that the preceding regular expression be found at the end of the line.
\	Treats the following special character as an ordinary character. For example "\." stands for a period and "\*" for an asterisk. Also, to search for a tic ('), the search string must be "\'".
'	Marks the beginning and end of a pattern to be matched.
Note:	Nonportable characters such as [ ] and ^ should be replaced (e.g., ^ → ! and [ ] → < >) if portability is required. However, all the preceding characters are recognized by MSC Nastran.

10. Labels for use with the MALTER have been included in the solution sequences. See [Table 1](#). These labels will be maintained in future versions and it is strongly suggested that alters which use the MALTER command take advantage of the unique MALTER labels. Use of the MALTER labels will significantly reduce the time required to convert alters between versions.

Table 1 DMAP Labels and Corresponding SubDMAP Positions

DMAP MALTER Labels
\$MALTER:AFTER PREFACE MODULES
\$MALTER:TOP OF PHASE 1 SUPERELEMENT LOOP, AFTER PARAMETERS AND QUALIFIERS SET
\$MALTER:AFTER SUPERELEMENT STIFFNESS, VISCOUS DAMPING, MASS, AND ELEMENT STRUCTURAL DAMPING GENERATION (KJJZ, BJJZ, MJJZ, K4JJ)
\$MALTER:AFTER X2JJ MATRICES READ (K2JJ, M2JJ, B2JJ)
\$MALTER:AFTER TOTAL SUPERELEMENT STIFFNESS, VISCOUS DAMPING, AND MASS FORMULATED, STRUCTURAL + DIRECT INPUT
\$MALTER:AFTER SUPERELEMENT LOAD GENERATION (PJ)
\$MALTER:AFTER UPSTREAM SUPERELEMENT MATRIX AND LOAD ASSEMBLY (KGG, BGG, MGG, K4GG, PG)
\$MALTER:AFTER SUPERELEMENT MATRIX AND LOAD REDUCTION TO A-SET, STATIC AND DYNAMIC (KAA, KLAA, MAA, MLAA, BAA, K4AA, PA)
\$MALTER:BOTTOM OF PHASE 1 SUPERELEMENT LOOP
\$MALTER:AFTER X2PP MATRICES READ (K2PP, M2PP, B2PP)
\$MALTER:AFTER SUPERELEMENT DISPLACEMENT RECOVERY (UG)
\$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT1 (OUGV1, OES1, OEF1, ETC.)
\$MALTER:AFTER ELEMENT STRESS, STRAIN, ETC. DATA RECOVERY, SORT2 (OUGV2, OES2, OEF2, ETC.)
\$MALTER:BOTTOM OF SUPERELEMENT DATA RECOVERY LOOP
\$MALTER:USERDMAP - AFTER CALL PREFACE

## MODEL\_CHECK

### Specifies Model Check Options

Specifies model checkout run and specifies options to be used.

Format:

$$\text{MODEL\_CHECK} \left[ \text{MAT\_DENSITY} = \left\{ \begin{array}{c} \text{OFF} \\ \rho \\ \text{DEFAULT} \end{array} \right\} \right] \left[ \text{MAT\_TECO} = \left\{ \begin{array}{c} \text{OFF} \\ \alpha \\ \text{DEFAULT} \end{array} \right\} \right],$$

$$\left[ \text{MAT\_TEIJ} = \left\{ \begin{array}{c} \text{OFF} \\ \alpha_{ij} \\ \text{DEFAULT} \end{array} \right\} \right] \left[ \text{MAT\_DAMPING} = \left\{ \begin{array}{c} \text{OFF} \\ g_e \\ \text{DEFAULT} \end{array} \right\} \right],$$

[CHECKOUT][PRINT = item\_list]

Examples:

1. Execute a basic model checkout run. No special output is required.  
MODEL\_CHECK CHECKOUT
2. Execute a model checkout run. Print coordinate system and basic grid point data.  
MODEL\_CHECK CHECKOUT PRINT=(CSTM,BGPDT)
3. Execute a full solution. Modify the material density temporarily to a value of 0.0.  
MODEL\_CHECK MAT\_DENSITY=OFF  
or  
MODEL\_CHECK MAT\_DENSITY=0.0
4. Execute a full solution. Temporarily modify the values for material density and thermal expansion coefficient.  
MODEL\_CHECK MAT\_DENSITY=0.001 MAT\_TECO=1.0 MAT\_TEIJ=0.0

Descriptor	Meaning
MAT_DENSITY	Selects material density processing option.
$\rho$	Value to be used for the density.
MAT_TECO	Selects material thermal expansion direct coefficient processing option.
$\alpha$	Value to be used for the thermal expansion direct coefficients.
MAT_TEIJ	Selects material thermal expansion shear coefficient processing option.
$\alpha_{ij}$	Value to be used for the thermal expansion shear coefficients.



Describer	Meaning
MAT_DAMPING	Selects material structural element damping processing option.
$g_e$	Value to be used for the structural element damping coefficient.
OFF	Sets material property value to zero.
DEFAULT	Material property value is set to system default value. See Remark 3.
CHECKOUT	Selects model checkout solution option. See Remark 5.
PRINT	Selects items to be printed during model checkout solution.
item_list	List of model data items to be printed during model checkout run. If more than one item is specified, enclose the list in parenthesis. See Remark 6.

**Remarks:**

1. The MODEL\_CHECK statement is ignored in RESTART runs.
2. The values specified for material properties using the MODEL\_CHECK statement will be used to temporarily update data for all MAT1, MAT2, MAT3, MAT8, and MAT9 Bulk Data entries only for the duration of the run. These values do not replace data specified on the MATi Bulk Data entries. Caution should be used when postprocessing results via the PARAM POST options since operations using inconsistent data could be performed. Furthermore, when layered composite element properties and materials (PSHELL and MAT2 MID1/MID2/MID3/MID4) are generated, these equivalent MAT2 property entries are not considered to be original input data and the effects of the MODEL\_CHECK directives are permanently reflected in these MAT2 properties. Restarts should not be attempted in this case.
3. System default values of 0.0 have been defined for each of the properties. The defaults can be changed using the following Nastran statement keywords: DEF\_DENS for MAT\_DENSITY, DEF\_TECO for MAT\_TECO, DEF\_TEIJ for MAT\_TEIJ, and DEF\_DAMP for MAT\_DAMPING.
4. The MAT\_TECO describer causes the direct components of the thermal expansion coefficient to be modified. The MAT\_TEIJ describer causes the shear components of the thermal expansion coefficient to be modified.
5. The CHECKOUT option has the same effect as a PARAM,CHECKOUT,YES Bulk Data entry.
6. The following table summarizes the acceptable specifications for the PRINT item\_list.

Value	Output Generated	Parameter
CSTM	Coordinate systems	PRTCSTM
BGPDT	Basic grid point data	PRTBGPDT
GPTT	Grid point temperature data	PRTGPTT
MGG	G-set mass matrix	PRTMGG
PG	G-set load vectors	PRTPG

See the DMAP parameter descriptions in Section 5 for a discussion of the parameter name in the last column of the table and the output generated. The specification of a print item has the effect of adding a PARAM,parameter,YES entry to the Case Control Section of the file.

## SOL Executes a Solution Sequence

Specifies the solution sequence or main subDMAP to be executed.

Format:

$$\text{SOL } \left\{ \begin{array}{c} n \\ \text{subDMAP-name} \end{array} \right\} [\text{SOLIN} = \text{obj-DBset} \quad \text{NOEXE}]$$

Examples:

1. In the following example, SOL 103 is executed from MSCOBJ.

```
SOL 103
```

2. In the following example, the PHASE0 subDMAP is altered, SOL 103 is relinked onto the OBJSCR DBset (which is the default for SOLOUT), and SOL 103 is executed.

```
SOL 103
COMPILE PHASE1
ALTER 'DTIIN'
TABPT SETREE,,,,// $
.
.
.
ENDALTER $
```

3. In the following example, the solution sequence called DYNAMICS is executed from the USROBJ DBset.

```
SOL DYNAMICS SOLIN = USROBJ
```

Descriptor	Meaning
n	Solution number. See Remark 6. for the list of valid numbers (Integer > 0).
subDMAP-name	The name of a main subDMAP. See the <a href="#">MSC Nastran DMAP Programmer's Guide</a> (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).
obj-DBset	The character name of a DBset where the OSCAR is stored. See Remarks 1. and 2. (Character; 1 to 8 alphanumeric characters in length and the first character must be alphabetic).
NOEXE	Suppresses execution after compilation and/or linkage of the solution is complete. Also, the Bulk Data Section and Case Control Section are not read or processed.

Remarks:

1. If SOLIN keyword is not given and if there are no LINK statements within the input data, the program will perform an automatic link. The program will first collect the objects created in the current run by the COMPILE statement and the remaining objects stored in the MSCOBJ DBset. The program will then perform an automatic link of the collected objects.
2. If the SOLIN keyword is not given but a LINK statement is provided, the SOLIN default will be obtained from the SOLOUT keyword on the LINK statement.
3. The operation sequence control array (OSCAR) defines the problem solution sequence. The OSCAR consists of a sequence of entries with each entry containing all of the information needed to execute one step of the problem solution. The OSCAR is generated from information supplied by the user's entries in the Executive Control Section.
4. The SOLIN keyword will skip the automatic link and execute the OSCAR on the specified DBset.
5. The DOMAINSOLVER may be used in conjunction with solution sequences 101, 103, 108, and 111 to select domain decomposition solution methods.
6. The following Solution Sequences are currently available in MSC Nastran:

Table 2 Solution Sequences

SOL Number	SOL Name	Description
101	SESTATIC	Statics with options: Linear steady state heat transfer. Alternate reduction. Inertia relief.
103	SEMODES	Normal modes.
105	SEBUCKL	Buckling with options: Static analysis. Alternate reduction. Inertia relief.
106	NLSTATIC	Nonlinear or linear statics.
107	SEDCEIG	Direct complex eigenvalues.
108	SEDFREQ	Direct frequency response.
109	SEDTRAN	Direct transient response.
110	SEMCEIG	Modal complex eigenvalues.
111	SEMFREQ	Modal frequency response.
112	SEMTRAN	Modal transient response.
114	CYCSTATX	Cyclic statics with option: Alternate reduction.
115	CYCMODE	Cyclic normal modes.
116	CYCBUCKL	Cyclic buckling.

Table 2 Solution Sequences

SOL Number	SOL Name	Description
118	CYCFREQ	Cyclic direct frequency response.
128	SENLHARM	Nonlinear Harmonic Response
129	NLTRAN	Nonlinear or linear transient response.
144	AESTAT	Static aeroelastic response.
145	SEFLUTTR	Aerodynamic flutter.
146	SEAERO	Aeroelastic response.
153	NLSCSH	Static structural and/or steady state heat Transfer analysis with options: Linear or nonlinear analysis.
159	NLTCSH	Transient structural and/or transient heat Transfer analysis with options: Linear or nonlinear analysis.
200	DESOPT	Design optimization.
400	NONLIN	Nonlinear Static and Implicit Transient Analysis and all linear sequences from statics, modes, frequency, and transient inclusive with perturbation analysis based on previous nonlinear analysis
600	SESTATIC ( See <a href="#">SOL 600, ID, 154</a> )	MSC Nastran API into MSC MARC nonlinear
700	NLTRAN ( See <a href="#">SOL 700, ID, 173</a> )	Nonlinear Explicit Transient Analysis

## SOL 600,ID

## Executes Marc from Inside Nastran

Creates Marc input and optionally executes Marc from inside SOL 600.

## Format:

```
SOL 600, ID PATH= COPYR= NOERROR MARCEXE=SOLVE NOEXIT
OUTR=op2,xdb,pch,f06,eig,dmap,beam, sdrc,pst,cdb=(0, 1, 2, or 3) STOP= CONTINUE=
S67OPT= SCRATCH= TSOLVE= SMEAR PREMGLUE MRENUCLE= MRENUGRD=
MRENUMBR= SYSabc= S6NEWS=
```

## Examples:

```
SOL 600,106 STOP=1
SOL 600,106 OUTR=OP2,F06
SOL 600,106 PATH=/progs/marc2003/tools OUTR=op2,f06
SOL 600,129 PATH=1 STOP=1
SOL 600,129 PATH=1 OUTR=OP2,CDB=0
SOL 600,106 PATH=1 CONTINUE=1
SOL 600,106 PATH=1 MARCEXE=SOLVE OUTR=OP2
SOL 600,153 PATH=1 STOP=1 TSOLVE=M
SOL 600,106 OUTR=OP2,F06 SMEAR
SOL 600,106 OUTR=OP2 PERMGLUE
SOL 600,106 PATH=1 STOP=1 MRENUCLE=2
SOL 600,106 PATH=1 STOP=1 MRENUGRDD=2
SOL 600,106 PATH=1 STOP=1 MRENUMBR=2
SOL 600,106 STOP=1 SYS001=8193 SYS9=6000
SOL 600,106 STOP=1 S6NEWS=YES
```

SOL 600,ID is an Executive Control statement similar to SOL. The difference between SOL and SOL 600,ID is that the computations (element matrix formulations, matrix decomposition, etc.) will be performed by Marc rather than by Nastran. Inputs and outputs as much as possible will be the same as (or similar to) the familiar Nastran inputs and outputs, however standard Marc inputs and outputs are also available. SOL 600 is primarily intended for nonlinear static and dynamic analysis of 3D structures that have already been manufactured and assembled. Although it has capabilities for 2D structures and for certain manufacturing processes, those capabilities should only be used for “simple” cases. For more complex 2D and manufacturing analyses either a standalone version of Marc or SOL 400 is recommended.

The SOL 600,ID statement should normally be used only for nonlinear analysis, but it may also be used for certain classes of linear static or dynamic analyses. The recommended form of this command is shown with the options provided above. If entered with “SOL 600,ID” only, it acts just like SOL except a Marc input data file “jid.marc.dat” will be generated (“jid” is the name of the Nastran input file without the extension.) For example, if the Nastran input file is named abcd.dat, (or abcd.bdf) then “jid”=abcd.

The required ID represents many valid solution sequence integer or names shown in [Table 2](#) for the SOL statement. Examples are 106, 129, NLSTATIC, NLTRAN. The following solutions are not available: 107, 108, 110, 111, 112, 114, 115, 116, 118, 144, 145, 146, 190, 200, and 400 (and their equivalent names). Solutions specified in [Table 2](#) of the SOL statement may be used. If the model has contact, ID must be 106, 129, 153, 159 or their equivalent names unless PERMGLUE is used.

Although SOL 600 supports 2D analyses (axisymmetric and plane strain), the support is not complete. It is strongly recommended that 2D analyses use some other solution sequence.

Most items on the SOL 600,ID after ID itself may be specified in environmental variables. This may be done any way environmental variables can be set. They may be set by the Nastran user at run time or by the system administrator when Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the . A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user's rc file, a local rc file or in a script used to submit Nastran. Any string or value listed on the SOL 600,ID statement is also valid as an environmental variable. If the environmental variables are placed in the system-wide rc file, they may be used by a company for all Nastran users and even hide the fact that Marc is being spawned if so desired.

The following environmental variables are available:

Environmental Variable	Item on SOL Statement
NASM_PATH	PATH
NASM_COPYR	COPYR
NASM_OUTR	OUTR
NASM_STOP	STOP
NASM_NOERROR	NOERROR
NASM_STRFILE	Path and name of marcfilt.txt file (see below)

## PATH

PATH is an optional item which determines the location of the version of Marc to be executed. If PATH is omitted, the version of Marc included with Nastran will be used if it can be located. In this case, the run script for Marc (run\_marc or run\_marc.bat) will be expected to be in a directory under /MSC\_BASE. MSC\_BASE is an environmental variable set when Nastran first starts execution that defines the base installation directory for Nastran. If for some reason MSC\_BASE cannot be determined, the commands to spawn Marc will fail and the user must re-run Nastran with one of the PATH options set or the NASM\_PATH environmental option set to the desired location of Marc's tools directory.

## PATH=1

If PATH=1 is specified, Nastran will determine the proper command to execute the companion program. To aid Nastran in determining the program's location, a file named marcrun.pth must be available in the same directory where the Nastran input file resides. The marcrun.pth file must contain one line providing the location (complete path) of the run\_marc script. A typical example of the line in the file marcrun.pth would be

```
/mycomputer/marc200x/tools
```

To this path is appended the string `"/run_marc -jid name.marc -v no"` and possibly other items to form the complete string used to execute Marc. This complete string looks like the string shown in the following PATH=3 example. Note that on Windows systems, substitute a back slash for the forward slashes shown. Do not terminate the line with a forward slash or back slash.

### PATH=2

If PATH=2 is specified, it is expected that the directory with the run\_marc script is on the PATH. If PATH=2 is specified, Marc will be executed from inside Nastran using the command:

```
run_marc -jid jid.marc.dat -v no
```

### PATH=3

When PATH=3 is specified, the complete command to execute Marc must be contained in a file named marc.pth (lowercase). This file should typically contain one line of the form:

```
/mycomputer/marc200x/tools/run_marc -jid name.marc -v no
```

### COPYR

COPYR is an optional item. If COPYR is specified, Marc output files will be copied to Nastran output files and/or deleted according to the options shown in the following table:

COPYR Option	Copy Marc Output Files to Nastran Output Files	Delete Marc Input & Output Files
0 (default)	No	No
1 or -1 (see below)	Yes	Yes
2 or -2 (see below)	Yes	No
3	No	Yes

If COPYR is 1 or 2, Marc's out and log files will be copied exactly as produced by Marc.

If COPYR is -1 or -2 the actions as shown above for +1 or +2 will occur, and Marc-type text will be converted to Nastran-type text (or any other desired text) using an ASCII file named marcfilt.txt. This file must be located in the same directory where the Nastran input resides or in the same directory where the Marc executable resides. The marcfilt.txt file can contain as many lines as desired like the one shown below:

```
"Marc string 1" "Replacement String 1"  
"Marc string 2" "Replacement String 2"
```

That is, each line contains two strings. Each string starts and ends with a double quote sign ("). The Marc string must match the exact content and case as found in the Marc .out or .log files. The replacement string may be any string desired and can be the same length, shorter or longer than the Marc string. The two strings must be separated by at least one space, but more spaces are acceptable. Line lengths for marcfilt.txt, as well as Marc's .out and .log files are limited to 200 characters for the text replacement option.



The following Marc files are potentially affected by the COPYR option:

Marc Output File	Nastran Output Copied to	COPYR
name.marc.out	name.f06	1, 2, -1, -2
name.marc.log	name.log	1, 2, -1, -2
name.marc.t16	Not copied, will remain if produced	
name.op2, fort.11 or ftn11	Not copied, will remain if produced	

### MARCEX=SOLVE

MARCEX=SOLVE is an optional item. If MARCEX is entered, an existing input file named jid.marc.dat is assumed to exist in the directory where the run was submitted. Nastran will execute Marc using the existing jid.marc.dat file. A new Marc file will not be created. Other options available when MARCEX is used are PATH and OTR. Options not available with MARCEX are COPYR, STOP, NOEXIT, NOERROR and CONTINUE. Beware that the original jid.marc.dat will be renamed to jid.marc.dat.1 automatically by Nastran just like an existing jid.f06 is renamed to jid.f06.1

### NOERROR

NOERROR is an optional item. If NOERROR is specified, errors due to features that are available in Nastran but not available in Marc, and/or features not yet supported by the translator will be ignored (see Restrictions and Limitations). If NOERROR is entered and STOP=2 (or 3) is not specified, Marc will be executed even though the complete Nastran model may not have been completely translated. We recommend that NOERROR only be used by experienced analysts and then only with extreme caution.

### NOEXIT

NOEXIT is an optional item. If entered, the DMAP generated “on the fly” to process the OTR options will not contain EXIT and Nastran will proceed. This means in most cases, the Nastran solution as well as the Marc solution will occur. If .f06 is specified as one of the OTR options, this could cause confusing output as both the Marc and Nastran results will be in the .f06 file. Confusion could also result from both outputs being in .op2, .xdb and/or .pch files. Therefore, this option should only be used with great care.

Listing of the DMAP generated on the fly for SOL 600 can be suppressed by placing ECHOFF just after the SOL 600 entry.

### Restrictions:

- OTR options may not be used with restart jobs.
- OTR options are not available for SOL 600 2D analysis such as axisymmetric or plane strain.
- OTR options are not available for SOL 600 heat transfer.
- Spaces in the list of OTR items are not allowed.

## OUTR

OUTR is an optional item. If OUTR is specified, Marc output results will be converted to various types of Nastran formats. The OUT option on the Nastran command should not be used with any OUTR options. The type of output to be produced depends on the OUTR options entered as well as any DMAP entered in the executive control. If OUTR is omitted, no Marc output will be brought back into Nastran, but standard Marc .out, .t16 and/or .t19 as well as an op2 file will be available depending on the options selected with PARAM, MARCT16, PARAM, MARCT19 and other options. OUTR options may not be used with restart jobs.

The following options are available:

### Option 1 -- Specify a String of Desired Output Types (Preferred Method)

OUTR=OP2,F11,F06,PCH,XDB,T16,T19,PST

Use any or all of the above to request the following options except that both OP2 and XDB should not be entered:

- |      |  |
|------|--|
| OP2  | Create output2 file named jid.op2 consisting of input model and output results datablocks. This option requires PARAM,POST,-1 or PARAM,POST,-2 in the Bulk Data.   |
| F11  | Create output file fort.11 or ftn11 (depending on the computer system) consisting of output results datablocks only.   |
| F06  | Put Marc output results (displacements, stresses, strains) in Nastran's jid.f06 file using OFP. The resulting output will look just like any standard Nastran run.   |
| PCH  | Create punch file named jid.pch with Marc's output in standard Nastran punch format.   |
| XDB  | Create .xdb database file named jid.xdb with input model and output results. This option requires PARAM,POST,0 in the Bulk Data. XDB is not available with the eig option and if entered will switch to the OP2 option.  |
| eig  | The eig option must be specified if .op2, .xdb, .pch, or .f06 options are specified and Marc performs natural frequency or buckling eigenvalue analysis. The reason it must be provided on the SOL entry is to enable Nastran to create DMAP on the fly which include the LAMA data block. If the eig option is omitted, eigenvectors will be present in the Nastran output but no eigenvalues will be available. The beam and eig options are mutually exclusive (you cannot specify both). |
| BEAM | The BEAM option must be specified if .op2, .xdb, .pch, or .f06 options are specified and you want to place internal loads in any of these files. The BEAM option is not available for Windows systems.   |
| SDRC | An SDRC op2 file will be produced. PARAM,POST,-2 is also necessary in the Bulk Data for this option. Note: The datablocks might be in a different order than for other solution sequences.   |
| T16  | Marc's results will be saved during the Marc execution on a binary (or unformatted) file named jid.marc.t16 (this happens by default and does not need to be specified on the SOL 600 line).   |

- T19 Marc's results will be saved during the Marc execution on an ASCII file named `jid.marc.t19`. The `t19` file will normally be saved if `param,maract19,1` is entered.
- PST Nastran will be run to output a previous Marc run's results contained on `t16` file in the desired forms (`OP2,F11,F06,PCH` and/or `XDB`). The appropriate `OUTR T16` options must be selected in addition to `PST` (specify one or more of `OP2,F11,F06,PCH` and/or `XDB`.) Nastran will not be run past `IFP` and is used only to perform the desired output results conversions. A previous Marc `t16` file must be copied to the new `jid.marc.t16.1` (you may not process `XDB` and `OP2` in the same run.)
- DMAP The user will enter his own DMAP to create whatever type of output that is desired, such as `.op2`, `.xdb`, `.pch`, or `.f06`. For all other options, DMAP as needed is generated internally by Nastran.
- cdb 3D Contact will be output in one of the datablocks described below:
- 0 Store output in `OESNLBR` and `OESNLXR` (`OESNLXR` will be empty like SOL 106)
  - 1 Store output in `OESNLBR`
  - 2 Store output in `OESNLBD`
  - 3 Store output in `OESNLXD`

- Note:**
1. SOL 106 outputs both `OESNLBR` and `OESNLXR` but `OESNLXR` is empty.
  2. SOL 129 only outputs the `OESNLXD` datablock and it is empty.
  3. Case Control `BOUTPUT` is also required to obtain this type of output.
  4. The default is 0 if contact is present in the model and `OUTR=op2` (or `.xdb`, `punch` and/or `.f06`).
  5. This option is specified like the example shown:  
`OUTR=OP2,F06,CDB=0`
  6. The datablocks have the same names and type of information whether executing SOL 600,106 or SOL 600,129.

### Option 2 -- Specify an Integer to Select Certain Options (Not Recommended)

`OUTR=1` or `2` and an `op2` file named `fort.11` or `ftn11` will be produced and DMAP as shown below is required to bring the Marc output results back into the Nastran database.

```
COMPILE NLSTATIC
ALTER 'SUPER1' $
INPUTT2 /OUGV1,OES1,OSTR1,TOL,/-1/11 $
OFFP OUGV1,OES1,OSTR1//0/1 $
EXIT $
```

The 1 at the end of the `OFFP` statement produces output in the `.f06` file. If a punch file is also needed, change the 1 to a 5. If an `XDB` file is also needed, add the following lines just after the `OFFP` line:

```
DBC TOL,CASECC,,,,,,,,,,,,,,,,,,,,,///
'OL''CASECC'/////////////////////////////////
-1/DBCPTH/S,N,CP/'TRAN' '//GEOMU/LOADU/POSTU/
DBC DIAG/DBC CONV/DBC OVWRT $
```

```

DBC OUGV1, OES1, //
' OUG ' / ' OES ' //
-1 / DBCPATH / S, N, CP / ' ' TRAN ' ' // GEOMU / LOADU / POSTU /
DBC DIAG / DBC CONV / DBC OVWRT $
DBC OSTR1, //
' OES ' //
-1 / DBCPATH / S, N, CP / ' ' TRAN ' ' // GEOMU / LOADU / POSTU /
DBC DIAG / DBC CONV / DBC OVWRT $

```

OUTR can be set to one of the following values to automatically produce the output in Nastran form without entering any DMAP. In fact, no DMAP should be entered for the options greater than 2 shown:

Table 4-3 Integer Options Available Using SOL 600 OUTR Option --  
Nastran Output Results Produced When Marc Exits

OUTR (IO)	OP2 with Input Datablocks	fort.11 or ftn11 Output Datablocks Only	.f06 (Print)	.pch (Punch)	.xdb	Marc File Used
1	N	Y	N	N	N	.t19
2	N	Y	N	N	N	.t16
16	Y	Y	N	N	Y	.t16
166	Y	Y	Y	N	Y	.t16
266	Y	Y	N	Y	Y	.t16
366	Y	Y	Y	Y	Y	.t16
19	Y	Y	N	N	Y	.t19
199	Y	Y	Y	N	Y	.t19
299	Y	Y	N	Y	Y	.t19
399	Y	Y	Y	Y	Y	.t19

If OUTR = -1, -2, -16, -166, -266, -366, -19, -199, -299 or -399 only the output conversion process takes place. An Marc input file is not produced, Marc is not spawned from Nastran, but .op2, .xdb, .pch and/or .f06 results can be produced. For such cases, the Case Control and Bulk Data files can be dummies (for example, they can contain several nodes and one element) or a full file could be used. These options are handy if Marc is run by modifying the Marc input file (jid.marc.dat) with an editor or for someone who creates Marc input and runs Marc outside the Nastran environment, but wants output in one of the Nastran formats (see Remark 6.)

### STOP

STOP is an optional item. STOP is used to prevent execution of Marc or exit Nastran after IFP, if so desired. DO NOT ENTER any of the STOP options if any of the OUTR options are entered as the DMAP generated automatically by Nastran will put an EXIT in the proper place. The various options are as follows:



## STOP=0

If STOP=0 Nastran will not be stopped after Marc exits. Nastran will attempt to obtain its own solution to the problem if possible. Use of this option can lead to confusion because results from both Marc and Nastran will be available. If the Marc results are placed in the .f06 file and if the Nastran results are also available in the .f06 file, it will be difficult to tell which results came from Nastran and which results came from Marc. This also applies to .op2 files and .xdb files. It is suggested the STOP=0 option be used by extremely experienced SOL 600 users and even then with great care.

## STOP=1

If STOP=1 Nastran will be gracefully stopped after IFP. This option is used to prevent Nastran from performing its own solution (normally used when the solution is performed by the Marc). STOP=1 should be normally used if OUTR is not specified. STOP=1 is the default if no STOP, CONTINUE or OUTR options are entered.

## STOP=2

For STOP=2 Marc will not be executed. This option is used if you wish to examine the Marc input file and make changes prior to running Marc. However, if STOP=2 is entered, the OUTR options will not be available.

## STOP=3

STOP=3 is a combination of STOP=1 and STOP=2. Nastran is stopped after IFP and Marc is not executed. This would be the normal STOP option if you want to examine a Marc input file, then execute Marc manually. The STOP=2 option is normally used if you want to obtain comparative results between standard Nastran solutions and Marc solutions (in which case, all input options must be fully supported by both programs). If STOP=3 is entered, the OUTR options will not be available.

## CONTINUE=

CONTINUE= specifies an option as to how Nastran will continue its analysis after Marc finishes. For this to happen, do not enter any STOP or OUTR options. It is not usually possible to perform more than one of these operations if necessary.

- |   |  |
|---|--|
| 0 | Nastran will continue the current solution sequence as normal. For example if SOL 600,106 is entered, SOL 106 will continue as normal after Marc finishes. Of course, no 3D contact or materials not supported by SOL 106 may be used.   |
| 1 | Nastran will switch to SOL 107 to compute complex eigenvalues. Marc will generate DMIG matrices for friction stiffness (and possibly damping) on a file specified by param,marcfil1,name and time specified by param,marcstif,time. This is accomplished by making a complete copy of the original Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file. The user must put CMETHOD and CEIG in the original Nastran input file. |

- 2 (Option not presently available.) Nastran will switch to SOL 107 to compute complex eigenvalues. Marc will generate OUTPUT4 matrices for friction stiffness (and possibly damping) on a file specified by `pram,marcfil2,name` and time specified by `param,marcstif,time`. This is accomplished by making a complete copy of the original Nastran input file and spawning off a new job with the SOL entry changed and an include entry for the DMIG file.

The original Nastran file should include `CMETHOD=id` in the Case Control command and a matching `EIGC` entry in the Bulk Data.

In addition, the DMIG entries specified by `MDMIOUT` will be included in a separate Nastran execution spawned from the original execution. Case Control and Bulk Data will be added to the original input to properly handle these matrices in the spawned Nastran execution.

- 6 Same as option 1 except SOL 110 is run. For this option, the original Nastran input file must contain `METHOD=ID1` and `CMETHOD=ID2` in the Case Control as well as matching `EIGRL` (or `EIGR`) and `EIGC` entries in the Bulk Data.

- 7 Same as option 1 except SOL 103 is run for real eigenvalues/eigenvectors. The database can be saved to restart into SOL 110 if desired. This should be done on the command line or in a rc file with `scratch=no`. For this situation, the original Nastran input file must include `METHOD=id` in the Case Control command and a matching `EIGRL` or `EIGR` entry in the Bulk Data. (`CMETHOD` and `EIGC` can also be included.) The actual restart from SOL 103 to 110 must be performed manually at the present time.

- 101+ Continue options 101 to 400 are used to convert Marc's initial contact tying constraints to MPC's and then continue in SOL 101 to 112 as a standard Nastran execution. For example, if `CONTINUE=101`, a SOL 101 run with all the geometry load cases, etc. from the original run would be conducted with the addition of the initial contact MPC determined from Marc. The `continue=101+` options are frequency used to model dissimilar meshes as well as glued contact which does not change throughout the analysis. This option can be used for any standard Nastran sequence where the initial contact condition does not change. In order for initial contact to work, the surfaces must be initially touching. If they are separated by a gap, the MPC's will be zero until the gap closes and thus the initial MPC's are zero. This option automatically sets `BCPARAM INITCON=1`.

### S67OPT=NO

If `S67OPT=NO` is entered the following action will be taken for SOL 600 or SOL 700: `TA1MCK` and `EMGPRO` will not be disabled (when these routines are disabled, materials used only by SOL 600 or SOL 700 such as `MATG`, `MATE`, `MATHP`, etc. may be in the model and the `t16op2` conversion will take place, otherwise the job will fail with a `FATAL ERROR`). Also, Case Control `FATAL` error termination will occur at the same place as other Nastran Solution Sequences. If `S67OPT=YES` or `S67OPT` is omitted entirely, `TA1MCK` and `EMGPRO` will be disabled and Case Control `FATAL ERRORS` will cause job termination immediately. `S67OPT=YES` is the default.

## SCRATCH=

Determines what will happen when a SOL 600 job is initiated with SOL 600 database files (\*.3dc, \*.prp) present in the run directory (this usually means another job is running and conflicts can occur.) The default is SCRATCH=WAIT01. Options are as follows:

SCRATCH=DELETE	Attempt to delete all *.3dc and *.prp files, continue with present job. If the attempt to delete them fails, the job will terminate with an appropriate message.
SCRATCH=ABORT	If any *.3dc or *.prp files are found, abort the present job with an appropriate message.
SCRATCH=WAIT	If any *.3dc or *.prp files are found, wait until they disappear, then begin current run. This option will wait an “indefinite” amount of time.
SCRATCH=WAITxx	If any *.3dc or *.prp files are found, wait for xx minutes or until they disappear, then begin current job. If they do not disappear within xx minutes, abort the current job with an appropriate message. Examples, to wait up to 1 minute, enter SCRATCH=WAIT01, to wait up to 15 minutes, enter SCRATCH=WAIT15. Note: xx can range from 01 to 99.

## SCRATCH= Remarks:

1. For the WAIT options, if no \*.3dc or \*.prp files are found, the job will start immediately.
2. No spaces are allowed.

## TSOLVE

Determines which “solver” (Nastran or Marc) is used to solve a heat transfer analysis. The default is Nastran.

TSOLVE=M	Marc is used as the thermal solver.
TSOLVE=MS	Marc is used as the thermal solver followed by a structural analysis using the temperatures from the end of thermal analysis.
TSOLVE=N	Nastran is used as the thermal solver (default).

## TSOLVE Remarks:

1. If the default is used and thermal contact is present, SOL 600 spawns Marc to calculate initial thermal contact variables which are then read by Nastran, turned into Nastran CELAS and other variables. A second Nastran run is spawned for the primary Nastran run to complete the heat transfer calculations.
2. If OUT or OUTDIR are used with thermal contact in SOL 600 they must both reference the same directory.
3. This option should be entered for heat transfer analysis only.

## SMEAR

The term SMEAR, as used by SOL 600, is different than that used on the PCOMP Bulk Data entry. For SOL 600, SMEAR is the same as LAM=BLANK on the PCOMP entry. Other LAM options are not available using SOL 600, however complete integration and fast integration methods are available, see the PCOMP Bulk Data entry.

If the string SMEAR is entered on the SOL 600,ID command line, composite shell entries using PCOMP will use the smeared approach. If SMEAR is not entered, the through-the-thickness integration approach will be used. The smeared approach is identical to other Nastran solution sequences where PCOMP entries are converted to PSHEL and MAT2 entries. The through-the-thickness integration approach is more accurate for post-buckling and nonlinear analyses but takes more computer time. OP2.f06 and punch outputs are available and are controlled by the OUTR options OUTR=xxx where xxx is .op2, .f06 and/or .pch. If any OUTR options are specified, .op2 must be included. In addition, standard Case Control requests are required.

### SMEAR Option Restrictions

1. The SMEAR option may only be used if all composite materials in the model are made of shell elements (if there are any composite solid elements, this option may not be used.)
2. Case Control requests for DISP(options)=ALL, the STRESS(options)=ALL must be entered. STRAIN(options)=ALL is optional. (options) consist of any combination of (print,plot,punch)
3. The SMEAR output options may not be controlled using sets.
4. It is suggested that the Marc t16 file be limited to only those output “items” absolutely necessary as composite output can be large and take significant computer time.
5. If OUT or OUTDIR are used with this option, they must reference the same directory.

## PERMGLUE

Specify PERMGLUE if permanent glued contact is to be used. Permanent glued contact is glued contact where the glued condition is determined using initial contact. This glued condition will remain throughout the analysis. The MPC's produced by the PERMGLUE option are identical to those formed in SOL 101 or SOL 103 when the permanent glue option is specified. When this option is used, set BCONTACT=ALLGLUP. For SOL 600, the PERMGLUE option is the only way contact can be used with SOL600,101 or SOL 600,103 or other “linear” analyses.

## MRENUCLE

Determines if SOL 600 elements will be renumbered or not. (Default = 0)

- |   |  |
|---|--|
| 0 | No renumbering will occur (suggested for models with largest element number less than approximately 20000)   |
| 1 | All elements will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file elenum.txt               |
| 2 | All elements will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis. |



Remarks concerning MRENULE:

1. MRENULE must be set on the SOL 600 entry if the maximum element number is greater than 9,999,999.
2. The "=" and an integer of "0" "1" or "2" must follow "MRENULE" with no spaces when MRENULE is entered on the SOL 600 entry.
3. If the maximum element is 9,999,999 or smaller MRENULE may be set as a parameter in the bulk data or placed in a rc file.
4. MRENULE should not be set on the SOL 600 and as a parameter.
5. For MRENULE=1 an equivalence list of original and re-numbered element numbers is output on file elenum.txt

## MRENUGRD

Determines if SOL 600 grid ID's will be renumbered or not. (Default = 0)

- |   |   |
|---|---|
| 0 | No renumbering will occur (suggested for models with largest grid ID less than approximately 20000)   |
| 1 | All grid ID's will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt                |
| 2 | All grid ID's will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis. |

Remarks concerning MREUGRD:

1. MREUGRD must be set on the SOL 600 entry if the maximum element number is greater than 9,999,999.
2. The "=" and an integer of "0" "1" or "2" must follow "MRENULE" with no spaces when MREUGRD is entered on the SOL 600 entry.
3. If the maximum grid ID is 9,999,999 or smaller MREUGRD may be set as a parameter in the bulk data or placed in a rc file.
4. MREUGRD should not be set on the SOL 600 and as a parameter.
5. For MREUGRD=1 an equivalence list of original and re-numbered grid id's is output on file gridnum.txt

## MRENUMBR

Determines both grid and element ID's for SOL 600 will be renumbered or not. (Default = 0)

- |   |   |
|---|---|
| 0 | No renumbering will occur (suggested for models with largest grid ID less than approximately 20000) |
|---|---|

- 1 All grid and element ID's will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt
- 2 All grid and element ID's will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis.

Remark concerning MREUMBR: All remarks for MRENUCLE and MRENUGRD apply.

### S6news

S6news can be set to YES or NO. If set to YES, a list of new features currently considered to be in beta tests are output complete with the Case Control and/or Bulk Data descriptions anticipated once these features are released.

### SYSabc

Allows Nastran system cells to be set on the SOL 600 Executive Control statement if desired. Abc must be an integer ranging from 1 to 500 and be a valid system number. INTT must be an integer with 1-8 digits. More than one SYSabc=INTT statement may be made if so desired, however abc may not be repeated. Note that the limit on the integer following the equal sign is 8 digits.

### Running SOL 600 in Steps -- Modification of the SOL 600 Statement Using Environmental Variables:

It is possible to run the main portions of SOL 600 in single steps without changing the Nastran input file. This is accomplished using one of the two environmental variables discussed below. A user can set these variables in a script that runs Nastran, from the command line or for Windows using the control panel. Note that on Linux systems, the name of the environmental variable must be in upper case. The string to which it is set can be in upper or lower case and will be converted to upper case.

### To Run SOL 600 in Three Steps Without Changing the SOL 600 Statement in the Input File:

First, make sure that your SOL 600 input file has a SOL 600 statement that contains all of the features you would want if all steps were done in a single run. For example, if you wish to make an op2 file and place the results in the .f06 file, a typical SOL 600 statement would be as follows:

```
SOL 600,NLSTATIC PATH=1 OTR=OP2,F06
```

or

```
SOL 600,NLSTATIC OTR=OP2,F06
```

(if the default path to Marc is to be used).

It is important to have the OTR options specified at the end of the SOL 600 statement. The following environmental variable can be set as shown to run the three steps (a Linux shell example is shown):

1. export MARC\_RUN="stop"

This will tell Nastran to run the internal Nastran-to-Marc translator only. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OTR=OP2,F06 STOP=3
```

This change will be shown in the .f06 file.

2. export MARC\_RUN="solv"

This will tell Nastran to run Marc from inside Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OUTF=OP2,F06 MARCEXE=SOLVE
```

For this run, it is assumed that a file named jid.marc.dat resides in the input file directory created from a previous translator-only run. The Nastran script will automatically rename jid.marc.dat to jid.marc.dat.1, but the Nastran executive processing will name it back to jid.marc.dat

3. export MARC\_RUN="pst" This will tell Nastran to run the t16 to op2 translator inside Nastran. The first SOL 600 statement shown would be changed internally just for this run above to the following:

```
SOL 600,NLSTATIC PATH=1 OUTF=OP2,F06,PST
```

For this run, it is assumed that a file named jid.marc.t16 as well as the original Nastran input file jid.dat resides in the input file. The "t16" file should have been created from a previous Marc execution using the same computer system (cross-platform support is not available for this step). The Nastran script will automatically rename jid.marc.t16 to jid.marc.t16.1, and the Nastran t16op2 conversion routines will look for files with names jid.marc.t16, jid.marc.t16.1, jid.marc.t16.2 up to jid.marc.t16.5 in that order. If no such files are found, the t16op2 job will exit with a message.

### Method to Completely Modify the SOL 600 Statement:

For maximum versatility without having to modify the Nastran input file, the SOL 600 statement can be modified completely using the environmental variable SOL600\_CMD. Assuming that the original SOL 600 statement in jid.dat contains the string:

```
SOL 600,NLSTATIC OUTF=OP2,F06
```

and the environmental variable is set as follows:

```
export SOL600_CMD="SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE"
```

the new SOL command line internal to Nastran will be

```
SOL 600,NLSTATIC PATH=1 MARCEXE=SOLVE
```

and Nastran will stop after creating the Marc input file. This would be the same as if the following SOL 600 statement was entered:

```
SOL 600,NLSTATIC PATH=1 STOP=3
```

Any valid SOL 600 statement can be issued using the SOL600\_CMD environmental variable without changing the original Nastran input file at all.

### Remarks:

1. Only one SOL 600,ID job may be run in a directory at any given time. However, if a previous run was made and output files such as name.marc.t16 were produced, they will be renamed name.marc.t16.1, etc. following the Nastran re-naming convention.
2. If OUTF is specified, STOP must not be specified.

3. The COPYR option can be used to delete all files directly created by Marc if the output desired are Nastran files only.
4. When OUTR is specified, the Marc files such as jid.marc.out, jid.marc.t16 will be renamed to jid.marc.out.1 and jid.marc.t16.1 at the start of the run. This renaming is accounted for when opening the files
5. To generate .xdb files, PARAM,POST,0 must be included in the Bulk Data Section. To generate OP2 files with geometry, PARAM,POST,-1 (for Patran and Femap) or PARAM,POST,-2 (for SDRC) should be included in the Bulk Data.
6. Although SOL 600,ID supports linear analysis (ID=101, 103, 105), not all features are available. For example, Case Control commands, STATSUB, SUBCOM, SUBSEQ, SYMCOM, AUXMODEL, AXISYMMETRIC, CLOAD, DEFORM, HARMONICS, MFLUID, NSM, and REPCASE are not available. For nonlinear analyses (ID=106, 129) Case Control commands, NNLOAD and NONLINEAR are not available.
7. To output displacements in the jid.marc.out file, do the following:  
In the Case Control, set DISP(PRINT)=ALL or DISP(PRINT,PLOT)=ALL  
In the Bulk Data, include the following two parameters:

PARAM,MARCPRNR,1  
PARAM,MARCND99,-1

8. All SOL 600 character variable parameters, such as MRAFFLOW, must be left justified in the starting in field 3.
9. Fixed load stepping (or time stepping) is controlled primarily by PARAM,MARCITER and PARAM,MARCAUTO rather than NLPARM or NLAUTO.
10. 2D and 3D contact and elements may not be mixed in the same model.
11. For the OUTR options, the only stress tensor available is Cauchy stress (E341). If some other stress tensor is selected using MARCOUT, and E341 is not selected, no stresses will be available in the .op2, .xdb, .pch, or .f06 files.
12. For multi-layer composites, stresses in the preferred direction (E391), also known as the layer direction, are usually necessary. The default Cauchy stresses (E341) will be automatically changed to E391 for composites.
13. SOL 600 supports a new field on RFORCE (2nd line 4th field) to allow different portions of the structure to have different rotation accelerations.
14. 2D Plain stress is available and this type of analysis is achieved by setting MID=-1 on all PSHELL entries and adding the following parameters to the bulk data.

PARAM,MRALIAS,011003  
PARAM,MALIAS02,006003

SOL 600 will use the alias commands to convert plain strain elements that normally are introduced with MID2=-1 into the input into plain stress. Although MALIAS02 specifies that Marc element 3 be used for CTRIAi, the new Marc element 201 plain stress triangle will be used instead of a degenerate element 6 plain stress quad. By default, the CTRIAi node numbers will not be reversed, however if the user wants to reverse them enter PARAM,MREVPLST,1 in the bulk data. Any contact entered into the file needs to specify 2D on all BCBODY entries. 2D plain stress analysis for SOL 600 is available through Patran which will generate the above alias parameters automatically.

### Restrictions and Limitations of SOL 600

Certain features are available in Nastran that are not available in Marc, and vice versa, the following restrictions/limitations are imposed on Nastran. Those restrictions indicated by (\*) will be removed as soon as possible. Items with (\*\*) will issue a FATAL error (for the Nastran-to-Marc translator internal to Nastran) and Marc will not be “spawned” from Nastran unless NOERROR is entered on the SOL 600 statement.

- External superelements are supported. Other types of superelements are not currently supported.
- Scalar points are not supported.
- PBCOMP is not supported.
- CCONEAX is not supported.
- CBARAO is not supported.
- Output set definitions that contain grid or element numbers greater than the largest grid or element in the model will produce errors in Marc.
- Output set definitions may include the word BY as in output plot set definitions for use by Marc only. Nastran must be stopped using STOP=1 or one of the OUTR options since BY is a FATAL ERROR to Nastran.
- Nastran’s CREEP entry must be changed to the MATVP entry.
- For orthotropic materials using MATORT, all shear moduli must be entered.
- SPOINTs are mapped to GRIDs with x, y, z coordinates at (0., 0., 0.)
- SLOAD and other scalar features are not supported.
- CELAS3 and CELAS4 are not supported.
- CLOAD is not supported.
- Fracture Mechanics is available.
- Aerodynamics is not supported.
- Bulk data entries with + or \* in column 73 must have an actual continuation card for most entries. Nastran does not require this, but the internal Marc translator does. (\*)
- Slideline contact is not supported (BLSEG, BWIDTH, BFRIC, BCONP, and BOUTPUT, if entered will cause FATAL ERRORS in Nastran).

- Offsets are available for CBAR, CBEAM, CQUAD4 and CTRIA3 in all types of structural analysis (linear or nonlinear). The offsets must be specified in the global coordinate system (displacement output coordinate system) unless PARAM,MAROFSET is 1 (2 or 3, see description in [Parameter Descriptions, 794](#)). Offsets are available for CQUAD8 and CTRIA6 but only if all 8 or 6 grids are defined for these elements, respectively. If PARAM,MAROFSET,1 (2 or 3) is included in the bulk data, the offsets will be incorporated using a new Marc feature that does not need extra grids or elements.
- For SOL 600, it is required that a Case Control LOAD or DLOAD entry be made for each subcase. If there are no subcases, one LOAD or DLOAD entry must be made.
- For SOL 600, it is required that all enforced displacements (other than motion of rigid contact surfaces applied using fields on the BCBODY entry) be applied using SPCD rather than SPC. The ID's of the SPCD must correlate with the Case Control LOAD or DLOAD entries.
- MPCs must be the same for all load cases.
- MAT10 is not presently supported.
- The following Solution Sequences are not presently supported: 107, 108, 110, 111, 114-116, 118, 144-146, 190, 200, 400 and 700 and will cause Severe Warnings (FATAL ERRORS) in the internal translator.
- DOMAINSOLVER is not supported. If this Executive Control statement is entered, and Nastran Implicit Nonlinear is requested by the SOL 600, ID statement, the DOMAINSOLVER request will be commented out internally by Nastran.
- IDs Grids, elements, properties, materials, etc. are limited to 9,999,999 unless one of the MRENUxxx items is specified in which case the limit is 10 digits.
- CGAP does not completely map to Marc's gap element. The user should change all Nastran gaps to contact before running SOL 600. If the gaps are not changed to contact, some options will fail to translate as indicated by warning messages. Certain simple gaps translate as expected and will produce nearly the same results as standard Nastran solution sequences, but the user is responsible for ensuring that any model with gaps gives the behavior he expects when using SOL 600.
- Nastran MATS1 Mohr-Coulomb is mapped to Marc's Linear Mohr Coulomb option.
- Nastran MATS1 Drucker-Prager is mapped to Marc's Parabolic Mohr Coulomb option.
- Solid element composite output is not presently available using the OUTF options, It must be postprocessed directly using the t16 file - Patran is recommended.
- If layered output for Composite Structures is desired, the following bulk data parameters or bulk data entry, MARCOUT with LAYCODE of 1 or 2 should be included in the Bulk Data. If output for all layers is desired, set LAYCODE to zero and enter the following parameters:

```
param,mroutlay,N
param,marcslht,N
```

where N is the maximum number of layers in any composite PCOMP description. The preferred option is MARCOUT with LAYCODE=1.

- The rotational acceleration portion of RFORCE (RACC) is not supported and if entered will generate a Severe Warning and Marc will not be spawned.

- The following Case Control option for displacement/velocity/acceleration/spcforce/ mpcforce are not supported and will be ignored if entered:
  - SORT2, REAL, IMAG, PHASE, PSDF, ATOC, CRMS, RALL, RPRINT, RPUNCH, NORPRINT, CID, TM, RM
- Elements with mid-side nodes must have all mid-side nodes. For example CTETRA must either have 4 or 10 nodes.
- For PC systems, if SOL 600 is run from a command prompt (DOS box), if any old DOS programs are used prior to running SOL 600 the path where the job is being run is usually adjusted such that any names longer than 8 characters will be shortened (for example brake-squeal becomes BRAKE-~3). The continue options including brake squeal jobs will not work when this happens. Open a new command prompt and run SOL 600 before any old DOS programs are run in that window.
- The CID field on the RFORCE entry is not completely supported. If entered with a positive integer, the job will abort with a Severe Warning unless PARAM,MARCRCID is entered. PARAM,MARCRCID,1 may be used to ignore this field in which case R1,R2,R3 define the direction cosines of the rotation vector (see Marc Volume C, ROTATION A description) and the magnitude is given by  $A*\sqrt{R1**2+R2**2+R3**2}$  [see RFORCE description for definitions of CID, A, R1, R2, R3 as well as Remark 16].
- Filenames entered on SOL 600 Bulk Data entries must be entered in small field fixed format, must be left-justified in the first applicable field and must be entered in lower case unless otherwise noted.
- For any jobs using the CONTINUE option or brake squeal, the jobid must be entirely in lower case for Linux systems.
- Ixy of PBAR/PBEAM is ignored, if entered, for SOL 600.
- PARAM,MRDISCMB=1 must be used for models with multiple subcases with the same pressure loadings in each subcase. The program will automatically attempt to reset the default (mrdiscmb=0 to mrdiscmb=1 in such circumstances, however, it is recommended that the user does this himself.
- Some Bulk Data input entries are not checked as completely for SOL 600 as they are for other solution sequences, particularly those that apply only to SOL 600. In addition, certain checks for all types of entries, even those that can be used in other solution sequences may be made after Marc is spawned or the error messages not output until after Marc is spawned. For those cases, the error message, if any, will not be visible until Marc has finished and may not even be output unless one or more of the OUTR options is selected. Users should take special care that the SOL 600 input is free from errors and that no duplicates occur prior to running SOL 600. One way to do this which is highly recommended is to run a preliminary SOL 101 job with as much of the same input file as possible.
- For SOL 600, it is required that all enforced displacements (other than motion of rigid contact surfaces described by fields in BCBODY entries) be applied using SPCD rather than SPC. The ID's of the SPCD must correlate with the Case Control LOAD or DLOAD entries.
- If an .op2, .xdb, .f06, and/or punch file is requested using the OUTR option, static analyses must have "times" ranging from 0.0-1.0 for the first subcase, 1.0-2.0 for the second subcase, etc. If NLAUTO is used to change these times, the job will fail with an appropriate message. This means that if NLAUTO is used TFINAL (field 4) must always be 1.0.
- Concentrated masses are not considered in gravity loading for linear or nonlinear static analyses.

- Non-structural mass is ignored by SOL 600.



## SOL 700,ID

## Executes MSC Nastran Explicit Nonlinear (SOL 700)

### Format:

SOL 700,ID PATH= STOP= NP(or DMP700)= FSIDMP= INTELMP=

### Examples:

SOL 700,129 PATH=3 NP=4

(700,129 request nonlinear transient dynamics, path=3 requests use of the SOL 700 script called out in file sol700.pth, np=4 requests that 4 processors be used)

### Summary:

SOL 700 is an Executive Control statement like SOL that activates an explicit nonlinear transient analysis integration scheme. The calculations will not be performed directly within MSC Nastran. Instead, SOL 700 will use a separate solver spawned from MSC Nastran. This client-server approach is similar to SOL 600, using Marc.

The SOL 700 statement will spawn an executable which is a 3D, explicit nonlinear analyses code DMP (distributed memory parallel processing domain decomposition) capabilities.

For ID=129 or NLTRAN, SOL 700 will generate an intermediate input data file, `jid.dytr.dat`, where “jid” is the name of the MSC Nastran input file without the extension). For example, if the MSC Nastran input file is named `abcd.dat`, (or `abcd.bdf`) then “jid”=abcd).

Unless specified differently using the **STOP=3** option, the executable will be executed from MSC Nastran on any computer system capable of doing so (which includes most Linux systems and Windows systems). For it to run, it must be installed, properly licensed, and accessible from the directory where the MSC Nastran input data resides, `MSC_BASE` must be provided in the environment.

### Nastran SOL 700 Update:

Starting in MSC Nastran 2019.0 there is a change in the execution of MSC Nastran SOL 700. SOL 700 with LS-Dyna is no longer supported by MSC Nastran. Instead, a new explicit solver is introduced in MSC Nastran 2019.0. This version can be activated by adding `VERSION = PRIMARY` to SOL 700 in the input file or simply by leaving it blank as this will be the default. A new license feature will be needed to run this new version: `NA_Explicit_Dytran`.

The new SOL 700 will support both DMP and SMP parallelization. For this, the solver will pull licenses from the following license feature: `NA_Parallel`.

Customers that have a lease agreement will no longer be able to run MSC Nastran SOL 700 jobs from version of MSC Nastran before 2018.0. The binaries needed were in the prior version located in `msc20221/dytran` directory of the MSC Nastran installation. However, this directory will be missing after the installation of MSC Nastran 2022.1.

Paid-up customers of MSC Nastran before 2018.0 can continue to use MSC Nastran SOL 700 jobs from version of MSC Nastran before 2018.0. For those customers, there will be no change in the FlexLM License features.

### Executive Control Parameters:

The required ID may be one of several valid solution sequence integers or names shown in [Table 2](#) for the SOL statement. Examples are 129 and NLTRAN.

The following solutions are available: 101, 106, 109, 129 (and their equivalent names).

All items on the SOL 700,ID after ID itself may be specified by environmental variables. This may be done any way environmental variables can be set. They may be set by the MSC Nastran user at run time or by the system administrator when MSC Nastran is installed. Any values specified on the SOL statement override those in the environment. Environmental variables are fully described in the . A keywords file is available to describe the format of each variable. The variable is normally set in the system-wide rc file, a user's rc file, a local rc file or in a script used to submit MSC Nastran.

The following describes the various options for PATH. We suggest that PATH=3 for all computer systems.

#### PATH=1 (Windows Only)

If PATH=1 is specified, MSC Nastran will determine the proper command to execute a **serial run**. To aid MSC Nastran in determining where Dytran is located, the **dynrun.pth** file must be located in the same directory where the MSC Nastran input file resides. The dynrun.pth file must contain one line providing the location (complete path) of the SOL 700 run script. A typical example of the line in the file dynrun.pth follows.

```
Windows          c:\sol700\
```

A string is appended to this path to form the complete command used to execute the SOL 700 executable. "dytran jid=name.dytr.dat"

For Windows, MSC Nastran will spawn the external executable using the following command assuming the MSC Nastran input data is named enf2e.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

```
c:\sol700/dytran jid=enf2e.dytr.dat
```

#### PATH=3 (All Systems)

If PATH=3 is specified, a script or batch file located in the same directory as the SOL 700 executable will be executed. The name of the executable is dytran (linux) or dytran.exe (Windows). This directory and name of the script is determined by the first line in a file named sol700.pth which must be in the same directory as the Nastran input file. Options are specified on subsequent lines of the sol700.pth file. For example, if Nastran is installed in C:\Program Files\MSC.Software\MSC\_Nastran\20221, the dytran.exe location is C:\Program Files\MSC.Software\MSC\_Nastran\2022.1\msc20221\dytran\win64\bin\dytran. To use sol700.pth file, the first line must be C:\Program Files\MSC.Software\MSC\_Nastran\20221\msc20221\dytran\win64\bin\dytran

Available PATH=3 options for Windows PC systems are as follows:

exe=	The full path to the executable that is to be used.  Optional -- If exe= is omitted, the directory where the script or batch file resides (first line of sol700.pth) will be used and dytran for Linux and dytran.exe for windows will be appended. If exe= is used, it must be the second line in the sol700.pth file.
nproc	Number of processors. (only for DMP run)  (Default is to used NP on the SOL 700 line. If NP and nproc are omitted, the default is 1). For parallel execution, the directory where the MSC Nastran input file exists must be shared with read/write privileges. If wdir is used, it must also be shared (see below). The directory where the Dytran executable resides must also be shared for parallel execution.
ncpus	Number of processors. (only for SMP)  (Default is not used.) If ncpus is greater than 1 in sol700.pth, Nastran SOL 700 automatically uses SMP capability.
bat	Run in background or foreground (Default).
hlist	Host file name. Name of a hostfile containing the same information as “machine” The format of hostfile is as follows for the example for machine: machine1 2 machine1 4
intelmpi	To activate the Intel MPI set intelmpi equal to yes. Default is no. If intelmpi=yes is placed directly on the command line (not in sol700.pth) Intel MPI will be activated too.
atb	the name of atb file.
imm	the name of imm file.

A Windows example of the file sol700.pth for the PATH=3 case follows.

```
C:\MSC.Software\MSC_Nastran\20221\msc20221\dytran\win64\bin\dytran nastran
nproc=4
```

For the above example, MSC Nastran will create the following command to spawn the SOL 700 executable assuming your input file is named abcd.dat. (Although the example appears like it is on multiple lines, it is actually on a single line.)

```
C:\Program Files\MSC.Software\MSC_Nastran\20221\msc20221\dytran\win64\bin\dytran
nastran nproc=6
```

Available PATH=3 options for Linux systems follows:

debug	Specifying debug=yes indicates if you want to keep scratch files and other debug information to investigate when a job fails to run. Default is no.
exe	The full path to the executable for Dytran that is to be used. (Optional)
fsidmp	Specifying fsidmp=yes indicates to run the FSI Distributed Memory Parallel version. Default is no. If FSIIDMP=YES is placed directly on the command line (not in sol700.pth) FSI Distributed Memory Parallel will be activated too.

hlist	The (local) filename containing the hosts list. If this file is not given or not found, a default local hosts list is used. Note that the MPI universe in which the selected nodes and CPUs reside is expected to exist and be accessible (i.e., be booted).
nproc	Number of processors. (only for DMP run)  (Default is to use NP on the SOL 700 line. If NP and nproc are omitted, the default is 1.) NOTE: The number of requested processes must be a power of 2.
ncpus	Number of processors. (only for SMP)  (Default is not used.) If ncpus is greater than 1 in sol700.pth, Nastran SOL 700 automatically uses SMP capability.
atb	the name of atb file.
imm	the name of imm file.

A Linux example of the file sol700.pth for the PATH=3 case is as follows:

```
/app/msc/msc20221/dytran/linux64/bin/dytran nastran nproc=4
```

For the above example, MSC Nastran will create the a command similar to the following to spawn the SOL 700 executable assuming your input file is named abcd.dat

```
/app/msc/msc20221/dytran/linux64/bin/dytran nastran nproc=4
```

If PATH is not specified, the default search path will be used to locate the dytran executable. This version will be located in a subdirectory named dytran/machine below the MSC Nastran base directory (MSC\_BASE). Not all PATH=3 options are available using this default path option.

## STOP

STOP is an optional item. STOP is used to prevent execution of Dytran or prevent execution of MSC Nastran after IFP if so desired. The various options are as follows:

### STOP=1

If STOP=1 MSC Nastran will gracefully stop after IFP. This option is used to prevent MSC Nastran from performing its own solution (normally used when the solution is performed with ID=129).

### STOP=3

STOP=3 MSC Nastran is stopped after IFP and Dytran is not executed. This would be the normal STOP option if the user wants to examine the intermediate input file, make some changes and then execute Dytran manually.

The following files are potentially affected by the COPYR option:

NP(or DMP700)=the Number of Processors

NP(or DMP700)=the number of processors (domains) for parallel processing. The default is one. In order to use more than one domain, MPI, Lam, POE, or whatever parallel program is needed must be properly installed on all computers involved and a hostfile designating which computers are to be used for each domain

must have been setup prior to running the job. If NP>1, PATH=3 is used and a file named sol700.pth is located in the same directory as the MSC Nastran input data. The sol700.pth file should contain all commands necessary to run in parallel. This file must have execute permissions.

Item	Case Control Commands Available in SOL 700
\$	Y
BCONTACT	Y
BEGIN BULK	Y (Other BEGIN forms are not allowed)
DLOAD	Y
ECHO	Y
ENDTIME	Y
ENDSTEP	Y
IC	Y
LOADSET	Y
\$S700	Y
SET	Y
SPC	Y
TITLE	Y
TSTEP	Y (Same as )
TSTEPNL	Y

The following summarizes the Bulk Data entries for SOL 700:

Item	Bulk Data Entries Available in SOL 700	Fatal Error
ABINFL	Y	
ATBACC	Y	
ATBJNT	Y	
ATBSEG	Y	
BARRIER	Y	
BCBODY	Y	
BCBODY1	Y	
BCBOX	Y	
BCELIPS	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
BCGRID	Y	
BCMATL	Y	
BCONNECT	Y	
BCONPRG	Y	
BCONPRP	Y	
BCPROP	Y	
BCSEG	Y	
BCTABL1	Y	
BCTABLE	Y	
BIAS	Y	
BJOIN	Y	
BSURF	Y	
CBAR	Y	
CBEAM	Y	
CDAMP1	Y	
CDAMP1D	Y	
CDAMP2	Y	
CDAMP2D	Y	
CELAS1	Y	
CELAS1D	Y	
CELAS2	Y	
CELAS2D	Y	
CHEXA	Y(8 Nodes only)	
CMARKB2	Y	
CMARKN1	Y	
COHFRIC	Y	
COMPUDS	Y	
CORD1C	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
CORD1R	Y	
CORD1S	Y	
CORD2C	Y	
CORD2R	Y	
CORD2S	Y	
CORD3R	Y	
COUCOHF	Y	
COUOPT	Y	
COUP1FL	Y	
COUPINT	Y	
COUPLE	Y	
CPENTA	Y(5 Nodes only)	
CQUAD4	Y	
CROD	Y	
CSPR	Y	
CTETRA	Y (4 Audio Nodes only)	
CTRIA3	Y	
CTUBE	Y	
CVISC	Y	
CYLINDR	Y	
DAREA	Y	
DETSPPH	Y	
DLOAD	Y	
DYFSISW	Y	
DYPARAM	Y	
ENDDATA	Y	
ENDDYNA	Y	
EOSDEF	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
EOSGAM	Y	
EOSIG	Y	
EOSJWL	Y	
EOSMG	Y	
EOSNA	Y	
EOSPOL	Y	
EOSTAIT	Y	
EOSUDS	Y	
EULFOR	Y	
EULFOR1	Y	
EULFREG	Y	
FAILJC	Y	
FAILMPS	Y	
FAILUDS	Y	
FFCONTR	Y	
FLOW	Y	
FLOWC	Y	
FLOWDEF	Y	
FLOWT	Y	
FLOWUDS	N	
FORCE	Y	
FORCE2	Y	
FORCUDS	Y	
GBAG	Y	
GBAGCOU	Y	
GRAV	Y	
GRDSET	Y	
GRID	Y	



Item	Bulk Data Entries Available in SOL 700	Fatal Error
HEATLOS	Y	
HGSUPPR	Y	
HTRCONV	Y	
HTRRAD	Y	
HYDSTAT	Y	
INCLUDE	Y	
INFLCG	Y	
INFLFRC	Y	
INFLGAS	Y	
INFLHB	Y	
INFLTNK	Y	
INFLTR	Y	
INITGAS	Y	
LOAD	Y	
LEAKAGE	Y	
LSEQ	Y	
MAT1	Y	
MAT2	Y	
MAT8	Y	
MATBV	Y	
MATDEUL	Y	
MATEP	Y	
MATF	Y	
MATFAB	Y	
MATHE	Y	
MATORT	Y	
MATRIG	Y	
MATVE	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
MESH	Y	
MOMENT	Y	
MOMENT2	Y	
NLOUTUD	Y	
PBAR	Y	
PBARL	Y	
PBEAM	Y	
PBEAML	Y	
PBELT	Y	
PCOMP	Y	
PDAMP	Y	
PELAS	Y	
PELAS1	Y	
PERMEAB	Y	
PERMGBG	Y	
PEULER	Y	
PEULER1	Y	
PLOAD	Y	
PLOAD2	Y	
PLOAD4	Y (Continuation supported)	
PMARKER	Y	
PMINC	Y	
PORFCPL	Y	
PORFGBG	Y	
PORFLOW	Y	
PORFLWT	Y	
PORHOLE	Y	
PORHYDS	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
PORUDS	Y	
PROD	Y	
PSHELL	Y	
PSHELL1	Y	
PSOLID	Y	
PVISC	Y	
PVISC1	Y	
RBE2	Y	
RBJOINT	Y	
RELEX	Y	
RFORCE	(CID, METHOD, continuation line not supported)	
SHREL	Y	
SHRPOL	Y	
SHRUDS	Y	
SPC	Y	
SPC1	Y	
SPCADD	Y	
SPHERE	Y	
SURFINI	Y	
TABLED1	Y	
TABLUDS	Y	
TIC	Y	
TIC3	Y (New Dytran type entry)	
TICEL	Y	
TICEUDS	Y	
TICEUL1	Y	
TICREG	Y	
TICVAL	Y	

Item	Bulk Data Entries Available in SOL 700	Fatal Error
TLOAD1	Y	
TLOAD2	N	
TODYNA	Y	
TSTEP	Y (Changed to TSTEPNL)	
TSTEPNL	Y	
WALL	Y	
WALL	Y (New rigid wall entry)	
YLDHY	Y	
YLDJC	Y	
YLDMSS	Y	
YLDPOL	Y	
YLDRPL	Y	
YLDSE	Y	
YLDTM	Y	
YLDUDS	Y	
YLDVM	Y	
YLDZA	Y	

## SPARSOLVER

## Sparse Solver Options

Specifies options used in sparse solution of equations operations.

Format:

SPARSOLVER {target} ([COMPMETH = {cmeth}][ORDMETH = {ometh}],

$$[FACTMETH = \{fmeth\}]MDTRATIO = \left( \left[ [NO]CHART \left\{ \begin{array}{l} T \\ R \end{array} \right\} [= nsegs] \right], \right.$$

$$\left. \left[ [NO]TABLE \left\{ \begin{array}{l} T \\ R \end{array} \right\} \right] \left[ NMAXRAT \left\{ \begin{array}{l} T \\ R \end{array} \right\} = nratio \right] \left[ MAXRAT \left\{ \begin{array}{l} T \\ R \end{array} \right\} = maxratio \right],$$

$$\left[ SORT = \left\{ \begin{array}{l} GRID \\ VALUE \\ BOTH \end{array} \right\} \right],$$

$$MDTSTATS = \left( \left[ [NO]CHART \left\{ \begin{array}{l} T \\ R \end{array} \right\} [= nsegs] \right], \right.$$

$$\left. \left[ [NO]TABLE \left\{ \begin{array}{l} T \\ R \end{array} \right\} \right] \left[ NMAXVAL \left\{ \begin{array}{l} T \\ R \end{array} \right\} = nmax \right] \left[ NMINVAL \left\{ \begin{array}{l} T \\ R \end{array} \right\} = nmin \right],$$

$$\left[ MAXVAL \left\{ \begin{array}{l} T \\ R \end{array} \right\} = vmax \right] \left[ MINVAL \left\{ \begin{array}{l} T \\ R \end{array} \right\} = vmin \right],$$

$$\left[ SORT = \left\{ \begin{array}{l} GRID \\ VALUE \\ BOTH \end{array} \right\} \right] )$$

Examples:

1. For the READ module, specify METIS reordering, and for the FRRD1 module, specify the UMFLU factorization method:

```
SPARSOLVER READ (ORDMETH=METIS)
SPARSOLVER FRRD1 (FACTMETH=UMFLU)
```

2. Request matrix diagonal term ratio output:

```
SPARSOLVER DCMP ( MDTRATIO )
```

- Request matrix diagonal term ratio output in chart format for translational DOFs, and limit the number of printed diagonal ratios to 50:

```
SPARSE Solver DCMP ( MDTRATIO=( CHART, TABLET, NMAXRATT=50) )
```

- Request Matrix Diagonal Term statistics output for:

- Chart format for translational and rotational DOF
- Table format for translational dof limited to largest 10 terms greater than 1.0E+08
- Table format for translational dof limited to smallest 20 terms smaller than 1.0
- Table format for rotational dof limited to smallest 30 terms smaller than 100.0

```
SPARSE Solver DCMP ( MDTSTATS=( CHART, TABLET, NMAXVALT=10,
MAXVALT=1.0E+08, NMINVALT=20, MINVALT=1.0, TABLER, NMINVALR=30,
MINVALR=100.00) )
```

Descriptor	Meaning
target	The target application for the options. The target application choices are: CEAD, DCMP, DECOMP, MDSTAT, NLSOLVE, READ, SOLVE, FRRD1, and TRD1.
cmeth	Compression method to be used (Default = GRID). The choices for compression method are: GRID, SUPER, GRDSUPER and NONE.
ometh	Ordering method to be used (Default = Automatic). The choices for ordering method are: AMF, BEND, MD, METIS, MMD, PRDMD, PRDMETIS, PRDSMPMS, and NONE. The default automatically selects among METIS, MMD and BEND
fmeth	Factorization method to be used (MSCLDL). The choices for factorization method are: MSCLDL, MSCLU, UMFLU, NUMFLU, PRDLDL, PRDLU and MUMPS. For default factorization methods, see Remarks 19.-24. below.
NO	No output is to be produced for the keyword.
T	The keyword applies to translational DOF only.
R	The keyword applies to rotational DOF only.
nsegs	Number of bars to be used in chart (Default = 0).
nratios	Number of diagonal term ratios to be included in table (Default = 25).
maxratio	Diagonal term ratio above which ratios are included in the table (Default = 1.0E+5).
GRID	Table output is sorted on GRID ID and component code.
VALUE	Sort table output on term values.
BOTH	Table is output once sorted on GRID ID and once on term value.
nmax	Number of largest values to print (Default = 25).

Describer	Meaning
nmin	Number of smallest values to print (Default = 25).
vmax	All terms larger than vmax are printed (Default = 1.0E+10).
vmin	All terms smaller than vmin are printed (Default = 1.0).

Remarks:

1. All of the keywords for the target application must be enclosed in parentheses.
2. The following table correlates target applications with available factorization methods.

Module	MSCLDL	MSCLU	UMFLU	NUMFLU	PRDLDL	PRDLU	MUMPS
CEAD	Yes	Yes	No	Yes	No	Yes	No
DCMP	Yes	Yes	No	No	Yes	No	Yes
DECOMP	Yes	Yes	No	No	No	No	No
FRRD1	Yes	Yes	Yes	No	No	Yes	No
MDSTAT	Yes	Yes	No	No	No	No	No
NLSOLV	Yes	Yes	No	No	Yes	No	Yes
READ	Yes	No	No	No	No	No	No
SOLVE	Yes	Yes	No	No	No	No	No
TRD1	Yes	Yes	Yes	No	No	No	No

3. More than one SPARSE Solver entry may be used, so that one may specify different options for different target modules.
4. System cell 206 can specify both a reordering method and a compression method by adding the respective values. For example, to specify GRDSUPER compression method with BEND reordering, SYSTEM(206) would be set to 68 (since 64+4=68). Or, to specify SUPER compression with METIS reordering, SYSTEM(206) would be set to 136 (since 128+8=136).
5. Note that SYSTEM(166), SYSTEM(206) and SYSTEM(209) have precedence over the SPARSE Solver command, whether set on the submittal line or in DMAP.
6. System cell 166 can also be used to turn on extra diagnostic output from the sparse factorization by setting SYSTEM(166)=2.
7. The choices for ordering method are: AMF, BEND, MD, METIS, MMD, PRDMD, PRDMETIS, PRDSMPMS, and NONE. The default automatically selects among METIS, MMD and BEND.
8. The UMFLU factorization method ignores COMPmeth and ORDMeth since it contains its own compression/reordering method.
9. The matrix diagonal term ratio (MDTRATIO) and value (MDTSTATS) keywords and options are used only by the DCMP target application. They will be ignored by other target applications.

10. The MDTRATIO and MDTSTATS keywords apply to both translational and rotational degrees of freedom unless modified by the T or R describer. Separate output is produced for each of the translational and rotational degrees of freedom in the matrix.
11. The MDTRATIO and MDTSTATS CHART option default produces a chart of values contained in powers of ten bandwidths. A specification for the nsegs describer causes the bandwidth to be internally computed to produce nsegs bars. Bars are produced for the bandwidths only if they contain terms.
12. MDTRATIO and MDTSTATS keywords are processed in the order given. It is possible for a keyword to modify the effects of a previously processed keyword. For example, the combination CHART, CHARTR=5 produces different outputs compared to CHARTR=5, CHART.
13. The MDTRATIO NMAXRAT describer limits the content of the TABLE output to the specified highest number of diagonal ratios that exceed the value of the MAXRAT describer.
14. MDTSTATS generates output for matrix diagonal term values. For the TABLE option, it produces a table containing the NMINVAL=nmin smallest terms smaller than MINVAL=vmin as well as the NMAXVAL=nmax largest terms larger than MAXVAL=vmax.
15. The ORDMETH option must be one of PRDMD, PRDMETIS, or PRDSMPMS for PRDLDL and PRDLU.
16. The COMPMETH option is ignored for PRDLDL and PRDLU.
17. The DOMAINSOLVER (RUNOPT=MULTIPAR) option is not supported with PRDLDL or PRDLU.
18. For PRDLDL and PRDLU, if there is not sufficient memory for an in-core solver, then an out-of-core solver will be attempted and performance will be negatively affected.
19. The following special rules apply for PRDLDL for NLSOLV:
  - a. The maximum number of threads used is given by DMP\*SMP, and PRDLDL will choose a number of threads, up to this maximum, most suitable to the problem size.
  - b. PRDLDL is supported only when FNT or PFNT method is used for nonlinear iteration (KMETHOD of NLPARM or NLSTEP).
  - c. If the problem fails to converge with PRDLDL, then add MDLPRM,LMT2MPC,1 to the bulk data section in order to convert Lagrange multipliers to MPCs.
20. The following special rules apply for PRDLDL for DCMP:
  - a. DMP is not supported for PRDLDL for DCMP as it only gets parallelism from SMP
  - b. PRDLDL automatically provides parallelization over multiple RHSs with SMP.
  - c. If PRDLDL encounters a zero pivot issue (SFM 11332), the user should rerun with bulk data entry MDLPRM,PRDMTYPE,-2 to have Intel MKL Pardiso regard the matrix as indefinite, and entry MDLPRM,PRDWMATCH,0 in order to turn off weighted matching.
21. The following special rules apply for PRDLU for CEAD:
  - a. PRDLU is only supported for the CLAN option within CEAD (refer to CMETHOD option).
22. The following special rules apply for PRDLU for FRRD1:
  - a. The Krylov solver is not supported with PRDLU for FRRD1.



- b. DMP is supported for PRDLU for FRRD1.
  - c. MKL Pardiso is not supported in the uncoupled solution algorithm for FRRD1.
  - d. MKL Pardiso is the default solver for FRRD1 except for the following cases:
    - SOL 111 symmetric solution without frequency dependence, in which case MSCDDL is used.
    - Iterative solver specified by user.
23. For CEAD, the default solver for unsymmetric matrices is UMFPACK while for Hermitian matrices the default is MSCLU.
  24. User specified factorization method (fmeth) via SPARSESolver entry or SYSTEM(209) may be overwritten automatically due to matrix type or symmetry by MSC Nastran.
  25. Several options for Intel MKL Pardiso are controlled using the [MDLPRM](#) bulk data entry. See the PRD\* parameters detailed in the MDLPRM section.
  26. Mumps sparse direct solver with DCMP and NLSOLV module is currently supported in SOL 101 only.

## **TIME**                      Sets Maximum CPU and I/O Time

Sets the maximum CPU and I/O time.

Format:

TIME[=]t1[,t2]

Examples:

1. The following example designates a runtime of 8 hours:  
TIME 480
2. The following example designates 90 seconds:  
TIME 1.5

Describer	Meaning
t1	Maximum allowable execution time in CPU minutes (Real or Integer>0; Default=1.89E9 seconds).
t2	Maximum allowable I/O limit in minutes (Real or Integer>0; Default is infinity, which is machine dependent).

Remarks:

1. The TIME statement is optional.
2. If t2 is specified then t1 must be specified.

# 5

## Case Control Commands

- Key to Descriptions
- The Case Control Section
- Case Control Command Summary
- Case Control Commands
- Case Control Applicability Tables
- OUTPUT(PLOT) Commands
- X-Y PLOT Commands
- OUTPUT(POST) Commands

## Key to Descriptions

Brackets [ ] indicate that a choice of descriptors is optional.

A brief description of the command is given.

Describes in uppercase letters are keywords that must be specified as shown.

Describes in lower case are variables.

Braces { } indicate that a choice of descriptors is mandatory.

If the descriptors are stacked vertically, then only one may be specified.

The default descriptors are shaded.

Requests the form and type of applied load vector output.

**Format:**

$$OLOAD \left( \left( \left[ \begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix} \right] \right), \text{PUNCH}, \left[ \text{PHASE} \right] \right) = \left\{ \begin{matrix} \text{ALL} \\ n \\ \text{NONE} \end{matrix} \right\}$$

**Examples:**

OLOAD = ALL  
OLOAD(SORT1,PH)

Descriptors	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency value, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT	The printer will be the output medium.
PUNCH	The punch card will be the output medium.
REAL or IMAG	Requests real or imaginary components of the load. If both are specified, either REAL or IMAG yields the same output.
PHASE	Requests polar angle of the load. If the descriptor is in lower case, then it is a variable and the descriptor's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, the descriptor must be specified by the user.
ALL	Applied loads from all cases.
NONE	Applied load from no cases.
n	Set identification of a previously appearing SET command. Only loads on points whose identification numbers appear on this SET command will be output. (Integer > 0).

**Remarks:**

- Both PRINT and PUNCH may be specified.
  - See the MSC Nastran User Manual for details on SORT1 and SORT2 formats and their default output.
  - In a statics problem, a request for output (frequency and/or time) is required (nonzero) to be output.
- The remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command, the command's relationship to other commands, restrictions and recommendations on its use, and further details regarding the descriptors.



## The Case Control Section

The Case Control Section has several basic functions. Specifically, it:

- Selects loads and constraints.
- Requests printing, plotting, and/or punching of input and out data (plotter commands are described in [Plotting](#) in the *MSC Nastran Reference Guide*).
- Defines the subcase structure for the analysis.

[Table 5-3](#) through [Table 5-8](#) at the end of this section indicate the applicability of each command in all solution sequences.

## Case Control Command Descriptions

Case Control commands may be abbreviated down to the first four characters if the abbreviation is unique relative to all other commands. If not, the full name of the command (or at least the first eight characters if the name exceeds eight characters) must be specified in order to avoid errors. Each command is described as follows:

### Description

A brief sentence about the function of the command is given.

### Format

Describers in uppercase are keywords that must be specified as shown. In addition, describers in lowercase indicate that the user must provide a value.

Braces { } indicate that a choice of describers is mandatory. If the describers are stacked vertically, then only one may be specified.

Brackets [ ] indicate that a choice of describers is optional. If the describers are stacked vertically, then only one may be specified.

Describers that are shaded indicate the defaults.

If the command line is longer than 72 columns, then it may be continued to the next line with a comma. For example:

```
SET 1 = 5, 6, 7, 8, 9,  
10 THRU 55
```

### Example

A typical example is given.

### Describers and Meaning

Each of the describers is briefly discussed. The describer's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The describer must be specified by the user if no default value is given.

### Remarks

The remarks are generally arranged in order of importance and indicate such things as which Bulk Data entries are selected by the Case Control command; the command's relationship to other commands, restrictions and recommendations on the command's use; and further descriptions of the describers.

## Case Control Command Summary

This section contains a summary of all Case Control commands under the following headings:

### Subcase Definition

#### 1. Output Request Delimiters

<a href="#">OUTPUT (Case)</a>		Delimits the various types of commands for the structure plotter, curve plotter and grid point stress.
OUTPUT(PLOT)		Beginning of structure plotter output request.
OUTPUT(POST)	}	Indicates beginning of grid point stress output requests.
or SETS DEFINITION		
OUTPUT(XYOUT)	}	Indicates beginning of curve plotter output request.
or OUTPUT(XYPLOT)		

#### 2. Subcase Delimiters

<a href="#">REPCASE (Case)</a>	Delimits and identifies a repeated output request subcase.
<a href="#">SUBCASE (Case)</a>	Delimits and identifies a subcase.
<a href="#">SUBCOM (Case)</a>	Delimits and identifies a combination subcase.
<a href="#">SYM (Case)</a>	Delimits and identifies a symmetry subcase.
<a href="#">SYMCOM (Case)</a>	Delimits and identifies a symmetry combination subcase.

#### 3. Subcase Control

<a href="#">MASTER (Case)</a>	Allows the redefinition of a MASTER subcase.
<a href="#">MODES (Case)</a>	Repeats a subcase.

<a href="#">SUBSEQ (Case)</a>	Gives the coefficients for forming a linear combination of the previous subcases.
<a href="#">SYMSEQ (Case)</a>	Gives the coefficients for combining the symmetry subcases into the total structure.

## Data Selection

### 1. Static Load Selection

<a href="#">DEFORM (Case)</a>	Selects the element deformation set.
<a href="#">CLOAD (Case)</a>	Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.
<a href="#">LOAD (Case)</a>	Selects an external static loading set.
<a href="#">LOADNAME (Case)</a>	Provides a name to be associated with a loading condition.

### 2. Dynamic Load Selection

<a href="#">DLOAD (Case)</a>	Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.
<a href="#">LOADSET (Case)</a>	Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.
<a href="#">NONLINEAR (Case)</a>	Selects nonlinear dynamic load set for transient problems.

### 3. Constraint Selection

<a href="#">AXISYMMETRIC (Case)</a>	Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.
<a href="#">AUTOSPC (Case)</a>	Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.
<a href="#">BC (Case)</a>	Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.
<a href="#">DSYM (Case)</a>	Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.
<a href="#">MPC (Case)</a>	Selects a multipoint constraint set.
<a href="#">SPC (Case)</a>	Selects a single-point constraint set to be applied.

**STATSUB (Case)** Selects the static solution to use in forming the differential stiffness for buckling, normal modes, complex eigenvalue, frequency response, and transient response analysis.

**SUPPORT1 (Case)** Selects the fictitious support set (SUPPORT1 entries only) to be applied to the model.

#### 4. Thermal Field Selection

**TEMPERATURE (Case)** Selects the temperature set to be used in either material property calculations, or thermal loading in heat transfer and structural analysis.

**TEMPERATURE (Case) (INITIAL)** Selects initial temperature distribution for temperature-dependent material properties and heat transfer problems.

**TEMPERATURE (Case) (LOAD)** Selects temperature set for static thermal load.

**TEMPERATURE (Case) (MATERIAL)** Selects temperature set for temperature-dependent material properties.

**TSTRU (Case)** Defines a temperature set ID for a structures run based on a heat transfer subcase.

#### 5. Static Solution Conditions

**SMETHOD (Case)** Selects iterative solver parameters.

#### 6. Dynamic Solution Conditions

**CMETHOD (Case)** Selects complex eigenvalue extraction parameters.

**FREQUENCY (Case)** Selects the set of forcing frequencies to be solved in frequency response problems.

**FRF (Case)** Specifies data for Frequency Response Function (FRF) generation or for the FRF Based Assembly (FBA) process.

**IC (Case)** Selects the initial conditions for direct transient analysis (SOLs 109, 129, and 159).

**METHOD (Case)** Selects the real eigenvalue extraction parameters.

**MODESELECT (Case)** Requests a set of computed mode shapes for inclusion in dynamic analysis.

**NSM (Case)** Selects nonstructural mass set.



<a href="#">RANDOM (Case)</a>	Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.
<a href="#">RESVEC (Case)</a>	Specifies options for and the calculation of residual vectors.
<a href="#">RGYRO (Case)</a>	Activates gyroscopic effects and selects RGRYO or UNBALNC entries.
<a href="#">SDAMPING (Case)</a>	Requests modal damping as a function of natural frequency in modal solutions, or viscoelastic materials as a function of frequency in direct frequency response analysis.
<a href="#">SMETHOD (Case)</a>	Selects iterative solver override options in frequency response analysis.
<a href="#">TSTEP (Case)</a>	Selects integration and output time steps for linear or nonlinear transient analysis.

## 7. Direct Input Matrix Selection

<a href="#">A2GG (Case)</a>	Selects direct input fluid-structure coupling matrix.
<a href="#">B2GG (Case)</a>	Selects direct input damping matrices.
<a href="#">B2PP (Case)</a>	Selects direct input damping matrices.
<a href="#">K2GG (Case)</a>	Selects direct input stiffness matrices.
<a href="#">K2PP (Case)</a>	Selects direct input stiffness matrices, which are not included in normal modes.
<a href="#">K42GG (Case)</a>	Selects direct input structural damping matrices.
<a href="#">M2GG (Case)</a>	Selects direct input mass matrices.
<a href="#">M2PP (Case)</a>	Selects direct input mass matrices, which are not included in normal modes.
<a href="#">MFLUID (Case)</a>	Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.
<a href="#">P2G (Case)</a>	Selects direct input load matrices.
<a href="#">TFL (Case)</a>	Selects the transfer function set(s) to be added to the direct input matrices.

## 8. Nonlinear Analysis

<a href="#">ENDSTEP (Case)</a>	Specifies final analysis step for SOL 700.
<a href="#">ENDTIME (Case)</a>	Specifies final analysis time for SOL 700.
<a href="#">NLBUCK (Case)</a>	Perform a nonlinear buckling analysis in SOL 400.
<a href="#">NLHARM (Case)</a>	Selects the parameters used for nonlinear harmonic response analysis.

NLOPRM (Case)	Controls MSC Nastran nonlinear output, debug printout and debug POST.
NLPARM (Case)	Selects the parameters used for nonlinear static analysis.
NLSTEP (Case)	Selects integration and output time steps for static and transient nonlinear analysis in SOL 400.
STEP (Case)	Defines and identifies a nonlinear analysis for SOL 400.
SUBSTEP (Case)	Delineates and identifies a nonlinear analysis SUBSTEP for COUPLED analysis in SOL 400.
TSTEPNL (Case)	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, 600 and SOLs 400 and 700.

## 9. Aerodynamic Analysis

AECONFIG (Case)	Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.
AESYMX Y (Case)	Aerodynamic XY plane of symmetry flag.
AESYMXZ (Case)	Aerodynamic XZ plane of symmetry flag.
AEUXREF (Case)	Define the reference aerodynamic extra point (controller) vector
CSSCHD (Case)	Aerodynamic control surface schedule.
DIVERG (Case)	Selects the divergence parameters in a static aeroelastic divergence problem.
FMETHOD (Case)	Selects the parameters to be used in the aerodynamic flutter analysis.
GUST (Case)	Selects the field in an aerodynamic response problem.
TRIM (Case)	Selects trim variable constraints in static aeroelastic response.

## 10. Design Sensitivity and Optimization (SOL 200)

ANALYSIS (Case)	Specifies the type of analysis being performed for the current subcase.
AUXCASE (Case)	Delimits Case Control commands for an auxiliary model in SOL 200.
AUXMODEL (Case)	References an auxiliary model for generation of boundary shapes in shape optimization.
DESGLB (Case)	Selects the design constraints to be applied at the global level in a design optimization task.
DESMOD (Case)	Assigns the design model parameter used to locate the associated datablocks for merging of two or more SOL 200 using the MultiOpt toolkit application.

<a href="#">DESOBJ (Case)</a>	Selects the DRESP1 or DRESP2 entry to be used as the design objective.
<a href="#">DESSUB (Case)</a>	Selects the design constraints to be used in a design optimization task for the current subcase.
<a href="#">DESVAR (Case)</a>	Selects a set of DESVAR entries for the design set to be used.
<a href="#">DRSPAN (Case)</a>	Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcase.
<a href="#">DSAPRT (Case)</a>	Specifies design sensitivity output parameters.
<a href="#">MODTRAK (Case)</a>	Selects mode tracking options in design optimization (SOL 200).

### 11. Adaptive Meshing

<a href="#">HADAPT (Case)</a>	Specifies Mesh adaptivity control parameters.
-------------------------------	---

### 12. Fluid-Structure Analysis

<a href="#">A2GG (Case)</a>	Selects a direct input fluid-structure coupling matrix.
<a href="#">ACFPMRESULT (Case)</a>	Requests output of acoustic field point mesh results.
<a href="#">ACPOWER (Case)</a>	Request output of the power radiated from the wetted surface.
<a href="#">FLSFSEL (Case)</a>	Fluid-structure parameter collector for frequency and fluid superelement selection.
<a href="#">FLSPOUT (Case)</a>	Fluid-structure parameter collector for mode participation.
<a href="#">FLSTCNT (Case)</a>	Fluid-structure parameter collector for symmetry and force request.
<a href="#">INTENSITY (Case)</a>	Requests output of acoustic intensity on wetted surface.
<a href="#">TRIMGRP (Case)</a>	Selection of Trim Component(s)

### 13. Nastran/ADAMS Interface

<a href="#">ADAMSMNF* (Case)</a>	Control for Nastran/ADAMS interface modal neutral file (MNF).  For MSC Nastran 2004, to ensure compatibility with the Adams msc2mnf tool kit, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any output2 files generated will have a pre-MS Nastran 2004 format.
----------------------------------	---

### 14. Contact

<a href="#">BCHANGE (Case)</a>	Selects the change of the definition contact bodies in contact analysis.
<a href="#">BCMOVE (Case)</a>	Contact body movement selection in contact analysis.
<a href="#">BCONCHK (Case)</a>	Requests contact model check.
<a href="#">BCONTACT (Case)</a>	Requests contact analysis.
<a href="#">BOUTPUT (Case)</a>	Requests output for contact analysis.
<a href="#">BSQUEAL (Case)</a>	Selects data for brake squeal analysis
<a href="#">UNGLUE (Case)</a>	Selects the grids should use standard contact instead of glued contact in glued bodies.

## 15. Monte-Carlo simulation

<a href="#">MONCARL (Case)</a>	Control for Monte-Carlo simulation.
--------------------------------	-------------------------------------

## Output Selection

### 1. Output Control

<a href="#">ECHO (Case)</a>	Controls echo (i.e., printout) of the Bulk Data.
<a href="#">ECHOOFF</a>	Suppresses echo of Case Control.
<a href="#">ECHOON</a>	Reactivates echo of Case Control.
<a href="#">LABEL (Case)</a>	Defines a character string that will appear on the third heading line of each page of printer output.
<a href="#">LINE (Case)</a>	Defines the maximum number of output lines per printed page.
<a href="#">MAXLINES (Case)</a>	Sets the maximum number of output lines.
<a href="#">PAGE (Case)</a>	Causes a page eject in the echo of the Case Control Section.
<a href="#">PLOTID (Case)</a>	Defines a character string that will appear on the first frame of any plotter output.
<a href="#">POST (Case)</a>	Activates postprocessor operations for selected output data.
<a href="#">SKIP (Case)</a>	Activates or deactivates the execution of subsequent commands in the Case Control Section (including plot commands).
<a href="#">SKIPON</a>	Defines commands in the Case Control Section that are not to be processed.
<a href="#">SKIPOFF</a>	Resumes processing of commands in the Case Control Section.

<a href="#">SUBTITLE (Case)</a>	Defines a subtitle that will appear on the second heading line of each page of printer output.
<a href="#">TITLE (Case)</a>	Defines a character string that will appear on the first heading line of each page of MSC Nastran printer output.

## 2. Set Definition

<a href="#">MAXMIN (Case)</a>	Specifies options for max/min surveys of certain output data associated with grid points.
<a href="#">OFREQUENCY (Case)</a>	Selects a set of frequencies for output requests.
<a href="#">OMODES (Case)</a>	Selects a set of modes for output requests.
<a href="#">OTIME (Case)</a>	Selects a set of times for output requests.
<a href="#">PARTN (Case)</a>	Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17). In SOLs 111 and 200, the PARTN Case Control command specifies the points at which modal participation factors are to be computed.
<a href="#">SET (Case)</a>	Defines a set of element or grid point numbers to be plotted.
<a href="#">SURFACE</a>	Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.
<a href="#">VOLUME</a>	Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

### 3. Physical Set Output Requests

ACCELERATION (Case)	Requests the form and type of acceleration vector output.
BOUTPUT (Case)	Requests line or 3D (SOLs 101, 106, 129, 153, 159, 400, and SOL 600) contact output.
CMSENRGY (Case)	Requests the output of component (superelement) modal strain, kinetic, and damping energies.
DISPLACEMENT (Case), VECTOR (Case), or PRESSURE (Case)	Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.
EDE (Case)	Requests the output of the energy loss per cycle in selected elements.
EKE (Case)	Requests the output of the kinetic energy in selected elements.
ELSDCON (Case)	Requests mesh stress discontinuities based on element stresses (see STRESS).
ENTHALPY (Case)	Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).
ESE (Case)	Requests the output of the strain energy in selected elements.
EQUILIBRIUM (Case)	Requests equilibrium force balance output.
FATIGUE (Case)	Request a fatigue analysis with life/damage output in SOLs 101, 103, 108, 111, or 112.
FLUX (Case)	Requests the form and type of gradient and flux output in heat transfer analysis.
FORCE (Case) or ELFORCE	Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.
GPFORCE (Case)	Requests grid point force balance at selected grid points.
GPKE (Case)	Requests the output of the kinetic energy at selected grid points in normal modes analysis only.
GPSDCON (Case)	Requests mesh stress discontinuities based on grid point stresses (see GPSTRESS).
GPSTRAIN (Case)	Requests grid points strains for printing only.
GPSTRESS (Case)	Requests grid point stresses for printing only.
GVECTOR (Case)	Requests the form and type of g-set eigenvector output in SOLs 200 and 400.
HDOT (Case)	Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).
MCFRACTION (Case)	Requests modal contribution fractions output.

MODALKE (Case)	Requests modal kinetic energy output.
MODALSE (Case)	Requests modal strain energy output.
MEFFMASS (Case)	Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.
MPCFORCES (Case)	Requests the form and type of multipoint force of constraint vector output.
NLSTRESS (Case)	Requests the form and type of nonlinear element stress output in SOL 106.
NOUTPUT (Case)	Requests physical output in cyclic symmetry problems.
OLOAD (Case)	Requests the form and type of applied load vector output.
RCROSS (Case)	Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.
ROTSEKE (Case)	Request rotor modal strain and kinetic energies in complex eigenvalue analysis for selected rotor IDs.
SPCFORCES (Case)	Requests the form and type of single-point force of constraint vector output.
STRAIN (Case)	Requests the form and type of strain output.
STRESS (Case) or ELSTRESS	Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.
STRFIELD (Case)	Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.
THERMAL (Case)	Requests the form and type of temperature output.
VELOCITY (Case)	Requests the form and type of velocity vector output.

#### 4. Solution Set Output Requests

AEROF (Case)	Requests the aerodynamic loads on aerodynamic control points.
APRESSURE (Case)	Requests the aerodynamic pressures in static aeroelastic response.
HARMONICS (Case)	Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.
HOUTPUT (Case)	Requests harmonic output in cyclic symmetry problems.
MPRES (Case)	Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.
NLOAD (Case)	Requests the form and type of nonlinear load output for transient problems.

<b>SACCELERATION (Case)</b>	Requests the form and type of solution set acceleration output.
<b>SDISPLACEMENT (Case)</b>	Requests the form and type of solution set displacement output.
<b>SVECTOR (Case)</b>	Requests the form and type of solution set eigenvector output.
<b>SVELOCITY (Case)</b>	Requests the form and type of solution set velocity output.
<b>TRIMF (Case)</b>	Specifies options for the output of trim loads from a static aeroelastic analysis as FORCE/MOMENT Bulk Data entries.

## 5. Model Checkout

<b>ELSUM (Case)</b>	Requests a summary of element properties for output.
<b>GROUNDCHECK (Case)</b>	Requests grounding check analysis on stiffness matrix to expose unintentional constraints by moving the model rigidly.
<b>WEIGHTCHECK (Case)</b>	At each stage of the mass matrix reduction, computes rigid body mass and compares with the rigid body mass t the g-set.

## Superelement Control

<b>EXTDRIN (Case)</b>	Requests the job to perform an external superelement data recovery restart. Also specifies the storage media of the boundary solution data.
<b>EXTDROUT (Case)</b>	Requests the job to store external superelement boundary displacements and column labels; e.g., eigenvalues, forcing frequencies, time steps. Also specifies the storage media of the boundary solution data.
<b>EXTSEOUT (Case)</b>	Specifies the data to be saved for an external superelement, and the medium on which the data is to be saved.
<b>LDLABEL (Case)</b>	Assigns a character string to an external superelement creation run's subcase in order to identify the static load in the assembly run.
<b>SEALL (Case)</b>	Specifies the superelement identification numbers of phase 1 processing wherein all matrices and loads are generated and assembled. Controls execution of the solution sequence.
<b>SEDR (Case)</b>	Specifies the superelement identification numbers for which data recovery will be performed.
<b>SEDV (Case)</b>	Specifies the superelement identification numbers for which the design variables will be processed.
<b>SEEXCLUDE (Case)</b>	Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.



SEFINAL (Case)	Specifies the superelement identification number of the final superelement to be assembled.
SEKREDUCE (Case)	Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.
SELGENERATE (Case)	Specifies the superelement identification numbers for which static loads will be generated.
SELREDUCE (Case)	Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.
SEMGENERATE (Case)	Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.
SEMREDUCE (Case)	Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices are assembled and reduced.
SERESP (Case)	Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.
SOLUTION (Case)	Selects the solution ID for a 3 <sup>rd</sup> step external superelement data recovery restart in SOL 400.
SUPER (Case)	Assigns a subcase(s) to a superelement or set of superelements.

## Miscellaneous

\$	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
BEGIN BULK (Case)	Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.
INCLUDE (Case)	Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.
NSM (Case)	Request nonstructural mass distribution selection.
OUTPUT (Case)	Delimits the various types of commands for the structure plotter, curve plotter, grid point stress, and MSGSTRESS.
PARAM (Case)	Specifies values for parameters.
POST (Case)	Controls selection of data to be output for postprocessing.
RIGID (Case)	Selects type of rigid element formulations to be used.

## Case Control Commands

### \$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

For SOL 700 only, the string '\$S700' (starting in column 1 with a blank in column 6) may be used to transfer any Case Control string directly to Dytran. For example:

`$S700 ELOUT (TEST1) =XVEL, YVEL, ZVEL, DENSITY, SIE, PRESSURE, fmat`  
will become

`ELOUT (TEST1) =XVEL, YVEL, ZVEL, DENSITY, SIE, PRESSURE, fmat`

in the Case Control Section of the Dytran input (jid.dytr.dat) file. Please see descriptions of '\$S700' Case Control entries elsewhere in this section.

#### **Format:**

\$ followed by any characters out to column 80.

#### **Example:**

```
$ TEST FIXTURE-THIRD MODE
```

#### **Remarks:**

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.

**\$\$700**

## Direct Case Control Command for SOL 700

All entries which start with \$\$700 in the Case Control Section will be passed directly to the SOL 700 solver. Import of Euler results and the control of output generation are using this entry:

**Output Control:**

No need to use the Nastran classical case control output (STRESS,DISP,...). This option controls ARC (state file) and THS (time history file) output files.

You need to specify the following for a complete output specification:

1. Type of the file
2. What entities (e.g., grid points, elements, rigid bodies, etc.)
3. What results are output
4. How often it is saved
5. How often data is written

**Format:**

```

$$700 TYPE(logical_file) = ARCHIVE
$$700 Entity Type(logical_file) = setid
$$700 SET setid = idi
$$700 Entity Var(logical_file) = var
$$700 TIMES(logical_file) = ti
$$700 STEPS(logical_file) = stepi
$$700 SAVE(logical_file) = n

```

**Example:**

```

$$700 TYPE (elements) = ARCHIVE
$$700 ELEMENTS (elements) = 1
$$700 SET 1 = 16805 THRU 16875 16877 THRU 16947 16949 THRU 18080 ,
$$700          59 THRU 945 1067 1068 1069 1070 1181 THRU 1412 ,
$$700          1423 THRU 2965 3076 THRU 3930 4041 THRU 4895 ,
$$700          5006 THRU 5860 5971 THRU 7744 10855 THRU 12025 ,
$$700          12138 THRU 12990 13103 THRU 13955 14068 THRU 14920 ,
$$700          15033 THRU 15885 15998 THRU 16875 16877 THRU 16947 ,
$$700          16949 THRU 18080 16805 THRU 16875 16877 THRU 16947 ,
$$700          17005 THRU 18080 16949 THRU 17004 BY 1
$$700 ELOUT (elements) = EFFSTS, PRESSURE, FMAT, YMOM ,
$$700          QDIS,
$$700          MASST
$$700 TIMES (elements) = 0,THRU,END,BY,4e-4
$$700 SAVE (elements) = 1

```

Describer	Meaning
logical_file	The logical name of the file to which the user output is written. The logical name may not contain any spaces or special characters and must be restricted to 8 characters or less. (Characters; Required)
setid	Number of a SET command. Only data for elements that appear in the set are output. (Integer > 0; Required)
Idi	Identification numbers at which the output is requested. “THRU” and “BY”; for the “THRU” and “BY” options, respectively. See Remark 2. (Integer > 0 or characters; Required)
Var	Variable name to be output. See Remark 5. (Characters; Required)
Ti	Times at which output is required. “THRU” and “BY”; for the “THRU” and “BY” options, respectively. “END” indicates the end time of analysis. See Remarks 3. and 4. (Real $\geq 0.0$ or characters; Required)
stepi	Steps at which output is required. “THRU” and “BY”; for the “THRU” and “BY” options, respectively. See Remarks 3. and 4. (Integer $\geq 0$ or characters; Required)
N	The number of times an output file is written before it is closed and saved. (Integer > 0; Required)

### Remarks:

1. Default of ARC output control is set to the ARC file generation of all Eulerian elements with all available variables.
2. Continuation lines are supported for SET, ELOUT, TIMES and STEPS. The continuation lines must start with at least two spaces after the \$\$S700 string. A comma (,) at the end of a line signifies that the next line is a continuation.
3. Either STEPS or TIMES must be specified. STEPS and TIMES can not be used for the same logical\_file.
4. A list of times or steps should be in ascending order.
5. Check [Chapter 11: Outputting Results](#) in *MSC Nastran SOL700 User's Guide*. It includes all Entity types, Entity variables and Variable names.

### Import Euler results:

Specifies an Euler archive that was created during a previous simulation and is used as input for a transient analysis. The Euler archive is mapped onto a set of Euler elements that can equal in size or either finer or coarser.

### Format:

```
$$S700 EULINIT, filename,CYCLE,MESH-ID
```

### Example:

```
$$S700 EULINIT,AXISYM_ALLEULER_0.ARC,60
```



Describer	Meaning
filename	The filename of the ARC file to be used.
CYCLE	Cycle number.
MESH-ID	The MESH ID of the target elements.

### Remarks:

1. The target elements are the elements defined in the follow-up run and are the elements that will be initialized using the import archive.
2. Both defined and imported Euler elements need to be orthogonal in the global system.
3. MESH-ID enables support for multiple Euler domains. If MESH-ID is not set the import archive will be mapped onto all Euler elements.
4. For multi-material Euler analyses with multiple Eulerian materials all material variables in the import archive require the material number. These material variables are MASS, DENSITY, SIE, FMAT. The required list of variables for a MMHYDRO run are: MASSXX, SIEXX, FMATXX, XVEL, YVEL, ZVEL. Here XX denotes the material number. FOR MMSTREN the variables TXX, TYY, TZZ, TXY, TYZ, TZX, EFFPLS have to be added. If the multi-material run uses only one Eulerian material then the material numbers can be left out.
5. For multi-material Euler analyses with EOSIG, the following has to be added for the IG materials: MASS-EXX, MASS-PXX, RHO-EXX, RHO-PXX, IGBURNXX, FMAT-PXX, FMAT-PXX, SIE-EXX, and SIE-PXX. Here XX denotes the material number
6. For the single-material Hydro Euler solver the required list is MASS, DENSITY, SIE, FMAT, and FVUNC.
7. IF FVUNC is not included in the Import archive it is assumed that all elements in this archive are fully uncovered. It is allowed to import such an archive in a simulation with a coupling surface. In this follow up simulation the target elements can have uncover fractions different from one. In this case conservative quantities of imported elements are reduced by the uncover fraction of the target element. This is to avoid unwanted pressure increases. It simply means that any mass of the import archive that is located in the covered part of the target elements is thrown away. As a result not all mass in the import archive is mapped to the target elements. How much of the mass of the import archive is mapped is shown in the out file.
8. In the OUT file, a summary is shown of all variables that are mapped.
9. In the follow-up run, the cycle and time are taken from the import archive. The results of the first cycle of the follow-up run are determined from remapping only and has not gone through an equation of state yet. This will happen in the next cycle
10. If needed the remapping can be checked by doing only one additional cycle in the follow-up run with a quite small time step. Then, the follow-up OUT file shows two cycles and the results should be almost identical to the results of the import archive. Also, material summaries in the OUT file between first run and follow-up run should be identical. The only exceptions are the summaries of momentum, kinetic energy, and total energy per material. For these three quantities, only the total amounts will remain constant between first and follow-up run.

11. To remap a spherical symmetric or an axial symmetric Euler archive, the DYPARAMs [DYPARAM,SPREMAP](#) and [DYPARAM,AXREMAP](#) have to be used. Also, the remapping of a spherical symmetric Euler domain onto a 2-D axial symmetric Euler domain is supported.

**A2GG****Selects a Direct Input Fluid-Structure Coupling Matrix**

Selects a direct input fluid-structure coupling matrix.

**Format:**

A2GG = name

**Example:**

A2GG = AGG0

Descriptor	Meaning
name	Name of a fluid-structure coupling matrix that is input on the DMIG Bulk Data entry.

**Remarks:**

1. DMIG entries will not be used unless selected by the A2GG Case Control command.
2. This entry must be above subcase level or in the first subcase.
3. If the A2GG Case Control command selects a DMIG entry, then Nastran will add the selected fluid-structure coupling matrix to the computed coupling matrix. To replace the computed coupling matrix with the selected A2GG matrix, set PARAM,ASCOUP,NO. The user may still define panels with the panel selection procedure.
4. When filling out the DMIG entries: IFO = 1, NCOL = g-size, GJ-column index corresponds to fluid points, CJ = 0, Gi-row index corresponds to structural points, Ci-corresponds to DOF, Ai-the area values.
5. A2GG is supported in dynamic solutions with fluid-structure coupling.
6. Only one A2GG command should be used. It must appear above any subcase structure or in the first subcase only.

## ACCELERATION

### Acceleration Output Request

Requests form and type of acceleration vector output.

#### Format:

$$\text{ACCELERATION} \left[ \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{l} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \right), \right. \\
 \left. \left[ \begin{array}{l} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \text{RPUNCH}, \text{CID} \right) \\
 = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

#### Examples:

```

ACCELERATION=5
ACCELERATION (SORT2, PHASE)=ALL
ACCELERATION (SORT1, PRINT, PUNCH, PHASE)=17
ACCELERATION (SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
ACCELERATION (PRINT, RALL, NORPRINT)=ALL
    
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT or (blank)	Printer File (.f06)
PUNCH	Punch File (.pch)
PLOT	Plot File (.op2/.h5)

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.



Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.
ATOC	Requests the autocorrelation function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.
RALL	Requests all of PSDF, ATOC and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 5.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Requests printing of output coordinate system ID in printed output file (.f06).
ALL	Accelerations at all points will be output.
n	Set identification of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No results will be output (Default).

### Remarks:

1. Acceleration output is only available for transient and frequency response problems. Acceleration is only available for transient and frequency response problems and when response spectra is requested in eigenvalue analysis.
2. See Remark 1 under [DISPLACEMENT \(Case\), 296](#) for a discussion of SORT1 and SORT2.
3. ACCELERATION=NONE allows overriding an overall output request.
4. Acceleration results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
5. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
6. Note that the CID keyword affects only grid point related output, such as displacement (DISP), velocity (VELO), acceleration (ACCEL), OLOAD, SPCforce (SPCF), and MPCforce (MPCF). In addition, the CID keyword needs to appear only once in a grid-related output request anywhere in the Case Control Section to turn on the printing algorithm.

## ACFPMRESULT

## Acoustic Field Point Mesh Results Output Request

Requests output of field point mesh results. This Case Control command can be used in SOL 108 and SOL 111 only.

### Format:

$$\text{ACFPMRESULT} \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT,PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \text{VELOCITY} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right] \right. \\ \left. \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \text{POWER} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right] \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Descriptor	Meaning
------------	---------

SORT1	Output will be presented as tabular listing of grid points for each excitation frequency (Default).
SORT2	Output will be presented as a tabular listing of excitation frequencies for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

VELOCITY	Requests output of particle velocities (Default = NO).
REAL or IMAG	Requests rectangular format (real or imaginary) of complex output. Use of either REAL or IMAG yields the same output.
ALL	Radiated power will be processed for the wetted surface and all panels.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
POWER	Requests output of power through field point mesh (Default = YES).

Describer	Meaning
ALL	Results of all field point meshes, AFPMIDs, will be processed.
n	Set identification of a previously defined set of field point mesh identifiers, AFPMIDs, Results will be processed for the field point meshes in this set only.
NONE	Field point mesh results will not be processed. ACFPMRESULT = NONE overrides an overall request.

**Remark:**

1. If the acoustic model references multiple PACINF entries that do not have coincident pole locations and if the acoustic field point meshes contain element data, then a PACINFID parameter entry is required in the bulk data for each mesh to identify the PACINF bulk data entry that is used to define the location of the pole. Data recovery operations require that the element surface normal vector point away from the pole location.
2. The member of set for ACFPMRESULT must be the IDs of 'BEGIN AFPM='; not GRID IDs or element IDs under a 'BEGIN AFPM='.

## ACPOWER

## Acoustic Power Output Request

Requests output of the power radiated from the wetted surface.  
This Case Control command can be used in SOL 108 and SOL 111 only.

### Format:

$$\text{ACPOWER} \left[ \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], [\text{CSV} = \text{unit}] \right) \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Describer	Meaning
SORT1	Output will be presented as a tabular listing of panels for each excitation frequency.
SORT2	Output will be presented as a tabular listing of excitation frequencies for each panel (Default).

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

CSV	Results will be written to a .CSV file. See Remark 1.
unit	Unit of the .csv file as used on the ASSIGN statement.
ALL	Radiated power will be processed for the wetted surface and all panels.
n	Set identification of a previously defined set of panels. Radiated power will be processed for the wetted surface and all panels in the referenced set.
NONE	Radiated power will not be processed. ACPOWER = NONE overrides an overall request.

### Remarks:

1. If output to an .CSV file is requested, the file must be assigned with logical key “USERFILE” and FORM=FORMATTED, e.g.,

```
ASSIGN USERFILE = 'myfile.csv' UNIT=50 FORM=FORMATTED STATUS=NEW
```

2. The mean radiated acoustic power per period ( $P_A$ ) is computed as:

$$P_A = \frac{1}{2} \text{REAL}(\{v\}^* [A] \{p\})$$

where,  $\{v\}^*$  is the complex conjugate transpose of the complex velocity of the wetted surface,  $[A]$  is the coupling matrix, and  $\{p\}$  is the acoustic pressure.

## ACTIVAT

### Elements to be Reactivated in SOL 600

Indicates which Bulk Data ACTIVAT entry is used to control the elements to be activated in this subcase. This entry may only be used in SOL 600.

#### Format:

ACTIVAT=N

#### Example:

ACTIVAT=3

Describer	Meaning
N	ID of a matching Bulk Data ACTIVAT entry specifying the elements to be reactivated for this subcase.

#### Remarks:

1. Different sets of elements can be reactivated during different subcases using this Case Control command.
2. The elements specified in the matching ACTIVAT Bulk Data entry must currently be in a deactivated state.

## ACTRIM

## ACTRAN Trimmed Material Matrices for SOL 108/111

Select ACTRAN trimmed material matrices.

### Format:

ACTRIM = name1, name2, ... namen

### Example:

```
ACTRIM = FLOOR_F, FLOOR_R, DASH
SET 10 = FR_LH, RR_LH
ACTRIM = 10
```

Describer	Meaning
namei	Name of the ACTRAN trimmed material matrices that is input on the ACTRIM bulk data entry, or name list.

### Remarks:

1. This entry must be above subcase level or in the first subcase.
2. If the ACTRIM Case Control command selects ACTRIM bulk data entries, Nastran will add the selected ACTRAN matrices to fluid-structure coupling problem in all subcases.
3. ACTRIM is supported in frequency response analysis for fluid-structure coupling problem and the frequency dependent algorithm will be adopted automatically.
4. PARAM, ACSYM, YES should be set for ACTRIM (default).
5. The effect of ACTRIM will be considered in standard frequency response analysis and participation factor analysis by PFMODE, PFPANEL and PFGRID.

## ADAMSMNF\*

## Control for Nastran/ADAMS Interface

Control for Nastran/ADAMS Interface modal neutral file (.mnf)

### Format:

$$\begin{aligned}
 \text{ADAMSMNF} & \left[ \text{FLEXBODY} = \left\{ \begin{array}{c} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{FLEXONLY} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right], \\
 & \left[ \text{ADMCHECK} = \left\{ \begin{array}{c} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{ADMOUT} = \left\{ \begin{array}{c} \text{NO} \\ \text{YES} \end{array} \right\} \right], \\
 & \left[ \text{OUTGSTRS} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right], \left[ \text{OUTGSTRN} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right], \\
 & \left[ \text{OUTSTRS} = \left\{ \begin{array}{c} \text{NO} \\ \text{YES} \end{array} \right\} \right], \left[ \text{OUTSTRN} = \left\{ \begin{array}{c} \text{NO} \\ \text{YES} \end{array} \right\} \right] \\
 & \left[ \text{V1ORTHO} = \left\{ \begin{array}{c} -1.0 \\ \text{value1} \end{array} \right\} \right], \left[ \text{V2ORTHO} = \left\{ \begin{array}{c} 1.0e8 \\ \text{value2} \end{array} \right\} \right], \\
 & \left[ \text{MINVAR} = \left\{ \begin{array}{c} \text{PARTIAL} \\ \text{CONSTANT} \\ \text{FULL} \\ \text{NONE} \\ \text{RIGID} \end{array} \right\} \right] \\
 & \left[ \text{PSETID} = \left\{ \begin{array}{c} \text{NONE} \\ \text{setid}_{\text{plotel}} \\ \text{ALL} \end{array} \right\} \right] \left[ \text{EXPORT} = \left\{ \begin{array}{c} \text{MNF} \\ \text{DB} \\ \text{BOTH} \end{array} \right\} \right] \left[ \text{MONITOR} = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\} \right]
 \end{aligned}$$

### Example(s):

ADAMSMNF FLEXBODY = YES



**Note:** \*Nastran/ADAMS modal stress recovery (MSR) interface is also available. See Remark 19. CAUTION: Do not use mode=i8 (64 bit integer, 64 bit float) option when submitting the Nastran job if the ADAMSMNF EXPORT option specifies DB or BOTH and your intent is to use the DB in ADAMS. ADAMS does not currently support i8 DB files. If you use mode=i8, you must use the MNF in ADAMS.

NASTRAN/ADAMS MNF can be used with SOL 400 as described in Remark 21.

ADAMSMNF cannot be run in linear solution sequences with RIGID=LAGRANGE. A fatal will be issued. The main reason is that the Lagrange multipliers are in the wrong set for the boundary solution and resulting orthonormal modes will in general not be correct.

Describer	Meaning
FLEXBODY	Requests that the Nastran/ADAMS interface be executed.
NO	Executes standard Nastran.
YES	Executes Nastran/ADAMS interface.
FLEXONLY	Requests standard DMAP solution and data recovery following Nastran/Adams interface execution.
YES	Executes only the Nastran/ADAMS interface.
NO	Executes Nastran/ADAMS interface and standard DMAP solution and data recovery.
ADMCHECK	Requests Nastran/ADAMS diagnostic output.
YES	Prints diagnostic output.
NO	Suppresses diagnostic output.
ADMOUT	Requests that the Nastran/ADAMS interface outputs Nastran .op2 files.
YES	.op2 files are generated.
NO	Requests that .op2 files are not generated.
OUTGSTRS	Controls grid point stress output to .op2 file or .mnf or both.
YES	Grid point stress is output to .op2 file or .mnf or both.
NO	Grid point stress is not output to .op2 file, or .mnf file.
OUTGSTRN	Controls grid point strain output to .op2 file, or .mnf or both.
YES	Grid point strain is output to .op2 file or .mnf or both.
NO	Grid point strain is not output to .op2 file or .mnf.
OUTSTRS	Controls element stress output to .op2 file.
YES	Element stress is output to .op2 file.
NO	Element stress is not output to .op2 file.

Describer	Meaning
OUTSTRN	Controls element strain output to .op2 file.
YES	Element strain is output to .op2 file.
NO	Element strain is not output to .op2 file.
V1ORTHO	Lower frequency bound of the Craig-Bampton modes in cycles/unit time.
value1	Value of lower bound.
V2ORTHO	Higher frequency bound of the Craig-Bampton modes in cycles/unit time.
value2	Value of higher bound.
MINVAR	Requests the type of mass invariants to be computed. See Remark 3.
FULL	All nine mass invariants will be calculated.
CONSTANT	Only mass invariants (5-1), (5-2), (5-6), and (5-7) will be calculated.
PARTIAL	All mass invariants except (5-5) and (5-9) will be calculated.
NONE	No mass invariants are computed. (This option is invalid for structure/fluid coupled system. )
RIGID	No modal information is output to the .mnf file. Only units, grid point coordinates, element connectivity, interface nodes, and invariant (5-1), (5-2), and (5-7) data are shared in the .mnf file.
EXPORT	Controls modal output.
MNF	output to MNF file
DB	output to Nastran database
BOTH	output to Nastran database and MNF file
PSETID	Selects a set of elements (including PLOTEL) whose grids are retained in the MNF, and whose connectivity defines face geometry for ADAMS display.
setid <sub>plotel</sub>	Specified in the OUTPUT(PLOT) Section of Nastran.
ALL	Select all the sets defined in the OUTPUT(PLOT) Section of Nastran.
MONITOR	Request or suppress output of monitor point data to Adams MNF. Default is output (Yes). Specify No to suppress output.

### Remarks:

1. This entry represents a collection of PARAM,name,value entries. A license is required for the Nastran/ADAMS interface.
2. ADAMSMNF FLEXBODY = YES is required to execute the Nastran/ADAMS interface, all other ADAMSMNF items are optional. The ADAMSMNF FLEXBODY = YES must occur above subcase level.

If you want standard Nastran data recovery in addition to that produced for ADAMS, you need to run with FLEXONLY=NO. The orthonormal modes produced for ADAMS and the modes produced by the standard MSC Nastran run may be viewed in Patran if run with PARAM,POST,0.

3. The nine mass invariants are defined by Eqs. (5-1) through (5-9):

$${}^1_I = \sum_{p=1}^N m_p \quad (5-1)$$

$${}^2_I = \sum_{p=1}^N m_p \mathbf{s}_p \quad (5-2)$$

$${}^3_{I_j} = \sum_{p=1}^N m_p \Phi_p \quad j = 1, \dots, M \quad (5-3)$$

$${}^4_I = \sum_{p=1}^N m_p \tilde{\mathbf{s}}_p \Phi_p + \mathbf{I}_p \Phi_p^* \quad (5-4)$$

$${}^5_{I_j} = \sum_{p=1}^N m_p \tilde{\phi}_{pj} \Phi_p \quad j = 1, \dots, M \quad (5-5)$$

$${}^6_I = \sum_{p=1}^N m_p \Phi_p^T \Phi_p + \Phi_p^{*T} \mathbf{I}_p \Phi_p^* \quad (5-6)$$

$${}^7_I = \sum_{p=1}^N m_p \tilde{\mathbf{s}}_p^T \tilde{\mathbf{s}}_p + \mathbf{I}_p \quad (5-7)$$

$${}^8_{I_j} = \sum_{p=1}^N m_p \tilde{\mathbf{s}}_p \tilde{\phi}_{pj} \quad j = 1, \dots, M \quad (5-8)$$

$${}^9_{I_{jk}} = \sum_{p=1}^N m_p \tilde{\phi}_{pj} \tilde{\phi}_{pk} \quad j, k = 1, \dots, M \quad (5-9)$$

where  $\mathbf{s}_p = [xyz]^T$  are the coordinates of grid point  $p$  in the basic coordinate system;

$$\tilde{\mathbf{s}}_p = \begin{bmatrix} 0 & -z & y \\ z & 0 & -x \\ -y & x & 0 \end{bmatrix}$$

is the skew-symmetric vector cross product operator;  $\Phi_p$  is the partitioned orthogonal modal matrix that corresponds to the translational degrees-of-freedom of grid  $p$ ;  $\mathbf{I}_p$  is the inertia tensor;  $\Phi_p^*$  is the partitioned orthogonal modal matrix that corresponds to the rotational degrees-of-freedom of grid  $p$ ;  $\tilde{\Phi}_{pf}$  is the skew-symmetric matrix formed for each grid translational degree-of-freedom for each mode;  $M$  is the number of modes; and  $N$  is the number of grid points.

4. The preceding mass invariant calculation currently depends on a lumped mass formulation. The parameter PARAM,COUPMASS should not be specified when executing the Nastran/ADAMS interface.
5. If the CONM1 is used, M21, M31, and M32 entries should be left blank.
6. If PARAM,GRDPNT,value specified, mass invariants  $^1I$ ,  $^2I$ , and  $^7I$  will be obtained from an Nastran grid point weight generator execution in the basic system.
7. The following DTI,UNITS Bulk Data entry is required for a FLEXBODY=YES run:

Since ADAMS is not a unitless code (as is Nastran), units must be specified. A DTI Bulk Data entry provides 'UNITS' (a unique identifier) input as the following example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed in the following table.

Format:

DTI UNITS 1 MASS FORCE LENGTH TIME

Example:

DTI UNITS 1 KG N M S

Mass:

kg - kilogram  
 lbm - pound-mass  
 slug - slug  
 gram - gram  
 ozm - ounce-mass  
 klbm - kilo pound-mass (1000.lbm)  
 mgg - megagram  
 slinch - 12 slugs  
 ug - microgram  
 ng - nanogram  
 uston - US ton

Length:

km - kilometer  
 m - meter  
 cm - centimeter  
 mm - millimeter  
 mi - mile  
 ft - foot  
 in - inch  
 um- micrometer  
 nm - nanometer  
 ang - angstrom  
 yd - yard  
 mil - milli-inch  
 uin - micro-inch

Force:

n - newton  
 lbf - pounds-force  
 kgf - kilograms-force  
 ozf - ounce-force  
 dyne - dyne  
 kn - kilonewton  
 klbf - kilo pound-force (1000.lbf)  
 mn - millinewton  
 un - micronewton  
 nn - nanonewton

Time:

h - hour  
 min - minute  
 s - sec  
 ms - millisecond  
 us - microsecond  
 nanosec - nanosecond  
 d - day

A note of clarification about UNITS and its relation to Nastran's WTMASS parameter: WTMASS, though necessary to achieve units consistency in Nastran, is ignored in the output for ADAMS. Units data for ADAMS is supplied on the UNITS DTI entry. For example, consider a model with mass in grams, force in Newtons, length in meters, and time in seconds. A WTMASS parameter equal to 0.001 would ensure that Nastran works with a consistent set of units (kg, N, and m). The units reported to ADAMS should then be: "DTI, UNITS, 1, GRAM, N, M, S."

8. OUTSTRS or OUTSTRN entries require the use of the standard Nastran STRESS= or STRAIN= Case Control commands to produce element stress or strain. STRESS(PLOT)= or STRAIN(PLOT)= will suppress stress or strain output to the Nastran .f06 file. The OUTSTRS or OUTSTRN entries are required for importing ADAMS results into MSC Fatigue. See the Nastran/ADAMS/durability documentation for more information.

9. OUTGSTRS or OUTGSTRN entries require the use of the standard Nastran STRESS= or STRAIN= Case Control commands used in conjunction with GPSTRESS= or GPSTRAIN= Case Control commands to produce grid point stress or strain. GPSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain output to the Nastran .f06 file.
10. To reduce the finite element mesh detail for dynamic simulations, PSETID=set\_entry is used to define a set of PLOTELS or other elements used to display the component in ADAMS. If a mass invariant computation is requested, this option can significantly reduce the size of the .mnf without compromising accuracy in the ADAMS simulation.

If PSETID specifies an existing set from the OUTPUT(PLOT) Section of Nastran, this single set is used explicitly to define elements to display in ADAMS. Otherwise, the Nastran Case Control Section will be searched for a matching set ID. This matching set ID list then represents a list of OUTPUT(PLOT) defined elements sets, the union of which will be used to define a set of PLOTELS or other elements used to display the component in ADAMS. If the user wishes to select all of the sets in the OUTPUT(PLOT) Section, then use PSETID=ALL.

The elements defined may include rigid element IDs. When defining these sets, do not use EXCLUDE and EXCEPT descriptions.

If a superelement analysis is being executed, any element defined on the PSETID=set\_entry that lies entirely on the superelement boundary (i.e., all of its grids are a-set or exterior to the superelement) must also be specified on a SEELT Bulk Data entry. The SEELT entry would not be required for part superelements, as boundary elements stay with their component.

```
OUTPUT (PLOT)
SET 7722 = 10001 THRU 10010
```

11. The ADMOUT=YES option is intended for users who plan to import ADAMS results into MSC Fatigue. This option requires the following assignment command in the File Management Section of the Nastran file:

```
ASSIGN OUTPUT2='name.out' STATUS=UNKNOWN UNIT=20 FORM=UNFORM
```

It causes .op2 files with an .out extension to be generated for input into MSC Fatigue. FLEXBODY=YES is required with its use. The files' outputs are: DTI-units, SE-number of superelements (9999 if residual), SEIDX-superelement id, ASETX-size of a-set, BGPPTS-grid location table, GEOM2S-element connections, GEOM4S-constraints and sets, MGGEW-physical mass external sort with weight mass removed, VAEXT-a-set partition vector, VGEXT-g-set partition vector, VAPEXT-eigenvalue size partition vector, MAAEW-modal mass, KAAE-modal stiffness, BAAE-modal damping, RAE-modal preload, PAE-modal loads, CMODEXT-component modes, OES1-element stress shapes, OSTR1-element strain shapes, OGS1-grid point stress shapes, OGSTR1-grid point strain shapes, OGS1PL-grid point physical preload stress, OGTR1PL-grid point physical preload strain. The files are output for each superelement and their generation depends on the loading and output requests.

To ensure compatibility with the ADAMS .op2-to-.mnf translator, if the ADAMSMNF Case Control command has the keyword ADMOUT=YES, the Nastran SYSTEM word OP2NEW is automatically set to OP2NEW=0. This means that any .op2 files generated will have a pre-MSC Nastran 2004 format.

12. Environment variables controlling .mnf generation can be set before submitting the Nastran job, or by using the Nastran keyword 'MNFWRITEOPTIONS' which can be abbreviated to any short unique string such as 'MNFW'. The Nastran keyword can be entered on the NASTRAN submittal command line or in a user .nastran rc file. For example, interior grids and elements can be removed in the .mnf by entering, prior to the Nastran submittal, with Korn shell:

```
export MDI_MNFWRITE_OPTIONS=strip_face
```

with C shell:

```
setenv MDI_MNFWRITE_OPTIONS=strip_face
```

Or, at the time of Nastran submittal:

```
nastran_submittal_command jid MNFW=strip_face
```

Consult the ADAMS/Flex documentation for more information on the use of environment variables during .mnf generation.

The command

```
nastran_submittal_command help mnfw
```

will generate the description of the symbol keyword.

The command

```
nastran_submittal_command help all
```

will generate a complete set of Nastran submittal keywords.

13. .mnf naming convention is as follows: for a single superelement run, 'jid.mnf'; for a residual only or multiple superelement run 'jid\_seid.mnf', etc.; where seid1 and seid2 are the integer numbers of the superelement. The default location of these files is the same directory as the jid.f06 file. See the ASSIGN .mnf command to change directory location.
14. When supplying SPOINT/QSET combinations, there should be a sufficient number of combinations to correctly capture the modal shapes. If  $n$  is the number of modes specified on the EIGR or EIGRL Bulk Data entries and  $p$  is the number of loadcases specified, then the number of SPOINTs( $ns$ ) should be at least  $ns = n + (6 + p)$  assuming that residual flexibility is on. In general, there cannot be too many SPOINTS, as excess ones will simply be truncated with no performance penalty.
15. The user can have Nastran automatically specify the SPOINT/QSET by including, above the Case Control Section the parameter PARAM,AUTOQSET,YES. In this case no SPOINT/QSET can appear in the Bulk Data. See the PARAM,AUTOQSET description for detailed requirements or limitations.
16. By default, MSC Nastran 2005 will create a version 6.0 MNF. ADAMS 2005 is able to read the version 6.0 .mnf file. Earlier ADAMS versions are not able to read a 6.0 .mnf file. Nastran can be instructed to write a backward-compatible .mnf file by submitting the Nastran job with MNFWRITEOPTIONS=full\_str. Alternatively, the user may set the environment variable MDI\_MNFWRITE\_OPTIONS to 'full\_str'. See Remark 12. for more information on controlling the .mnf format.

17. In addition to modal stiffness and modal mass matrices, the modal damping matrix may also be output to the .mnf. The damping allowed is the standard Nastran damping matrix consisting of  $[B_{gg}]$  viscous damping,  $(1/w4) [K4_{gg}]$  structural damping,  $(g/(w3) + \alpha2)[K_{gg}]$  structural and Rayleigh damping, and  $(\alpha1)[M_{gg}]$  Rayleigh damping. Where g is set by PARAM,G,value, w3 is set by PARAM,W3,value; w4 is set by PARAM,W4,value;  $\alpha1$  is set by PARAM,ALPHA1,value,0.;and  $\alpha2$  is set by PARAM,ALPHA2,value,0.

Additionally,  $[B2H]$  modal damping can be included by use of the Case Control command SDAMP=n. For part superelement or superelement analyses, modal damping for each individual part or superelement can be controlled by PARAM,SESDAMP,YES (PARAM,SESDAMP,NO is the default).

SESDAMP		
sesdamp = no	Modal damping for each superelement using the free boundary modes.	SDAMP $\Rightarrow$ TABDMP1 SDAMP above subcase.
sesdamp = yes	Modal damping for each superelement using the fixed boundary CMS modes.	SDAMP $\Rightarrow$ TABDMP1 SDAMP in superelement subcase.
For part superelements each part may have a PARAM,SESDAMP		

The Nastran/ADAMS interface does not allow for adding modal damping to structural damping using PARAM,KDAMP,-1.

Direct input damping may also be included with the Case Control command B2GG=n. For part superelement or superelement analyses, use of this command with the Nastran/ADAMS interface requires fully expanded case control.

18. If preload is present in the model, physical gridpoint stress and strain for the preload may be output to the .mnf using standard GPSTRESS= or GPSTRAIN= commands.  
If preload is generated in a SOL 106 run for a SOL 103 restart, and the physical grid point stresses for the preload are desired for the SOL 103 MNF run, then PARAM,FLEXNLS,YES is required above the subcase level in the SOL 106 run.  
For preload generated in a SOL 106 run for a SOL 103 restart, the preload subcase must be replicated in the first subcase of the SOL 103 run.
19. After using the Nastran/ADAMS interface to produce an .mnf file and after performing an ADAMS solution, it is possible to bring the ADAMS results into Nastran for modal data recovery. ADAMS produces .op2 files for input to Nastran SOL 111 and SOL 112. The files are binary format with an .mdf extension. The File Management Section requires an assign command for each file:  
ASSIGN INPUTT2=' name .mdf' UNIT=ni  
with a DLOAD = ni in the appropriate subcase.



Also, in the Bulk Data Section, the parameter PARAM,ADMPOST = m (m = 0, by default no MSR performed) is required. If m = 1, rigid body motion is not considered in the structural deformation. If m = 2, rigid body motion is considered in the structural deformation.

Full details of the generation of the .mdf files and their use with Nastran are to be found in the Nastran/ADAMS durability documentation.

20. In order to obtain consistent ADAMS results in Nastran data recovery run, when ADAMS data brought back into Nastran SOL 111 or SOL 112, Nastran must be restarted from either of the following two options:
  - a. From the original Nastran database that was produced during SOL103 MNF creation run (created MNF is used in ADAMS run).
  - b. From the external SE (\*.op2) that was produced during SOL103 MNF creation run (created MNF is used in ADAMS run). The File Management section requires to assign command for external SE op2.

```
ASSIGN INPUTT2='EXTSE100.OP2' UNIT=ni
```

with following PARAM entries

```
PARAM ADMPOST,1 or 2 $
```

```
PARAM ADMEXTU , ni $ Where ni = unit # of external SE op2.
```

21. Typically SOL 400 is used to produce a preload for an ADAMS flexbody MNF run.
  - a. In the preload run the structure should be statically supported and follower loading must be applied as a self-equilibrating load set (not with SPC relationships!). In the ANALYSIS=MODES step the structure must be a free-free structure as the resulting orthonormalization requires that six rigid body modes be present.

In order to produce modal amplitudes and mode shapes and to ensure residual vector calculations, SPOINTs and Q-sets are required. The SPOINTs must be included in the MAIN Bulk Data as they are included in the overall matrix size.

A new BULK data section labeled BEGIN FLXBDY =id must be included with the run. This new bulk section must contain the Q-set associate with the SPOINTs (in main bulk!) for modal amplitudes and the A-set required for attachment point designation.

The example below is a typical SOL400 problem setup:

```
SOL 400
CEND
$ Case Control Section
$ Output ADAMSMNF REQUIRED ABOVE SUBCASE
ADAMSMNF flexbody=yes, psetid=all, outgstrs=yes, outgstrn=yes
SUBCASE 1
  $ Preload
  STEP 10
      $ Static load and support for preload
      SUBTITLE = PRELOAD
      ANALYSIS = NLSTATICS
      NLSTEP = 110
      LOAD = 120
      SPC = 130
```

```

                BCONTACT = 140
                SPCF = ALL
                $ Generate stress and strain grid shapes
                STRESS(PLOT) = ALL
                STRAIN(PLOT) = ALL
                GPSTRESS(PLOT) = ALL
                GPSTRAIN(PLOT) = ALL
                $ Modal Step for Producing MNF
                $ Default: Select the end of previous load step to output
$ ADAMSMNF
  STEP 20
    ANALYSIS = MODES
    $ Select real Eigen Value Parameters
    METHOD = 210
    $ Turn residual vectors on
    RESVEC = COMPONENT
    STRESS(PLOT) = ALL
    STRAIN(PLOT) = ALL
    GPSTRESS(PLOT) = ALL
    GPSTRAIN(PLOT) = ALL

```

- b. In the above example, the SPC set in the ANALYSIS=NLSTAT must be a static (non-redundant) constraint condition. Note that in the ANALYSIS=MODES STEP, the SPC constraint set has been removed.

In SOL 400, the definition of the attachment a-set for identifying attachment points and for q-set for specifying the desired number of modal amplitudes for orthonormalization is done in a separate new FLXBDY Bulk Data Section shown below:

```

$ FLEXBODY Bulk section
BEGIN BULK FLXBDY = 10
$ Attachment point and component mode (A-SET) selection
ASET1,123456,1,11,111,121
QSET1,0,100001,THRU,100020

```

In SOL400, the ASET/ASET1 and QSET/QSET1, MUST, appear in the FLXBDY bulk data section.

Currently, only one FLXBDY bulk data section (with a positive Flexbody ID) is supported in SOL 400. Any non set related Bulk entries appearing in this Section will be ignored.

- c. If CONTACT is required as part of the preloading for the FLEXBODY=YES run, it is Highly Recommended that the friction option be turned on by using an appropriate BCPARA bulk data entry setting, e.g.,

```

$ Select bilinear Coulomb friction for all subcases
BCPARA, 0, FTYPE, 6

```

If contact friction is not turned on, the tangential motion between the two parts coming into contact will most likely not be constrained and incorrect or fatal results will occur.

- d. If RIGID elements (RBE1/RBE2/RBE3/RBAR/RROD/RJOINT) are in the model, then the Case Control RIGID= LAGRANGE (default for SOL400) should be used to avoid possible wrong results. If an attachment point happens to touch a rigid element, the point should be associated with the independent degree of freedom of the rigid element. Though not recommended, if for some modeling requirement, a dependent rigid element grid is required to be in the attachment set, the user MUST include at least one independent/reference grid for that specific rigid element in the ASET.
22. For STRUCTURE and FLUID coupled system, METHOD(COUPLED) can be used for MNF generation. However, all nodes on fluid face need to be coupled with structure for ADAMSMNF otherwise, mass and inertia of moment will be incorrect in MNF.

## AECONFIG

## Aeroelastic Configuration Name

Assigns the aerodynamic configuration parameter used to locate the associated datablocks on the aerodynamic and aeroelastic databases.

### Format: Assign a DBset member name

```
AECONFIG    =config-name
```

### Example:

Assign a MASTER file for the aerodynamic and aeroelastic DBsets

```
AECONFIG    =PROTO_A
```

Describer	Meaning
config-name	The configuration name. This is the aerodynamic supergroup name identified as part of the aeroelastic model (Character; Default = AEROSG2D).

### Remarks:

1. Typically, the aeroelastic configuration name is specified as the aerodynamic supergroup as part of the model generation in MSC FlightLoads.
2. If AECONFIG is not present, aerodynamic and aeroelastic datablocks will be created from the data in the Bulk Data Section and assigned the default value AECONFIG=AEROSG2D.
3. Multiple configuration names are supported.
4. AECONFIG is typically assigned above the subcase level. If it is overridden at the subcase level, it is necessary to attach an existing aerodynamic database.

**AERCONFIG**

## Aerodynamic Model to be Used for the Rigid Aerodynamics

Enables the user to select a different mesh for the rigid portion of the aerodynamics than for the elastic portion.

**Format:**

AERCONFIG=config-name

**Examples:**

AERC=RAERO

Describer	Meaning
AERC	The configuration name. This is the aerodynamic supergroup name for the aerodynamic model that is used to create the rigid aerodynamics.

**Remarks:**

1. If the AERCONFIG Case Control command is not present in the subcase, the rigid portion of the aerodynamics is based on the same AECONFIG as the flexible aerodynamics.
2. The rigid aerodynamics must be precomputed and attached from an assigned database using FMS commands such as:

```
ASSIGN RMASTER = "raero.master"
DBLOCATE WHERE (AECONFIG=' rconfig' ) LOGICAL=RMASTER
```

**AEROF**

## Aerodynamic Force Output Request

Requests the aerodynamic loads on aerodynamic control points.

**Format:**

AEROF=*n*

**Examples:**

AEROF=ALL

AEROF=5

Describer	Meaning
<i>n</i>	Set identification of a previously appearing SET command (Integer > 0).
ALL	Forces at all points will be output.

**Remarks:**

1. This command is supported in SOLs 144, 146 (frequency response only) and 200 for ANALYSIS=SAERO.
2. The SET command references box or body element identification numbers.
3. Output is in the units of force or moment.
4. Only aerodynamic forces on points specified on the SET command will be output.

**AESYMXY****Aerodynamic Flow Symmetry About The XY Plane**

Aerodynamic XY plane of symmetry flag. This is used to indicate whether the aerodynamic model has symmetry with respect to the ground.

**Format:**

$$\text{AESYMXY} = \left\{ \begin{array}{l} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{array} \right\}$$

**Example:**

`AESYMXY = ASYMMETRIC`

<b>Describer</b>	<b>Meaning</b>
SYMMETRIC	Indicates that the aerodynamic model is moving in a symmetric manner with respect to the XY plane.
ANTISYMMETRIC	Indicates that the aerodynamic model is moving in an antisymmetric manner with respect to the XY plane.
ASYMMTRIC	Indicates that the aerodynamic model has no reflection about the XY plane.

**Remarks:**

1. If AESYMXY is not present in case control, aerodynamic XY symmetry will be determined from the SYMXY field of the AEROS Bulk Data entry for static aeroelastic analysis, and from the SYMXY field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXY is present above the subcase level, it is applied to all subcases until overridden.
3. SYMMETRIC implies ground effect, and ASYMMETRIC implies free air analysis.
4. Multiple aerodynamic symmetries are supported.

## AESYMXZ

## Aerodynamic Flow Symmetry About The XZ Plane

Aerodynamic XZ plane of symmetry flag. This is used to support symmetric models about the centerline.

### Format:

$$\text{AESYMXZ} = \left\{ \begin{array}{c} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \\ \text{ASYMMETRIC} \end{array} \right\}$$

### Example:

AESYMXZ = SYMMETRIC

Describer	Meaning
SYMMETRIC	Indicates that a half span aerodynamic model is moving in a symmetric manner with respect to the XZ plane.
ANTISYMMETRIC	Indicates that a half span aerodynamic model is moving in an antisymmetric manner with respect to the XZ plane.
ASYMMETRIC	Indicates that a full aerodynamic model is provided (Default).

### Remark:

1. If AESYMXZ is not present in case control, aerodynamic XZ symmetry will be determined from the SYMXZ field of the AEROS Bulk Data entry for static aeroelastic analysis, and from the SYMXZ field of the AERO Bulk Data entry for flutter and dynamic aeroelastic analysis.
2. If AESYMXZ is present above the subcase level, it is applied to all subcases until overridden.
3. Multiple aerodynamic symmetries are supported.



**AEUXREF****Defines the Reference Aerodynamic Extra Point (Controller) Vector**

Reference UXVEC selector for the aeroelastic trim analysis. This is used to indicate an aerodynamic extra point vector about which the stability derivatives are to be computed and printed. The stability derivatives are the change in force due to a unit perturbation of each parameter in the aerodynamic extra point set. Due to the nonlinear nature of the aeroelastic loads, the stability derivatives can be (but are not required to be) a function of the point about which the slope is computed. This input defines which point is to be used in computing the stability derivatives for printing (local slopes will be computed as needed in the trim solver). This selection is typically done within each subcase, but a case control default can be defined by placing an entry above the subcase level.

**Format:**

$$\text{AEUXREF} = \left\{ \begin{array}{c} \text{n} \\ \text{TRIM} \end{array} \right\}$$

**Examples:**

```
AEUXREF=100
AEUXREF=TRIM
```

Describer	Meaning
n	The identification number of a UXVEC Bulk Data entry that defines the point about which stability derivatives will be computed in TRIM cases.
TRIM	Indicates that the stability derivatives should be computed about the trimmed state.

**Remarks:**

1. If, for a particular subcase, AEUXREF is not defined, the “free stream” state will be used (that is, the stability derivatives will be computed about zero values for all parameters). This results in upward compatibility with the linear database paradigm.
2. Only one of a TRIM or a UXVEC ID may be specified on any given subcase. To see stability derivatives about additional points, you must define additional trim subcases.

## ANALYSIS

## Analysis Discipline SUBCASE/STEP/SUBSTEP Assignment

Specifies the type of analysis being performed for the current SUBCASE/STEP/SUBSTEP.

### Format:

ANALYSIS=type

### Examples:

SOL 200

```
SUBCASE 10
  ANALYSIS=STATIC
SUBCASE 20
  ANALYSIS=MODES
```

Describer	Meaning
type	Analysis type. Allowable values and applicable solution sequences (Character):
STATICS	Linear Static Analysis (SOLs 200 & 400)
MODES	Normal Modes Analysis (SOLs 110, 111, 112, 106, 200, & 400) See Remarks <a href="#">2</a> , <a href="#">3</a> , and <a href="#">4</a> .
BUCK	Buckling (SOLs 200 and 400)
DFREQ	Direct Frequency (SOLs 106, 200 and 400). See Remark <a href="#">4</a> .
MFREQ	Modal Frequency (SOLs 200 and 400)
MTRAN	Modal Transient (SOLs 200 and 400)
DCEIG	Direct Complex - Eigenvalue Analysis (SOLs 200 & 400)
MCEIG	Modal Complex - Eigenvalue Analysis (SOLs 200 & 400)
SAERO	Static Aeroelasticity (SOLs 200 and 400)
DIVERGE	Static Aeroelastic - Divergence (SOLs 200 & 400).
FLUTTER	Flutter (SOLs 200 and 400)
HEAT	Heat Transfer Analysis (SOLs 153, 159 and 600 only)
STRUCTURE	Structural Analysis (SOLs 153 & 159 only, default for these two SOLs is HEAT)
HSTAT	Steady State Heat Transfer (SOL 400)
HTRAN	Transient Heat Transfer (SOL 400)
NLSTATICS	Nonlinear Static Analysis (SOL 400). See Remark <a href="#">3</a>
NLTRAN	Nonlinear Transient Analysis (SOL 400). See Remark <a href="#">3</a> .
HOT2COLD	Hot-to-cold Analysis (SOL 106 only). See Remark <a href="#">4</a> .

Describer	Meaning
RCNS	RC Network Thermal Solver Steady State
RCNT	RC Network Thermal Solver Transient

**Remarks:**

- This entry is used in solution sequences that are capable of multiple analysis types and selects the actual analysis to be performed in the SUBCASE (or for SOL 400 the SUBCASE, STEP, or SUBSTEP).
  - For linear solution sequences SOLs 110, 111, and 112 and the nonlinear SOLs 106, 153, and 159, and for Optimization SOL 200 there can be only one **ANALYSIS=** per SUBCASE
  - SOL 600,153 and SOL 600,159 require **ANALYSIS=HEAT**. For other SOL 600,ID executive control entries, **ANALYSIS=** is not used and should not be entered.
  - For SOL 200, EVERY SUBCASE including any superelement SUBCASE must be assigned an **ANALYSIS=** Case Control command either in the explicit SUBCASE or above all SUBCASEs in which case it defaults to all the subcases.
  - For SOL 400 there can, for single physics analysis or chained multi physics analysis, be only one **ANALYSIS=** per STEP.
  - For SOL 400 coupled multi physics analysis there can be only one **ANALYSIS=** per SUBSTEP.
- In the linear solution sequences SOLs 110, 111, 112, 145, and 146 this entry allows data recovery of the normal modes data used in the in the complex Eigenvalue analysis, modal frequency analysis, or modal transient analysis. All commands which control the boundary conditions (SPC, MPC, and SUPORT) and METHOD selection should be copied inside the **ANALYSIS=MODES** SUBCASE or specified above the SUBCASE level. Note in the example below, SOL111 is a frequency response solution sequence so in the frequency response SUBCASE 2 below an **ANALYSIS=MFREQ** is allowed but not required.

```

SOL 111
  METH=40
  SPC=1
  SUBCASE 1 $ Normal Modes
    ANALYSIS=MODES
    DISP=ALL
  SUBCASE 2 $ Frequency response
    STRESS=ALL
    DLOAD=12
    FREQ=4

```

- SOL 400 analysis allows for seven analysis type combinations (nonlinear single physics, nonlinear chained physics, nonlinear coupled physics, linear perturbation analysis, standard linear physics, nonlinear chained analysis with mesh/time change physics, and nonlinear response optimization with ESLNRO (Equivalent Static Loads Nonlinear Response Optimization)).

If there are linear, nonlinear and perturbation subcases, the linear subcases will be solved first. The linear subcases are reordered for processing and the output will be in the following order regardless of original subcase number: STATICS, MODES, BUCKLING, DFREQ or MFREQ, DCEIG or MCEIG, SAERO, FLUTTER.

The general rule is: The solutions of all SUBCASEs are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP in the same SUBCASE. The solutions of the SUBSTEPS occur simultaneously within a STEP (coupled analysis):

Additionally the following rules should be observed:

- All Coupled Multi-Physics steps have to come before the Single-Physics steps.
  - Single-Physics steps can follow the Multi-Physics Steps.
  - All linear perturbation steps need to be at the end after definition of all possible coupled multi-physics steps and single-physics steps.
  - The Case Control Command NLIC used for Nonlinear Initial Condition should be referenced for further requirements between the analysis types allowed in SOL 400.
- a. Nonlinear single physics: Nonlinear structures or nonlinear heat: For nonlinear structures a **ANALYSIS=NLSTAT** must come before a **ANALYSIS=NLTRAN**. Only Statics to Transient is allowed for structures. For heat transfer a nonlinear steady state heat transfer to a nonlinear transient heat transfer is not allowed.

A structural example:

```
SUBCASE 1
STEP 10
  ANALYSIS=NLSTAT
STEP 20
  ANALYSIS=NLSTAT
STEP 30
  ANALYSIS=NLTRAN
```

- b. Nonlinear chained physics: A nonlinear steady state heat with results used for a nonlinear static structural analysis. Only a steady state heat to a structural nonlinear Statics is allowed.

```
SUBCASE 1
STEP 10
  ANALYSIS=HSTAT
STEP 20
  ANALYSIS=NLSTAT
```

- c. Nonlinear coupled physics: This allows for four combinations of STEP/SUBSTEP within the SUBCASE
- **ANALYSIS=HSTAT** for the first SUBSTEP and **ANALYSIS=NLSTAT** for the second SUBSTEP
  - **ANALYSIS=HTRAN** for the first SUBSTEP and **ANALYSIS=NLTRAN** for the second SUBSTEP
  - **ANALYSIS=HTRAN** for the first SUBSTEP and **ANALYSIS=NLSTAT** for the second SUBSTEP

- **ANALYSIS=HSTAT** for the first SUBSTEP and **ANALYSIS=NLTRAN** for the second SUBSTEP

A coupled analysis example is:

```
SUBCASE 100
STEP 10
  STRESS= ALL
  NLSTRESS=ALL
  NLSTEP=84
SUBSTEP 1
  ANALYSIS=HSTAT
  THERMAL=ALL
  FLUX=ALL
  SPC=35
  LOAD=11
ANALYSIS=NLSTAT
  SPC=2
  LOAD=110
  DISP (PLOT)=1456
STEP 20
  ANALYSIS=NLTRAN (single physics rules follow)
```

- d. Linear perturbation analysis: Linear perturbation analysis is run directly after a nonlinear static (ANALYSIS=NLSTAT) analysis using additional STEP commands containing ANALYSIS=BUCK, MODES, DFREQ, MFREQ, MTRAN, DCEIG, and MCEIG entries.

It should be noted that ANALYSIS=BUCK for linear perturbation is not recommended if the model is highly nonlinear. NLBUCK Case Control command is recommended for highly nonlinear models.

An example is:

```
SOL 400
CEND
TITLE=MSC Nastran SOL 400, Linear Perturbation Analysis
SUBTI=3D General Contact with Large Displacement Turned on
$
SUBCASE 1
STEP 1
  LABEL=Nonlinear Static Analysis with Contact
  ANALYSIS = NLSTATIC
  NLPARM = 1
  BCONTACT = 1
  BOUTPUT=ALL
  SPC = 2
  LOAD = 3
  DISPLACEMENT (SORT1,REAL)=ALL
STEP 2
  LABEL=Linear Perturbation, DFREQ
  ANALYSIS = DFREQ
  DLOAD=200
  FREQ =10
  AUTOSPC=YES
  SPC = 2
  DISPLACEMENT = ALL
```

```
STEP 3
  LABEL=Linear Perturbation, MFREQ
  ANALYSIS = MFREQ
  NLIC STEP 1 LOADFAC 1.0
  METHOD = 30
  DLOAD=200
  FREQ =10
  AUTOSPC=YES
  RESVEC =NO
  SPC = 2
  DISPLACEMENT = ALL
```

- e. Standard linear physics: Runs can include ANALYSIS=STATICS, MODES, BUCKL, DFREQ, MFREQ, MTRAN, DCEIG, MCEIG, SAERO, and FLUTTER. It is recommended to use RIGID=LINEAR if the same answers are desired as in non-SOL 400. These are standard SUBCASE type of analysis. If there are rigid elements present then RIGID=LINEAR is recommended if the same results are desired as in SOLs 103, 105, 107, 108, 110, 111, 112, 145, and 146.

An example is:

```
SUBCASE 2
  LABEL=Linear Static
  ANALYSIS = STATIC
  LOAD      = 1001
  DISP     = 10
  STRESS   = ALL
SUBCASE 101
  DISP     = 10
  STRESS   = ALL
$
STEP 11
  LABEL=Nonlinear Statics, Load 1001
  ANALYSIS = NLSTATIC
  NLPARM   = 11
  LOAD     = 1001
$
STEP 12
  LABEL=Nonlinear Statics, Load 1005
  ANALYSIS = NLSTATIC
  NLPARM   = 11
  LOAD     = 1005
$
STEP 13
  LABEL=Linear Perturbation, Modes
  ANALYSIS = MODES
  METHOD    = 1003
  RESVEC   = NO
  AUTOSPC (NOPRINT) = YES
  DISPL    = ALL
SUBCASE 1004
  LABEL=Get Linear Normal Modes
  ANALYSIS = MODES
  SVECTOR  = ALL
  METHOD    = 1004
```

- f. Nonlinear chained analysis with mesh/time change physics: A standard single physics nonlinear steady state ANALYSIS=HSTAT or transient heat transfer ANALYSIS=HTRAN with either scratch=no or scratch=mini on the job submittal. This is followed by a mechanical job submittal with an ASSIGN hrun='name\_of\_heat\_run.MASTER' and DBLOC DATBLK=(HEATDB) LOGI=hrun in the File Management Section of the Nastran executive and an ANALYSIS=NLSTAT or ANALYSIS=NLTRAN with a TEMP(Load,HSUB,HSTEP,HTIME) in the subsequent mechanical job STEP. The subsequent mechanical job can have both a different mesh than the heat job and different time steps.

An example:

```

SOL 400 (submitted with SCRATCH=MINI from a bulk file named
Course_Mesh_Heat.dat)
CEND
SPC = 1
IC = 10
THERMAL=ALL
SUBCASE 3
  STEP 4
  ANALYSIS=HSTAT
  NLPARM = 1
  LOAD = 202
  SUBCASE 10 (Note if STEP not provided STEP=1 defaulted)
  ANALYSIS=HTRAN
  = 2
  DLOAD = 404
BEGIN BULK

SOL 400 (Mechanical run using solution from
Course_Mesh_Heat.dat)
ASSIGN hrun= 'Course_Mesh_Heat.MASTER' (the ticks ( ' ) are
required)
DBLOC DATBLK=(HEATDB), LOGI=hrun
CEND
TEMPERATURE (INITIAL) = 1
SUBCASE 1
  ANALYSIS=NLTRAN
  STEP 1
  = 1
  SPC = 2
  TEMP (LOAD, HSUBC=3) = 3
  DISPLACEMENT (SORT1, REAL)=ALL
  NLSTRESS = ALL
  STRESS = ALL
  STEP 2
  = 2
  SPC = 2
  TEMP (LOAD, HSUBC=10, HTIME=0.80) = 4
  DISPLACEMENT (SORT1, REAL)=ALL
  NLSTRESS = ALL
  STRESS = ALL
SUBCASE 2
  ANALYSIS=NLTRAN

```

```

STEP 3
= 3
SPC = 2
TEMP (LOAD, HSUBC=10, HTIME=ALL) = 5
DISPLACEMENT (SORT1, REAL)=ALL
NLSTRESS = ALL
stress = all
BEGIN BULK
  
```

- g. Nonlinear response optimization with ESLNRO: A standard single physics nonlinear **ANALYSIS=NLSTAT** is performed with a **NASTRAN ESLOPT=1** in the bulk data file. Besides the usual model grid, loads, element data etc., the bulk data file should contain the standard SOL 200 design criteria.

An example:

```

NASTRAN ESLOPT=1 $
SOL 400
CEND
DESOBJ (MIN) = 10000
ANALYSIS = NLSTATIC
DESSUB = 1
LOAD = 300
SPC = 1
STRESS = ALL
DISP = ALL
NLPARM = 1
BEGIN BULK
  
```

- 4. In SOL 106, the ANALYSIS Case Control command may be used to define a 'linear' perturbation analysis SUBCASE or a user input of the 'stressed' or deformed geometry (normal bulk data input) and 'unload' the structure to determine the unstressed shape SUBCASE.
  - a. 'Linear' perturbation analysis: The ANALYSIS Case Control command may be used to define a 'linear' perturbation analysis SUBCASE, separate from the subcases used to load the model. Normal modes and frequency response subcases with **ANALYSIS=MODES** or **ANALYSIS=DFREQ** will use the final displacement results and loads from the previous nonlinear subcase to generate the stiffness, differential stiffness, and follower force matrices for use in the 'linear' response analyses. Data recovery will be based on the requests above and within the subcase.

An example is:

```

SUBCASE 1
$ LOAD STRUCTURE
LOAD= 100
NLPARM= 100
SUBCASE 2
$ NORMAL MODES
ANALYSIS= MODES
METHOD= 100
DISP= ALL
SUBCASE 3
$ FREQUENCY RESPONSE
ANALYSIS= DFREQ
  
```



```

SET 100= 1 THRU 1000
DISP= 100
SUBCASE 4
$ CONTINUE LOADING STRUCTURE
LOAD= 200
NLPARM= 100
DISP= ALL
SUBCASE 5
$ NORMAL MODES AT THE NEW LOADING
ANALYSIS= MODES

```

In the previous example, subcases 2 and 3 will use the results from subcase 1, subcase 4 will continue the loading application, and subcase 5 will use the results from subcase 4. For the 'linear' analyses, the mass matrix will be based on the undeformed geometry and the damping matrix will be generated using the deformed geometry. This will allow analyses of 'large' displacement results (PARAM, LGDISP, 1), in addition to material nonlinear analyses.

- b. User input of the 'stressed' or deformed geometry: [ANALYSIS=HOT2COLD](#) allows the user to input the 'stressed' or deformed geometry (normal bulk data input) and 'unload' the structure to determine the unstressed shape. This uses an iterative technique with each iteration a nonlinear analysis. This feature will also 'reset' the geometry to the 'unstressed' position for additional subcases. Related PARAMETERS are HTOCTOL, HTOCITS, and HTOCPRT.

An example is:

```

SUBCASE 1
$ UNLOAD STRUCTURE
ANALYSIS=HOT2COLD
LOAD= 100
NLPARM= 100
SUBCASE 2
$ NORMAL MODES UNLOADED STRUCTURE
ANALYSIS= MODES
METHOD= 100
DISP=ALL
SUBCASE 3
$ LOAD the STRUCTURE
LOAD= 100
NLPARM= 100
SUBCASE 4
$ NORMAL MODES OF LOADED STRUCTURE
ANALYSIS= MODES
METHOD= 100
DISP= ALL

```

SUBCASE 1 unloads the model. SUBCASE 2 calculates the modes of the 'undeformed' structure; the differential stiffness and follower force effects will not be included. SUBCASE 3 loads the model. SUBCASE 4 calculates the modes of the loaded structure; the differential stiffness and follower forces effects will be included.

## APRESSURE

## Aerodynamic Pressure Output Request

Requests the aerodynamic pressures in static aeroelastic response.

### Format:

$$\text{APRES} = \left\{ \begin{array}{c} n \\ \text{ALL} \end{array} \right\}$$

### Examples:

APRES=ALL

APRES=6

Describer	Meaning
n	Set identification number of a previously appearing SET command. Only aerodynamic pressures on the referenced aerodynamic boxes will be output (Integer > 0).
ALL	Pressures at all points will be output.

**AUTOSPC**

## Constrains Stiffness Singularities via m-sets or s-sets

Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.

**Format:**

$$\text{AUTOSPC} \left[ \left( \left[ \text{RESIDUAL} \right] \left[ \begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], \left[ \begin{array}{c} \text{NOPUNCH} \\ \text{PUNCH} \end{array} \right], [\text{SID} = n], [\text{EPS} = r1], \right. \right. \\ \left. \left. [\text{EPSSING} = r2], \left[ \begin{array}{c} \text{SPC} \\ \text{MPC} \end{array} \right], \left[ \begin{array}{c} \text{ZERO} \\ \text{NOZERO} \end{array} \right] \right) \right] = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$
**Examples:**

AUTOSPC=YES

AUTOSPC (PRINT, PUNCH, SID=100, EPS=1.E-6, MPC)=YES

Describer	Meaning
RESIDUAL	For SOL 400, applies AUTOSPC to both the residual structure and superelements. See Remarks 6. and 7.  For SOL 101, applies AUTOSPC to linear analysis with contact. See Remark 9.
PRINT	Enables the printout of a summary table of singularities (Default).
NOPRINT	Disables the printout of a summary table of singularities.
NOPUNCH	Disables the creation of SPC or MPC Bulk Data entries in the PUNCH file (Default).
PUNCH	Generates SPC or MPC Bulk Data entry format in the PUNCH file.
SID=n	Specifies a set identification number for option PUNCH (Default = 999).
EPS=r1	Identifies singularities with a stiffness ratio smaller than r1 to be automatically constrained with single or multipoint constraints. See Remark 2. (Default = 1.E-8).
EPSSING=r2	Identifies the potential singularities with stiffness ratios less than r2. See Remark 2. (Default=1.E-8).
SPC	Applies single-point constraints on degrees of freedom identified as singular. (Default)
MPC	Applies multipoint constraints on degrees of freedom identified as singular.

Describer	Meaning
ZERO	Requests the printout of singularities with zero stiffness ratios in the singularity summary table (Default).
NOZERO	Disables the printout of those singularities with zero stiffness ratios in the singularity summary table.

**Remarks:**

- AUTOSPC specifies the action to take when singularities exist in the stiffness matrix. AUTOSPC=YES means that singularities will be constrained automatically. AUTOSPC=NO means that singularities will not be constrained. If AUTOSPC=NO, then the user should take extra caution analyzing the results of the grid point singularity table and the computed epsilons. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for details of singularity and mechanism identification and constraint.
- Singularity ratios smaller than EPSSING are listed as potentially singular. If AUTOSPC=YES, then the identified singularities with a ratio smaller than EPS will be automatically constrained. If EPSSING has the same value as EPS, then all singularities are listed. If EPSSING is larger than EPS, the printout of singularity ratios equal to exactly zero is suppressed. EPSSING must be greater than or equal to EPS. If not, the program will set EPSSING equal to EPS.
- If the PUNCH keyword is specified, then automatically generated SPCs or MPCs are placed in SPCi or MPCi Bulk Data entry format on the PUNCH file.
- By default, in all solution sequences except 106, 129 and 400, the auto-SPC operation is performed for both superelements and the residual structure. (Auto-SPC processing is disabled in heat transfer analysis). In SOLs 106, 129 and 400, the default is to perform the operation only on superelements. If it is desired to perform auto-SPC on the residual structure in:
  - SOLs 106, then specify PARAM,AUTOSPCR,YES and is only applied to the omitted degrees-of-freedom. The AUTOSPC command is ignored for the residual structure.
  - SOLs 129, then specify PARAM,AUTOSPCR,YES. The AUTOSPC command is ignored for the residual structure.
  - For SOL 400, see Remarks 6. and 7.
- The MPC option may be somewhat more expensive than the SPC option. However, it provides more realistic structural modeling than the SPC. When the MPC option is selected, the multipoint constraint may be applied on some degree of freedom for which the stiffness matrix is identified as singular. If the MPC is inapplicable to some degree of freedom, the SPC is used instead.
- For SOL 400, if RESIDUAL option is requested, the AUTOSPC operation is applied to both the residual structure and the superelements. Without RESIDUAL option, the AUTOSPC operation is applied to the superelements only. For default (no AUTOSPCE command), AUTOSPC operation is not applied to the residual structure, but it is applied to the superelements. Both parameters PARAM,AUTOSPC and PARAM,AUTOSPCR have no effect in SOL 400. Please note that the AUTOSPC (RESIDUAL) command should not be used in the geometrical nonlinear analysis, because it may over constrain the structural model.

7. For SOL 400, the AUTOSPC (RESIDUAL) command can be placed above the subcase level, between subcase and step, and below the step level. The AUTOSPC operation is performed each step of a subcase if it is required. In the following example, step 10 uses SPC option, step 20 uses MPC option, and no AUTOSPC operation is performed for step 30.

```
SUBCASE 1
  STEP 10
    AUTOSPC (RESIDUAL, SPC) = YES
    LOAD = 10
  STEP 20
    AUTOSPC (RESIDUAL, MPC) = YES
    LOAD = 20
  STEP 30
    LOAD = 30
```

For superelements, only one AUTOSPC command can be specified. If there are multiple AUTOSPC commands in the Case Control packet, the one for the first step of the first subcase will be used. In the previous example, the AUTOSPC under step 10 is used.

8. For SOL 400, AUTOSPC equals YES for linear perturbation if DOMAINSOLVER ACMS is defined in the executive system.
9. For SOL 101 with contact, user can turn on AUTOSPC by using RESIDUAL option. i.e., when AUTOSPC(RESIDUAL)=YES, AUTOSPC will be turned on for linear analysis with contact.
10. This entry is not supported by SOL 600. Please enter PARAM,AUTOSPC in the bulk data.

## AUXCASE

### Auxiliary Model Case Control Delimiter

Indicates (delimits) the beginning of Case Control commands for an auxiliary model in SOL 200.

#### Format:

AUXCASE

#### Examples:

AUXCAS  
AUXC

#### Remarks:

1. AUXCASE commands must follow the primary model Case Control commands.
2. All Case Control commands following this entry are applicable until the next AUXCASE command, or the BEGIN BULK delimiter. Commands from preceding Case Control Sections are ignored.
3. Each auxiliary model Case Control must be delimited with the AUXCASE command.
4. The AUXMODEL command is used to associate the auxiliary model Case Control with a particular auxiliary model.

**AUXMODEL**

## Auxiliary Model Identification Number

References an auxiliary model for generation of boundary shapes in shape optimization.

**Format:**

AUXMODEL=*n*

**Examples:**

AUXMODEL=4

AUXM=4

Describer	Meaning
<i>n</i>	Auxiliary model identification number. (Integer > 0)

**Remarks:**

1. AUXMODEL references a particular auxiliary model for analysis and may only be specified in the auxiliary model Case Control Section.
2. See the BEGIN BULK delimiter for the Bulk Data definition of an auxiliary model.

## AVLEXB

### Control for MSC Nastran-AVL EXCITE™ Interface

Control for MSC Nastran-AVL EXCITE™ Interface: (1) EXB file export from SOL 103 and SOL 400, (2) EXCITE™ results import and data recovery in SOL 111, 112 and 400.

#### Format:

```
AVLEXB [EXBBODY = {YES, NO}],  
      [MASINVAR = {FULL, FIRST, NONE}],  
      [RECOVRYM = {YES, NO}],  
      [EXBOSET = {U1-U5, ALL}],  
      [OUTGSTRS = {YES, NO}],  
      [OUTGSTRN = {YES, NO}],  
      [V1ORTHO = {-1.0, value1}],  
      [V2ORTHO = {1.0e8, value2}],  
      [MFFEXP = {YES, NO}],  
      [NOD6 = {YES, NO}],  
      [AVLPOST = {YES, NO}],  
      [POSTUNT = {INP4_UNIT_NUM}],  
      [EXBONLY = { YES, NO}]
```

#### Examples:

EXB file export

```
AVLEXB EXBBODY = YES
```

AVL EXCITE™ results import and data recovery

```
AVLEXB EXBBODY=NO AVLPOST=YES POSTUNT=130
```



Describer	Meaning
EXBBODY	Controls the output of MSC Nastran-AVL EXCITE™ (Default = NO): <ol style="list-style-type: none"> <li>1. NO: Do not output AVL EXB Flexible body.</li> <li>2. YES: Output AVL EXB Flexible body.</li> </ol>
MASINVAR	Requests the type of mass invariants to be computed (Default = FULL) : <ol style="list-style-type: none"> <li>1. FULL: All inertia invariants are computed.</li> <li>2. FIRST: Only first order inertia invariants are computed.</li> <li>3. NONE: No inertia invariants are computed.</li> </ol>
RECOVRYM	Controls the output of recovery (output transformation) matrix (Default = No) : <ol style="list-style-type: none"> <li>1. YES: Output A-set to (O+M)-set recovery matrix.</li> <li>2. NO: Do not output the A-set to (O+M)-set recovery matrix.</li> </ol>
EXBOSET	Selects a set of DOFs for controlling the output of recovery matrix (Default = ALL): <ol style="list-style-type: none"> <li>1. U1-U5: DOFs in the user defined set only are included in the O-set output to the EXB file and the partial recovery matrix output for these DOFs. These need to be defined in the bulk data using the USET/USET1 bulk data entry.</li> <li>2. ALL: Complete O-set in the model is retained for recovery.</li> </ol>
OUTGSTRS	Outputs Grid Point Modal Stresses (Default = NO) to the MSC Nastran op2 requires PARAM,POST,1 or PARAM,POST,-1.
OUTGSTRN	Outputs Grid Point Modal Strains (Default = NO) to the MSC Nastran op2 requires PARAM,POST,1 or PARAM,POST,-1.
V1ORTHO	Lower frequency bound of the Craig-Bampton modes in cycles/unit time. value1: Value of the lower bound.
V2ORTHO	Higher frequency bound of the Craig-Bampton modes in cycles/unit time. value2: Value of higher bound.
MFFEXP	Requests output of Mass matrix of O+M set (Default = No). MFFEXP = YES requires the user to request full recovery (RECOVRYM=YES and EXBOSET = ALL).

Describer	Meaning
NOD6	<p>If model is pure bar/beam elements with concentrated masses, this option can output the element dictionary table as well as element stiffness matrix (Default = No):</p> <ol style="list-style-type: none"> <li>1. YES: Output KDICT &amp; KELM</li> <li>2. NO: Do not output KDICT &amp; KELM</li> </ol> <p>If NOD6 = YES these parameters are automatically set MASINVAR = NONE, RECOVERYM = NO, MFFEXP = NO, EXBOSET = YES</p>
AVLPOST	<p>Requests data recovery using the INP4 file generated by AVL EXCITE™ (Default = No):</p> <ol style="list-style-type: none"> <li>1. YES: Request data recovery</li> <li>2. NO: data recovery</li> </ol> <p>AVLPOST = YES requires EXBBODY = NO as the MSC Nastran AVL EXCITE™ interface cannot generate the EXB file and conduct data recovery in the same run.</p>
POSTUNT	<p>Unit number of the assigned INP4 file generated by AVL EXCITE™ which will be used for data recovery.</p>
EXBONLY	<p>Requests standard DMAP solution and data recovery following MSC Nastran-AVL EXCITE™ interface (Default = NO):</p> <ol style="list-style-type: none"> <li>1. NO: Proceed with standard DMAP solution and data recovery after generating EXB file.</li> <li>2. YES: Executes only MSC Nastran-AVL EXCITE™ interface.</li> </ol>

### Remarks:

1. It is highly recommended that the user should run MSC Nastran for Step-1 with maximum smp setting. Calculation of higher order mass invariants are very computation intensive; to make their calculation faster MSC Nastran leverages efficient shared memory parallelization (SMP).
2. To minimize the data storage and enable efficient data recovery in MSC Nastran, the use of MSC Nastran-AVL EXCITE™ Interface requires the use of EXTSEOUT feature. The use of this feature would be in its standard MSC Nastran capacity in SOL 103 or 400, e.g.:

```

ASSIGN OUTPUT2='crankextse.op2',UNIT=80,DELETE
...
EXTSEOUT (ASMBULK,EXTBULK,EXTID=20) DMIGOP2=80
...

```

During step-1 (EXB file export), to limit the size of external superelement (SE) .op2 the user should only request outputs for sets of physical quantities that are of interest. For, e.g., displacement and stress on surface nodes:

```
DISP (PLOT) = 101
...
STRESS (PLOT) = 102
```

above, the displacement and velocity can only be recovered on set 101 and stresses on set 102 during step-2 (data recovery).

DISP=ALL, STRESS=ALL, STRAIN=ALL, etc., should be used carefully for large models during step-1 as it could lead to significant performance degradation and large external SE .op2 file size.

3. AVL EXCITE™ supports the following flexbody types which will be set automatically based on the user inputs of AVLEXB case controls:
  - a. CON6: Is General large motion flexbody which is selected when the user enters “NOD6=NO” and “MASINVAR=FULL/FIRST”.
  - b. SMOT: Is Small motion flexbody which is selected when the user enters “NOD6=NO” and “MASINVAR=NONE”.
  - c. NOD6: Is a special flexbody model consisting of pure bar/beam (1-D) elements with concentrated masses. It is selected when the user enters “NOD6=YES”. Here the dictionary table for bar/beam elements for each bar/beam element shall be outputted into the EXB file. NOD6 automatically means that the following are not calculated and output into the EXB file: (1) Mass invariants, (2) Recovery Matrix, (3) A-set Orthonormalization and associated Eigenvalue/Eigenvector and (4) Component modes.
4. MSC Nastran does not export KELM to EXB file for NOD6 body. AVL EXCITE™ v2019 provides a utility to calculate and write KELM to existing EXB file.
5. Degree of freedom (dof) table (DOFT table) is written to EXB file for the following dof sets:
  - a. A-set : Dof table for a-set are always written out to the EXB file.
  - b. O+M-set : Dof table for o+m-set are only written out to the EXB file if the recovery matrix is requested by the user. In case the user selects partial recovery by specifying the EXBOSET only a subset of o+m-set would be output in the dof Table.
6. GEOM table contains the coordinates of nodes in MSC Nastran basic coordinate system. Two table of coordinates are always written to the EXB file: (1) A-set and (2) F+M-set.
7. OUTGSTRS or OUTGSTRN entries require the use of the standard MSC Nastran STRESS= or STRAIN= Case Control commands used in conjunction with GPSTRESS= or GPSTRAIN= Case Control commands to produce modal grid point stress or strain. They also require the use of PARAM,POST,-1 or PARAM,POST,1. GSTRESS(PLOT)= or GPSTRAIN(PLOT)= will suppress grid stress or strain output to the MSC Nastran .f06 file. The modal grid point stress or strain are not output to the EXB file but to standard MSC Nastran .op2 file.
8. The default value of “MASINVAR = FULL” calculates all the mass invariants. In case of large models the calculations of higher order mass invariants can be time consuming; hence, it is recommended that the user first run with either “MASINVAR = FIRST” or “MASINVAR =NONE” to validate the model.
9. MFFEXP = YES requests output of Mass matrix of F+M set and requires the user to request full recovery (RECOVRYM=YES and EXBOSET = ALL).

10. Typically SOL 400 is used to produce a preloaded AVL EXCITE™ EXB file.
  - a. Standard SOL 400 nonlinear analysis prohibits the existence of an o-set; hence the EXTSEOUT case control must be used (see above Remark 2.).
  - b. In the preload run the structure should be statically supported and follower loading must be applied as a self equilibrating load set (not with SPC relationships!). In the ANALYSIS=MODES step, usually, the structure should be a free-free structure to preserve its six rigid body modes. In order to produce modal amplitudes and mode shapes and to ensure residual vector calculations, SPOINTs and Q-sets are required.

The example below is a typical SOL400 problem setup:

```

ASSIGN OUTPUT2='sepl3rbe.out2',UNIT=80,DELETE
SOL 400
CEND
$
$ Initiate an Nastran-AVL-EXCITE™ interface run
$ AVL REQUIRED ABOVE SUBCASE
AVLEXB EXBBODY=YES, EXBONLY=YES
SUBCASE 1
$ Preload
STEP 10
$ Static load and support for preload
SUBTITLE = PRELOAD
ANALYSIS = NLSTATICS
NLSTEP = 110
LOAD = 120
SPC = 130
BCONTACT = 140
SPCF = ALL
$ Generate stress and strain grid shapes
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL
$ Modal Step for Producing EXB file
$ Default: Select the end of previous load step to output
$ AVL EXCITE™ EXB
STEP 20
$ EXTSEOUT must be used to as Standard SOL 400 nonlinear analysis
$ prohibits the existence of an o-set
EXTSEOUT(ASMBULK,EXTBULK,EXTID=100) DMIGOP2=80
ANALYSIS = MODES
$ Select real Eigen Value Parameters
METHOD = 210
$ Turn residual vectors on
RESVEC = COMPONENT
STRESS(PLOT) = ALL
STRAIN(PLOT) = ALL
GPSTRESS(PLOT) = ALL
GPSTRAIN(PLOT) = ALL

```

- c. In the above example, the SPC set in the ANALYSIS=NLSTAT must be a static (nonredundant) constraint condition. Note that in the ANALYSIS=MODES STEP, the SPC constraint set has been removed. The definition of the attachment a-set for identifying attachment points and for q-set for specifying the desired number of modal amplitudes for orthonormalization should be done in the main bulk data section, e.g.:

```

$ Main Bulk section
$ Attachment point and component mode (A-SET) selection
ASET1,123456,1,11,111,121
QSET1,0,100001,THRU,100020

```

- d. If CONTACT is required as part of the preloading, it is highly recommended that the friction option be turned on by using an appropriate BCPARA bulk data entry setting, e.g.,

```

$ Select bilinear Coulomb friction for all subcases
BCPARA, 0, FTYPE, 6

```

If contact friction is not turned on, the tangential motion between the two parts coming into contact will most likely not be constrained and incorrect or fatal results will occur.

11. The AVL EXCITE™ solver expects singularities, other than the rigid body modes, of the model to be constrained; hence, it is recommended that the user constrain singularities automatically using the PARAM,AUTOSPC in SOL 103 and AUTOSPC (RESIDUAL) command in SOL 400.
12. After conducting the AVL EXCITE™ simulation the data recovery is conducted in SOL 111, SOL 112 and SOL 400 using the EXTSEOUT feature. Using External SE to conduct data recovery would make the data recovery faster and minimize the database size by storing only the information required for recovery. The user interface for data recovery is straightforward requiring the attachment of the SE databases, stored on the .op2 file, generated in the SE generation step during EXB file export together with the ASM and PCH files.
13. During the data recovery it should be noted that PARAM,POST setting for the External SE are not picked up from the main bulk. If physical quantities inside External SE (stresses,etc) are to be output to op2 the user can add PARAM,POST in the case control if they want it applied globally or add PARAM,POST manually to the External SE pch file.
14. Virtual mass is supported in MSC Nastran-AVL EXCITE™ Interface through the use of standard VMOPT parameter entry.
15. For STRUCTURE and FLUID coupled system, METHOD(COUPLED) can be used for EXB generation. However, all nodes on fluid face needs to be coupled with structure for AVLEXB otherwise, mass and inertia of moment will be incorrect in EXB.

## AXISYMMETRIC

## Conical Shell Boundary Conditions

Selects boundary conditions for an axisymmetric shell problem or specifies the existence of fluid harmonics for hydroelastic problems.

### Format:

$$\text{AXISYMMETRIC} = \left\{ \begin{array}{c} \text{SINE} \\ \text{COSINE} \\ \text{FLUID} \end{array} \right\}$$

### Example:

AXISYMMETRIC=COSINE

Descriptor	Meaning
SINE	Sine boundary conditions will be used.
COSINE	Cosine boundary conditions will be used.
FLUID	Existence of fluid harmonics.

### Remarks:

1. This command is required for conical shell problems.
2. If this command is used for hydroelastic problems, at least one harmonic must be specified on the AXIF command.
3. See the [Surface Elements](#) in the *MSC Nastran Reference Guide* for a discussion of the conical shell problem.
4. The sine boundary condition will constrain components 1, 3, and 5 at every ring for the zero harmonic.
5. The cosine boundary condition will constrain components 2, 4, and 6 at every ring for the zero harmonic.
6. SPC and MPC Case Control commands may also be used to specify additional constraints. See [Case Control Commands](#).

**B2GG**

## Direct Input Damping Matrix Selection

Selects direct input damping matrix or matrices.

**Format:**

B2GG=name

**Examples:**

B2GG = BDMIG

B2GG = BDMIG1, BDMIG2, BDMIG3

B2GG = 1.25\*BDMIG1, 1.0\*BDMIG2, 0.82\*BDMIG3

SET 100 = B1, B2

B2GG = 100

Describer	Meaning
name	Name of $[B_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list, with or without factors (see Remark 5.).

**Remarks:**

1. DMIG matrices will not be used unless selected.
2. Terms are added to the damping matrix before any constraints are applied.
3. The matrix must be symmetric, and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
4. A scale factor may be applied to this input via the PARAM, CB2 entry. See [Parameters](#).
5. The formats of the name list:
  - a. Names without factor.  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be with a factor including 1.0.

**B2PP**

## Direct Input Damping Matrix Selection

Selects direct input damping matrix or matrices.

**Format:**

B2PP=name

**Example:**

```
B2PP = BDMIG
B2PP = BDMIG1, BDMIG2, BDMIG3
B2PP = 5.06*BDMIG1, 1.0*BDMIG2, 0.85*BDMIG3
B2PP = (1.25, 0.5) *BDMIG1, (1.0, 0.0) *BDMIG2, (0.82, -2.2) *BDMIG3
```

Descriptor	Meaning
name	Name of $[B_{pp}^2]$ matrix that is input on the DMIG or DMIAx Bulk Data entry, or name list, with or without factors. See Remark 7. (Character).

**Remarks:**

1. DMIG entries will not be used unless selected.
2. B2PP is used only in dynamics problems.
3. DMIAx entries will not be used unless selected by the B2PP command.
4. The matrix must be square or symmetric, and field 4 on the DMIG,name Bulk Data entry must contain a 1 or 6.
5. It is recommended that PARAM,AUTOSPC,NO be specified. See the [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
6. The matrices are additive if multiple matrices are referenced on the B2PP command.
7. The formats of the name list:
  - a. Names without factor  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parentheses as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.



## BC Boundary Condition Identification

Identifies multiple boundary conditions for normal modes, buckling, and flutter analysis in SOLs 103, 105, 145, and 200.

### Format:

BC=n

### Example:

BC=23

Describer	Meaning
n	Identification number (Integer > 0).

### Remarks:

1. In SOLs 103, 105, 145, and 200, BC is required in each subcase if multiple boundary conditions are specified for normal modes, buckling, or flutter analysis.
2. If only one boundary condition is specified, then BC does not have to be specified, and n defaults to zero.

## BCHANGE

## Contact Bodies Definition Change Selection

Selects the changes of the definition of contact bodies in SOL 400.

### Format:

BCHANGE=n

### Example:

BCHANGE=10

Describer	Meaning
n	Set identification of the BCHANGE Bulk Data entry, see Remark 2. (Integer > 0)

### Remarks:

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of the BCHANGE Bulk Data entry is defined on the BCONTACT Case Control command if applicable; however, the SID on the BCHANGE Case Control command can overwrite it.

## BCONCHK

## Contact Model Check

This entry is used to activate contact model check before analysis in SOL 101, 103, 105, 107-112, 200 and 400.

With the contact model check, the initial contact status is checked and output. Displacement-like vectors and norm distance of active nodes to the corresponding contact body and distance to the body are generated. Sign of the distance stands for its status: positive for gap; 0.0 on surface; negative for penetration. It is a global case control and must be above SUBCASE.

Note that BCONCHK does not support BCONTACT=ALLBODY, i.e., no contact status check is reported when BCONTACT=ALLBODY.

### Format:

$$\text{BCONCHK} \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] = \left\{ \begin{array}{l} \text{RUN} \\ \text{STOP} \\ \text{STEP} \end{array} \right\}$$

### Example:

BCONCHK (PRINT) = Run

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

RUN	Execute contact model check and proceed analysis normally.
STOP	Only run contact check and exit job before normal analysis.
STEP	Execute contact check at each output request LOAD/TIME step.

## BCONTACT

## Selects 3D Contact Surfaces

This entry activates contact analysis and selects contact definition tables.

### Overview of contact type definition:

**General Contact** - two bodies may come into contact and separate at any point in the simulation and the bodies may slide along each other's surface with or without friction. The fundamental constraint is no relative normal displacement when bodies are in contact. General contact is available in SOL 101, SOL 400, SOL 600 or SOL 700 only. For SOL 101, contact is the only source of nonlinearity, no large displacements, large rotations or material nonlinearity is available.

**Glued Contact** - two bodies may come into contact and separate at any point in the simulation, but when in contact there is no relative sliding. The fundamental constraint is no relative normal displacement or tangential displacement when bodies are in contact. One can consider this equivalent to two surfaces that have infinite friction. Note that the word glued, only refers to the constraint on the tangential behavior. Bodies that are in glued contact may lose contact if the separation (force or stress based) separation criteria is exceeded, due to Unglue or Breaking Glue.

**Step Glued Contact** - this is available for SOL 400 only. Step Glued contact is activated using a value a negative value of IGLUE for each contact pair. It is similar to Glued Contact, there are two conditions.

1. The contact status will be checked at the beginning of the step, and those nodes or segments that are in contact will remain in glued contact for the entire step. The constraints will change due to large rotations. Furthermore if a large tensile force or stress developed over the interface in the current loadcase, no separation would occur for these regions which are initially in contact. Performing an Unglue or Breaking glue would also not be enforced during the step for these regions. This may be successfully used to model the union of dissimilar meshes, where at a later time one wanted to separate the bodies (like opening of a door).
2. When using Step Glue conventional contact occurs for the nodes/segments of the body which are not in contact at the beginning of the step. That means when they come into contact, they will glue, but they may separate within the same step.

**Permanently Glued Contact** -this is a special case of contact, where the initial configuration is used to determine the contact constraint, and these contact changes do not change throughout the analysis. Nodes or segments which are not initially in contact do not come into contact, and in fact may penetrate the model. The constraint is a glue type, meaning there will be no relative normal or tangential displacement. Permanent glued may be used to connect dissimilar meshes or for simple assembly modeling when no other contact occurs. The bodies will never separate. Permanent glue should not be used in models that experience large rotations. It is applicable to SOL 101, 103, 105, 107, 108, 109, 110, 111, 112, SOL 200 and SOL 400. Permanent Glued contact is activated if the BCTABLE or BCTABL1 that is referenced in the first Loadcase (SOL 100\*) or in the first Step (SOL 200) has a value of IGLUE greater than zero for all contact body pairs.

If the user requires conventional (general) contact for the complete simulation, but Permanent Glued contact is invoked, enter bulk data BCPARA,0,NLGLUE,1 to deactivate the Permanent Glue in a subsequent step.

If large rotation/deformation effect present, turn on SYSTEM(758)=2 will switch Permanently Glued Contact to Step Glued Contact automatically in SOL 400.

Because glued contact is very useful in assembly modeling problems encountered in engineering practice, several special cases are considered as well.

**Moment Carrying Glue** - For General Contact - when a node or surface comes into contact the default condition is that constraints are only placed on the translational degrees of freedom. When Moment Carrying Glue is activated the rotational degrees of freedom are also constrained to ensure moment carrying behavior. This may be used with either beam elements or shell elements.

**Symmetry Contact** - Deformable contact with a rigid Symmetry surface. In this case no friction is allowed. Furthermore no separation is allowed, and finally if the rigid surface is contacted by beams or shells - the rotations are automatically constrained to satisfy the symmetry constraints. You do not need to specify any additional input, other than specifying that the rigid surface is a SYMM body.

### Format:

$$\text{BCONTACT} = \left\{ \begin{array}{c} n \\ \text{ALLBODY} \\ \text{NONE} \end{array} \right\} \text{ or } \left\{ \begin{array}{c} \text{ALLELE} \end{array} \right\} \text{ or } \text{AUTO}(\text{ctype})$$

↑  
SOL 700 only

### Examples:

```
BCONTACT = 5
BCONTACT=ALLBODY
BCONTACT=AUTO, PGLUE
```

Describer	Meaning
n	Identification number of a BCTABLE, BCTABL1, BCONECT, BCHANGE, and/or BCMOVE Bulk Data entry.
ALLBODY	All bodies defined using all the BCBODY entries can potentially contact each other. This option can only be used if it applies to all subcases. (all SOLs) In SOL 400, when BCONTACT=ALLBODY, in any STEP, all BCBODY entries defined in the model can potentially contact each other. In this case, no BCTABLE, BCTABL1, BCONECT is required. See also Remark 3.

Describer	Meaning
NONE	All contact definitions (BCTABLE, BCTABL1, BCONECT, BCBODY, and BCBODY1) are ignored. For SOL 400 and SOL 101, if BCONTACT = NONE is entered in any subcase, it applies for all subcases. (Default for SOLs 101 and 400.)
AUTO(,ctype)	Automatic generate contact. ctype is optional. It may be  TOUCH (default) -- general touching contact;  PGLUE -- Permanently glued contact;  GGLUE -- General Glued Contact; and  SGLUE -- Step Glued Contact.  It works with bulk data section entry BCAUTOP, BCAUTOP is optional. See remark 5. and 6.

### Remarks:

1. BCONTACT is recognized in SOLs 101, 400, and 700, and under the special condition of permanent glued contact in SOLs 103, 105, 107, 108, 109, 110, 111, 112 and 200. The standard SOL 200(without calling SOL 400) can only support permanent glue; but when SOL 200 calls SOL 400(or say SOL 400 optimization), it can support all contact types.
2. For SOLs 101 and 400, if the form BCONTACT=n is applied in any loadcase (subcase or step), Nastran looks into the Bulk Data file to get all BCTABLE, BCTABL1 or BCONECT (required), BCMOVE (optional) and BCHANGE (optional), in the same SID=n. The user can always specify Case Control commands, BCMOVE and/or BCHANGE, to select different SID.  
  
BCONTACT=0 can invoke initial preload contact conditions, such that the contact bodies will just touch each other before analysis begins, but it's not necessary. If presence of BCTABLE, 0, initial contact condition will also be processed, no matter if BCONTACT=0 is given or not. (please refer to BCTABLE, 0 for more information).  
  
The contact analysis during any loadcase is dominated by BCONTACT=n (n>0). Without presence of BCONTACT=n (n>0), no contact analysis works in that particular loadcase.  
In SOL 400 one can have multiple contact interaction types in the model meaning general contact, glued contact, step contact on a contact pair basis. These can change from step to step by activating a new BCTABLE or BCTABL1.  
SOL 700 allows only one BCONTACT Case Control command and only one subcase.
3. For SOLs 101 and 400, if the form BCONTACT=ALLBODY is applied in any loadcase (subcase or step), Nastran does not look into corresponding BCTABLE, BCTABL1 or BCONECT but uses the defaults for all entries on BCTABLE, BCONPRG or BCONPRP. If the user wants to specify BCMOVE and/or BCHANGE Bulk Data entries, the BCMOVE and/or BCHANGE Case Control commands must be given.

4. Permanent glued contact for small deformation and rotation is initiated by BCONTACT=n pointing to BCTABLE/BCONNECT referring a valid positive IGLUE field.  
 If all secondary's for the BCTABLE or BCONPRG corresponding to the first loadcase (first subcase and first step) contain IGLUE >0, permanent glued contact with small rotation condition will be used for all SECNDRY entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified. If IGLUE < 0 exists, step glue is activated to support large deformation and large rotation.
5. For glue contact (PGLUE, GGLUE, and SGLUE), default is to implement the moment carrying glue without node projection, i.e., Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. Unless IGLUE is defined in BCAUTOP.
6. CTYPE can be specified by bulk data section BCAUTOP also, the value of CTYPE in BCAUTOP will be used if it is specified.
7. When BCONTACT=AUTO is specified, acg file will be generated. An acg file is a file consists of all generated contact bodies, contact pairs and parameters. The name of acg file can be specified by ASSIGN FMS statement.
8. When BCONTACT=AUTO is specified, only elements in residual structure are used to construct contact bodies in superelement models.
9. BEAMB in bulk data entry BCPARA is set to 1 when BCONTACT=AUTO is specified.
10. When BCONTACT=AUTO is specified and modules are present, the contact generation is in two levels: individual module and across modules. For individual module, contact data generation is activated when an BCAUTOP bulk data section entry exists in the module. If there is no BCAUTOP, no contact data will be generated for the module, and user supplied contact data can be used. For across modules, contact pair generation is activated by MDBCPR entry, and parameters are defined by MDBCATP entry.

## BCONTACT

Selects 3D Contact Surfaces in SOL 600

This entry is used to initiate and control 3D contact in SOL 600.

Standard subcase rules apply and BCONTACT may be specified within each subcase to define which bodies may make contact during that particular subcase. A BCONTACT = 0 above the subcase level is used in SOL 600 to invoke an option such that the contact surfaces will just touch each other before the nonlinear simulation begins. It is highly recommended that if contact is specified for any subcase in SOL 600, and if there are any rigid contact surfaces, BCONTACT = 0 and a matching BCTABLE with ID = 0 be included.

### Format:

$$\text{BCONTACT} = \left\{ \begin{array}{c} n \\ \text{ALLBODY} \\ \text{NONE} \end{array} \right\} \text{ or } \left\{ \begin{array}{c} \text{ALL} \\ \text{ALLELE} \\ \text{ALLELE2} \\ \text{ALLGLUE} \\ \text{ALLGLU2} \\ \text{ALLGLUM} \\ \text{ALLGLM4} \\ \text{ALLGLUP} \\ \text{BCBOX} \\ \text{BCPROP} \\ \text{BCMATL} \end{array} \right\}$$

### Examples:

```
BCONTACT = 5
BCONTACT=ALLBODY
```

Describer	Meaning
n	Identification number of a BCTABLE, BCHANGE, and/or BCMOVE Bulk Data entry. If the model has beams, this option or BCPROP or MCMATL must be used.
ALLBODY	All bodies defined using all the BCBODY entries can potentially contact each other. This option can only be used if it applies to all subcases.
NONE	All contact definitions (BCTABLE, BCBODY) are ignored. BCONTACT = NONE may be used for any subcase desired and/or for increment zero - some subcases can have contact and others no contact.
ALL	All elements in the model can potentially contact with each other (Default). When this option is specified, no 3D contact input is required in the bulk data and, if entered, will be ignored. Warning—this option may take excessive computer time. This option can only be used if it applies to all subcases.



Describer	Meaning
ALLELE	Same as ALL. All elements in the model must be the same kind (either shell or solid). This option may not be used if there are beams, bars or rods in the model.
ALLELE2	Same as ALLELE except shell and solid elements may both be in the model in which case separate bodies for each are formed. (This option requires that both shells and solids be in the model.) All beams, bars and rods in the model will be ignored with respect to contact.
ALLGLUE	Same as ALLELE or ALLELE2 except glued contact will be used for all bodies. The glued contact can vary from increment to increment, however when grids come into contact they cannot separate. BCBODY and BCTABLE entries are created automatically. BCBODY Secondary entries have IGLUE=I (see BCTABLE entry).
ALLGLU2	Same as ALLELE or ALLELE2 except glued contact will be used for all bodies. The glued contact can vary from increment to increment, however when grids come into contact they cannot separate. BCBODY and BCTABLE entries are created automatically. BCBODY Secondary entries have IGLUE=2 (see BCTABLE entry).
ALLGLUM	Same as ALLELE or ALLELE2 except glued contact with moment carrying glue will be used for all bodies. BCBODY and BCTABLE entries are created automatically. BCBODY Secondary entries have IGLUE=3 (see BCTABLE entry).
ALLGLM4	Same as ALLELE or ALLELE2 except glued contact with moment carrying glue will be used for all bodies. BCBODY and BCTABLE entries are created automatically. BCBODY Secondary entries have IGLUE=4 (see BCTABLE entry).
ALLGLUP	Same as ALLELE or ALLELE2 except “permanent glue” is used. This option determines which grids are initially in contact and uses this contact situation in a glued condition for the remainder of the analysis. To use this option, specify PERMGLUE on the SOL 600 Executive Control statement. (SOL 600 only) Note for SOL 600, the PERMGLUE option is the only way contact can be used with SOL 600,101 or SOL 600,103 or other “linear” analyses.
BCBOX	All elements defined within a box-like region as defined by the Bulk Data entry BCBOX can potentially contact each other. See Remark 2.
BCPROP	All elements defined by the Bulk Data entry BCPROP can potentially contact each other. See Remark 2.
BCMATL	All elements defined by the Bulk Data entry BCMATL can potentially contact each other. See Remark 2.

### Remarks:

1. Normally, only one form of this entry may be used in any given analysis. Analysis restarts must use the same form as the original run. An exception is that if BCONTACT = NONE is entered for any subcase, BCONTACT=N may also be specified for different subcases. BCONTACT=ALLxxx cannot be mixed with BCONTACT=NONE or BCONTACT=N in the same input file.

2. Bulk Data entries BCBOX, BCPROP, and BCMATL may be used with BCONTACT = n, wherein case IDs specified on the BCBODY entry and on the BCBOX, BCPROP, and/or BCMATL entries must match.
3. For options ALLELE, ALLELE2, ALLGLUE, ALLGLU2, ALLGLUM, and ALLGLM2, no BCBODY or BCTABLE entries should be included in the model. Nastran will automatically create BCBODY and/or BCTABLE entries as necessary for these options.
4. The ALLELE option requires that the model be made up of either all shells or all solids.
5. The ALLELE2 option requires that the model have both shells and solids.
6. The ALLGLUE, ALLGLU2, ALLGLUM and ALLGLM4 options can be used with models with shells only, solids only for a combination of shells and solids.
7. The ALLGLUP option can either be used with BCBODY/BCTABLE entries or without them. If there are not BCBODY/BCTABLE entries, SOL 600 will create them. If they exist, all SECNDRY lines on all BCTABLE entries must specify IGLUE=1.
8. The ALLG\* options are not available for edge contact. Edge contact may only be run using BCONTACT=N and the corresponding BCTABLE having a FBSH line with COPS1 and/or COPM1 properly specified.

## BCMOVE

## Contact Body Movement Selection

Selects movement of bodies in contact in SOL 400.

### Format:

BCMOVE=*n*

### Example:

BCMOVE=10

Describer	Meaning
<i>n</i>	Set identification of the BCMOVE Bulk Data entry. (Integer > 0)

### Remarks:

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of the BCMOVE Bulk Data entry is defined on the BCONTACT Case Control command if applicable; however, the SID on the BCHANGE Case Control command can overwrite it.

## BEGIN BULK

## Case Control and Bulk Data Delimiter

Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section.

### Format:

```

BEGIN[BULK] [
    AFPM = afpmid
    ARBMODEL = arbmid
    AUXMODEL = auxmind
    MASSID = massid[LABEL = masslabel]
    MODULE= moduleid[APPEND][LABEL = modlabel]
    FLXBDY = flexbody
    SUPER = seid
    TRMC = trimid
    UDS
]
    
```

### Examples:

```

BEGIN BULK
BEGIN AUXMODEL=22
    
```

```

BEGIN BULK TRMC=101
BEGIN TRMC=102
    
```

Descriptor	Meaning
AFPM	Indicates the beginning of an acoustic field point mesh Bulk Data Section.
afpmid	Acoustic field point mesh identification number (Integer > 0).
ARBMODEL	Indicates the beginning of a finite element model FEM for an arbitrary beam cross-section.
arbmid	FEM identification number (Integer > 0).
AUXMODEL	Indicates the beginning of an auxiliary model Bulk Data Section.
auxmid	Auxiliary model identification number (Integer > 0).
FLXBDY	Indicates the beginning of a Adams/Nastran MNF flexbody Bulk Data Section for SOL400 analysis.
flexbody	Flexbody MNF component identification number (Integer>0; Required). Component usually consists of ASET and QSET description for attachment points.
MASSID	Indicates the beginning of the incremental mass case Bulk Data Section.
massid	Incremental mass case identification number (Integer>0; Required).

Describer	Meaning
LABEL	Indicates a massid label.
MODULE	Indicates the beginning of a Module Bulk Data section
moduleid	Module identification number (Integer>0).See Remark 11.
APPEND	Indicates append data to a module.
LABEL	Indicates a Module label. This option will be converted to an MDLABEL Bulk Data entry in the main Bulk Data section.
modlabel	Module label.
SUPER	Indicates the beginning of partitioned superelement Bulk Data Section.
seid	Superelement identification number (Integer $\geq 0$ ).
TRMC	Indicates the beginning of Bulk Data Section for a trim component model. See Remark 9.
trimid	Trim component identification number (Integer>0; Required).
UDS	Indicates the beginning of user defined subroutine section.

### Remarks:

- BEGIN BULK is not required but it is highly recommended that the user supply it. If not specified, then the program will automatically insert one, before the first unique bulk type entry. Parameters are not considered unique bulk type entries so automatic BEGIN BULK inclusion will come after any parameter that precedes entries unique to bulk.
- For an auxiliary model, AUXMID is referenced by the AUXMODEL Case Control command.
- Partitioned Bulk Data Sections defined by BEGIN SUPER are used to define only one superelement each. Bulk Data commands which define superelements are ignored in partitioned Bulk Data Sections.  
  
Superelements specified by a BEGIN SUPER entry can be automatically attached to other superelements based on relative locations of grid points. For connection to the downstream superelement, the global coordinate directions of the attachment grid points of the upstream superelement will be internally transformed to the global coordinate directions of the grid points of the downstream superelement. For displacement data recovery, the output will be in the original global coordinate directions.
- An acoustic field point mesh section defined by BEGIN AFPM is used to define one acoustic field point mesh. Acoustic field point meshes are used for postprocessing of acoustic results in the far field only; i.e., at locations within the acoustic infinite elements.
- The BEGIN SUPER, BEGIN AUXMODEL, and BEGIN AFPM Bulk Data entries must lie between BEGIN BULK and ENDDATA.

6. When employing part superelements using the BEGIN BULK SUPER (or BEGIN SUPER) entry, it should be noted that any parameters that are specified in the main Bulk Data Section apply only to the residual and not to any of the part superelements. Accordingly, to apply certain parameters to all of the superelements, they must be specified in the Case Control Section, or explicitly in all of the BEGIN BULK SUPER (or BEGIN SUPER) portions of the Bulk Data. A common example of such a parameter specification is PARAM,POST, which is used to request postprocessing of results.
7. arbmids can be referenced under the OUTM keyword of the PBMSECT Bulk Data entry to define the geometry of the arbitrary beam cross-section.
8. For model using user defined subroutines, the subroutines can be put in the UDS section. Nastran can build user service from the subroutines using command line keyword uds=model. See the document User Defined Service User's Guide for detail.
9. The parameter WTMASS is supported for trim components with a PARAM, WTMASS, value in each BEGIN TRMC=n Bulk Data.
10. BEGIN MODULE command can only be specified after the BEGIN BULK or an ENDMODULE command has already been specified. In other words, a Module Bulk Data section cannot be nested inside another Module's Bulk Data section.
11. The same module identification number can appear multiple times if APPEND is specified, otherwise a user fatal message will be issued.

**BENDL**

## BAR/BEAM end load Subcase Selection

Function to calculate bar end loads and shear flow.

**Format:**

$$BENDL \left( PRINT = \begin{matrix} YES \\ NO \end{matrix} \right) = \left\{ \begin{matrix} ALL \\ n \\ NONE \end{matrix} \right\}$$

**Examples:**

To output a table of bar end loads/shear flows.

BENDL = ALL

To output a table of bar end loads/shear flows and print table to F06.

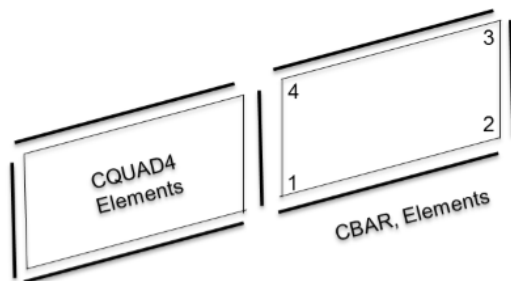
BENDL (PRINT=YES) = ALL

Describer	Meaning
PRINT	Prints bar end loads and panel shear flow to the F06 file.
YES	BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will be printed.
NO	BAR/BEAM end loads and associated CSHEAR, CQUAD4 and CTRIA3 panel shear flows will not be printed. (Default)
ALL	Select all bar/beam/rod type elements associated with “Shell” panels.
n	Identification of a previously defined SET listing “Flange/Boom” elements.
NONE	Turn off BENDL in selected subcases.

**Remarks:**

1. A BENDL Case Control Command is to appear in any subcase for which Bar End Loads and Panel shear flows are required. This entry is only applicable in SOL101 or ANALYSIS=STATIC.
2. The presence of BENDL in a linear statics run will automatically turn on GPFORCE=ALL as well as PARAM, NOELOF, 1.
3. Only CSHEAR, CQUAD4, CTRIA3, CQUADR, and CTRIAR panels will contribute to shear flow and bar end load calculations.
4. Include MDLPRM HDF5 to output OUTSFLOW data block to the HDF5 output file.

5. The following figure shows a typical bar end load and shear flow panel.



6. Print and table FORMAT is:

```

BAR END LOADS APPROXIMATIONS (CBAR)
ELEMENT          TOTAL END LOADS
ID.             END 1          END 2
6              -1.994413E+03  -1.136868E-13
11             2.082592E+03  3.638604E-01
16             -4.721992E+02  -7.329887E+01
21             4.984904E+02  -4.987337E+02
  
```

```

BAR END LOADS SHEAR FLOW APPROXIMATIONS (C
QUAD4)
ELEMENT ID.          SHEAR 12          SHEAR 23          SHEAR 34          SHEAR 41
                   1          9.972063E+01  -2.493060E+02  -9.972063E+01  3.989003E+01
  
```



**BOUTPUT**

## Line Contact or 3D Contact Output Requests

Selects contact regions for output.

**Format:**

$$\text{BOUTPUT} \left[ \left( \left[ \text{SORT1} \right], \left[ \text{PRINT}, \text{PUNCH} \right] \right), \left[ \text{PLOT} \right] \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

```
BOUTPUT=ALL
BOUTPUT=5
```

Describer	Meaning
SORT1	Output is presented as a tabular listing of secondary nodes for each load or time depending on the solution sequence.
SORT2	Output is presented as a tabular listing of load or time for each secondary node.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Histories of all the secondary nodes (all nodes for 3D Contact) listed in all the BOUTPUT Bulk Data entries are output. If no BOUTPUT Bulk Data entries are specified, histories of all the secondary nodes in all the contact regions are output.
n	Set identification of previously appearing SET command. Only contact regions with identification numbers that appear on the SET command are selected for output. If there is a BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for secondary nodes listed in the Bulk Data entry are output. If there is no BOUTPUT Bulk Data entry for a contact region selected via the set command, histories for all the secondary nodes in that contact region are output.
NONE	Result histories for secondary nodes are not calculated or output.

**Remarks:**

1. BOUTPUT is processed in SOLs 101, 106, 129, 153, 159, 400, and 600 only.  
For other solution sequences, only the initial contact status may be output when presence of BCTABLE or BCTABL1 with ID=0.
2. SORT1 is the default in SOLs 106 and 153. SORT2 is the default in SOLs 129 and 159.
3. Only SORT1 is available for 3D Contact.
4. The initial contact status may be output when presence of BCTABLE/ID=0. Please note that all the contact force/stress in the initial state are zero. Only the contact status is relevant.

**BSQUEAL**

## Brake Squeal Analysis Data Selection

Selects data for brake squeal analysis in SOL 400.

**Format:**

BSQUEAL= n

**Example:**

BSQUEAL=10

Describer	Meaning
n	Set identification number of a BSQUEAL Bulk Data entry (Integer > 0).

**Remark:**

1. This command is used only in SOL 400 for brake squeal analysis with 3D Contact.
2. This command is not available with segment-to-segment contact.

## CAMPBELL

### Campbell Diagram Parameters

Specifies Campbell Diagram parameters.

**Format:**

CAMPBELL= n

**Example:**

CAMPBELL= 10

Describer	Meaning
n	Identification number of a CAMPBLL Bulk Data entry (Integer > 0).

**Remark:**

1. CAMPBELL option is supported for SOL 107, SOL110, SOL 200 and SOL 400 analysis.
2. When there is a Case Control request for Campbell diagram, the selected RGYRO Bulk Data entry must have the ASYNC option specified in its SYNCFLG field (Field 3). Otherwise, the program terminates the execution with an appropriate fatal message.

**CLOAD**

## Static Load Request for Upstream Superelement Loads

Requests a CLOAD Bulk Data entry that defines a list of superelement loads and their scale factors in nonlinear static analysis only.

**Format:**

CLOAD=*n*

**Example:**

CLOAD=1 5

Describer	Meaning
<i>n</i>	Identification number of a unique CLOAD Bulk Data entry (Integer > 0).

**Remarks:**

1. This command may only appear in the residual structure subcases (see the Case Control command, [SUPER \(Case\)](#)) and, if used, it must be specified in all of them.
2. The CLOAD Bulk Data entry must reference previously processed LSEQ (load sequence) Bulk Data that was requested by LOADSET Case Control commands on the upstream (SUPER ≠ 0) subcases.
3. The resulting load is added to those produced by LOAD and TEMP(LOAD) Case Control commands in the residual structure subcases.

## CMETHOD

## Complex Eigenvalue Extraction Method Selection

Selects complex eigenvalue extraction parameters.

### Format:

CMETHOD=n

### Example:

CMETHOD=77

Describer	Meaning
n	Set identification of EIGC (and EIGP) Bulk Data entry (Integer > 0).

### Remarks:

1. The CMETHOD command must be specified in order to compute complex eigenvalues.
2. See description of the parameter, [UNSYMF](#), to perform complex eigenvalue analysis in SOL 106.

**CMSENRGY****Component Modal Synthesis Energy Output Request**

Requests the form and type of component modal synthesis (CMS) energy output.

**Format:**

$$\begin{aligned}
 & \text{CMSENRGY} \left( \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \text{ESORT} = \left\{ \begin{array}{c} \text{MODE} \\ \text{ASCEND} \\ \text{RATIO} \end{array} \right\} \right], \\
 & \left[ \text{RESPONSE} = \left\{ \begin{array}{c} \text{BOTH} \\ \text{MODAL} \\ \text{FORCED} \end{array} \right\} \right], \left[ \text{CMSE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{NONE} \\ \text{TOTAL} \\ \text{QSET} \end{array} \right\} \right], \left[ \text{CMKE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{NONE} \\ \text{TOTAL} \\ \text{QSET} \end{array} \right\} \right], \\
 & \left[ \text{CMDE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{NONE} \\ \text{TOTAL} \\ \text{QSET} \end{array} \right\} \right], \left[ \text{FILTER} = \left\{ \begin{array}{c} \text{0.001} \\ \text{fratio} \end{array} \right\} \right], [\text{TOPN} = m] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

**Example:**

CMSENRGY (PHASE, RESPONSE=FORCED, CMSE=TOTAL, CMKE=QSET) = ALL

SET 1001 = 10, 40

CMSENRGY (PUNCH, PRINT, RESPONSE=BOTH, CMSE=ALL, FILTER=0.01) = 1001

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with "PARAM,POST,X" (or the POST Case Control command), while the .h5 binary database file will be generated with "MDLPRM,HDF5,X" specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ESORT

Keyword selecting one of the output sorting options: MODE, ASCEND, or RATIO.

Describer	Meaning	
RESPONSE	MODE	Results are output in order of increasing CMS natural mode number.
	ASCEND	Results are output in order of increasing energy ratio magnitudes.
	RATIO	Results are output in order of decreasing energy ratio magnitudes.
	BOTH	Requests output for both free and forced response solutions.
	MODAL	Specifies that output is to be generated for the free (or real eigenvalue) response solution.
	FORCED	Specifies that output is to be generated for the forced response (modal frequency or modal transient) solution.
CMSE	Keyword requesting output of CMS strain energy ratios.	
CMKE	Keyword requesting output of CMS kinetic energy ratios.	
CMDE	Keyword requesting output of CMS damping energy ratios.	
	ALL	Requests both TOTAL and QSET output.
	NONE	Requests that no CMS energy output be generated.
	TOTAL	Requests CMS energy ratio totals in all component modes of a superelement.
	QSET	Requests CMS energy ratios for individual component modes.
FILTER	Keyword specifying the value of the printed output data filter.	
	fratio	Value of output filter ratio (Default = 0.001).
TOPN	Keyword specifying the number of largest CMS energy ratios to be output.	
	m	The number of largest CMS energy ratios to be output (Default is all ratios).
	n	Results for superelement IDs in SET n will be output.
	ALL	Results for all recovered superelements will be output.
	NONE	No CMS energy ratios will be output.

### Remarks:

1. The CMSENGY command may be requested in the modal solution sequences (SOLs 110, 111, 112, 145, 146, 200) and the real eigenvalue analysis solution sequences (SOLs 103 and 106). It is intended for use when superelements are defined and component modal synthesis techniques are employed. (See the MODALKE and MODALSE Case Control commands for other options.)
2. ESORT, FILTER, and TOPN describers apply only to QSET results output. TOTAL results output is always in increasing order of superelement ID number.
3. QSET CMS energy ratios are output in increasing order of component mode number unless the ESORT keyword specifies a particular sorting order. If a sorting order is specified, the magnitude of the energy ratio is sorted. DESCEND can be used as a synonym for RATIO.



4. The FILTER keyword specifies an absolute value that is used to limit the amount of printed output produced. It is applied to the magnitude of the CMS energy ratio. If the CMS energy ratio magnitude is less than fratio for any natural mode, no output for that natural mode is produced. THRESH can be used as a synonym for FILTER.
5. In order to obtain unforced response (RESPONSE=BOTH or MODAL) output in SOL 111 and SOL 112, a subcase containing the ANALYSIS = MODES option must be present.
6. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data have no meaning.

## COSMSEL

### Select CoSIM Service

Activates Co-Simulation service and select the definition of co-simulation.

#### Format:

COSMSEL = n

#### Example:

COSMSEL = 21

Describer	Meaning
n	Identification number of a COSMSEL Bulk Data entry.

#### Remark:

1. COSMSEL is recognized in SOL 400 only.

**CSSCHD**

## Aerodynamic Control Surface Schedule

Selects control system schedule information.

**Format:**

CSSCHD = n

**Example:**

CSSCHD=10

Describer	Meaning
n	Set identification of a control system schedule that appears on a CSSCHD Bulk Data entry.

**Remark:**

1. One or more CSSCHD Bulk Data entries can be invoked by this Case Control command.

## DBSAVE

## Saving Control Parameter of Datablocks for Advanced Nonlinear Elements

Selects saving of datablocks of advanced nonlinear elements for static and transient nonlinear analysis in SOL 400.

### Format:

DBSAVE=*n*

### Example:

```
DBSAVE =2
SUBCASE=1
  ANALYSIS=HSTAT
SUBCASE=2
  ANALYSIS=NLSTAT
```

```
DBSAVE = -1
SUBCASE=1
  STEP = 1
  ANALYSIS=NLSTAT
  DBSAVE = 1
  STEP = 2
  ANALYSIS=NLSTAT
```

```
SUBCASE=2
  ANALYSIS=NLSTAT
SUBCASE=3
  ANALYSIS=NLSTAT
SUBCASE=4
  ANALYSIS=NLSTAT
```

Descriptor	Meaning
<i>n</i>	selected value to control saving of datablocks of advanced nonlinear elements (Integer > = -1).(Default=0)
-1	No datablocks of advanced nonlinear elements is saved
0	Saving datablocks of advanced nonlinear elements at the end of each loadcase
>0	Saving datablocks of advanced nonlinear elements at the every <i>n</i> th output request of results

### Remarks:

1. DBSAVE can be applied above all the subcases, within subcases, and steps. DBSAVE above all the subcases is going to be applied to all the subcases.
2. If DBSAVE is present both above subcase and in step, the DBSAVE in step is dominant and used to control datablocks saving in this step.
3. When DBSAVE=-1, advanced nonlinear element can't be used in Linear Perturbation or other analysis step with NLIC.

**DEACTEL**

## Elements to be Deactivated for SOL 400 Analysis

Indicates which DEACTEL Bulk Data entry is used to control the elements to be deactivated in a single physics job, or a particular physics pass of a coupled job. SOL 400 for NLSTATIC, and NLTRAN, as well Perturbation analyses only.

**Format:**

DEACTEL=N

**Example:**

DEACTEL=2

Describer	Meaning
N	ID of a matching DEACTEL Bulk Data entry specifying the elements to be deactivated.

**Remarks:**

- Usage is limited to the first subcase (or prior to it), the first step or the first substep of a particular physics pass. Elements are deactivated at the start of the job or the start of the particular physics pass and remain deactivated for the entire job.
- Elements deactivated can not be reactivated at later stage in the analysis.

**DEFORM**

## Element Deformation Static Load

Selects the element deformation set.

**Format:**

DEFORM=n

**Example:**

DEFORM=27

Descriptor	Meaning
n	Set identification number of DEFORM Bulk Data entries (Integer > 0).

**Remarks:**

1. DEFORM Bulk Data entries will not be used unless selected by the DEFORM command in the Case Control Section.
2. DEFORM is only applicable in linear statics, inertia relief, differential stiffness, and buckling problems (SOLs 101, 105, 114, and 200), and will produce a fatal message in other solution sequences.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(Load)), element deformation (DEFORM), and constrained displacement loads (SPC, SPCD).
4. Static, thermal, and element deformation loads should have unique identification numbers.
5. In the superelement solution sequences, if the DEFORM Case Control command is used in a cold start, it must also be specified in the restart.

## DESGLB

## Request Design Constraints at the Global Level

Selects the design constraints to be applied at the global level in a design optimization task.

### Format:

DESGLB=n

### Examples:

DESGLB=10

DESG=25

Describer	Meaning
n	Set identification of a set of DCONSTR or a DCONADD Bulk Data entry identification number (Integer > 0).

### Remarks:

1. If used, this command must occur before the first subcase.
2. The DESGLB Case Control command is optional and invokes constraints that are to be applied independently of a particular subcase. These constraints could be based on responses that are independent of subcases (e.g., WEIGHT, VOLUME, WMPID, FRMASS, and FATIGUE). The DESGLB command must be used when applying constraints to FRMASS, WMPID and FATIGUE responses.
3. The DESGLB Case Control command can be used to invoke constraints that are not functions of DRESP1 entries; e.g., DRESP2 responses that are not functions of DRESP1 responses are subcase independent.
4. Constraints that are applied to responses that span subcases through the use of the DRSPAN command must be invoked using the DESGLB command.

**DESMOD**                      Design Model Name

Assigns the design model parameter used to locate the associated datablocks for merging of two or more SOL 200 models using the MultiOpt application.

**Format:**

DESMOD=name

**Examples:**

DESMOD=FLUTTER

Describer	Meaning
Name	User-specified name to designate the model. (The first 8 characters must be unique)

**Remarks:**

1. The name is an attribute of the datablocks that require merging across models in order to perform a simultaneous design.
2. The parameter is only used with MultiOpt, that merges simultaneous models.
3. A unique value of DESMOD is needed for each model invoked by MultiOpt.
4. If DESMOD is used in a SOL 200 input file that is not being used in MultiOpt, it will result in a fatal error.
5. MultiOpt supports Multi Model Optimization (MMO) and Global Optimization (GO), see [Design Sensitivity and Optimization User's Guide](#)



## DESOBJ

### Design Objective

Selects the DRESP1, DRESP2 or DRESP3 entry to be used as the design objective.

#### Format:

$$\text{DESOBJ} \left[ \begin{array}{c} \text{MAX} \\ \text{MIN} \end{array} \right] = n$$

#### Examples:

```
DESOBJ=10  
DESO=25
```

Describer	Meaning
MIN	Specifies that the objective is to be minimized.
MAX	Specifies that the objective is to be maximized.
n	Set identification number of a DRESP1, DRESP2 or DRESP3 Bulk Data entry (Integer > 0).

#### Remarks:

1. A DESOBJ command is required for a design optimization task and is optional for a sensitivity task. Only one DESOBJ command may appear in a Case Control Section.
2. The referenced DRESPi entry must define a scalar response (e.g., WEIGHT or VOLUME).
3. If the DESOBJ command refers to a global response, such as weight, it should appear above the first subcase. If the DESOBJ command refers to a subcase-dependent response such as an element stress, it should appear in that subcase. If it refers to a subcase dependent response but is inserted above the first subcase, it will select the response from the first subcase for the objective and ignore the responses in subsequent subcases.
4. Using DREPS2 with weight factors, SOL200 can support multiple objective optimization.
5. MSC Nastran MultiOpt utility supports Multi Model Optimization (MMO) and Global Optimization (GO), see [Design Sensitivity and Optimization User's Guide](#)

## DESSUB

### Design Constraints Request at the Subcase Level

Selects the design constraints to be used in a design optimization task for the current subcase.

#### Format:

DESSUB=n

#### Examples:

DESSUB=10  
 DESS=25

Describer	Meaning
n	Set identification of a set of DCONSTR and/or a DCONADD Bulk Data entry identification number (Integer $\geq 0$ ).

#### Remarks:

1. A DESSUB Case Control command is required for every subcase for which constraints are to be applied. An exception to this is 'global constraints', which are selected by the DESGLB Case Control command.
2. All DCONSTR and DCONADD Bulk Data entries with the selected set ID will be used.
3. Constraints cannot be applied to the FRMASS, FATIGUE, or FRFTG response using the DESSUB command. Use the DESGLB command instead.

## DESVAR

### Design Variable Selection

Selects a set of DESVAR entries for the design set to be used.

#### Format:

$$\text{DESVAR} = \begin{bmatrix} \text{ALL} \\ n \end{bmatrix}$$

#### Example:

DESVAR=10

Describer	Meaning
n	Set identification of a previously appearing SET command (Integer > 0). Only DESVAR Case Control commands with IDs that appear on this SET command will be used in the SOL 200 design task.

#### Remarks:

1. Only one DESVAR Case Control command may appear in the Case Control Section and should appear above all subcase commands.
2. The DESVAR Case Control command is optional. If it is absent, all DESVAR Bulk Data entries will be used.

## DISPLACEMENT

## Displacement Output Request

Requests the form and type of displacement or pressure vector output. Note: PRESSURE and VECTOR are equivalent commands.

### Format:

$$\text{DISPLACEMENT} \left[ \left( \begin{matrix} \text{[SORT1]} \\ \text{[SORT2]} \end{matrix} \right), \begin{matrix} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{matrix}, \begin{matrix} \text{[REAL or IMAG]} \\ \text{PHASE} \end{matrix} \right] \begin{matrix} \text{[PSDF, ATOC, CRMS]} \\ \text{RALL} \end{matrix} \right],$$

$$\left[ \begin{matrix} \text{[RPRINT]} \\ \text{[NORPRINT]} \end{matrix}, \text{RPUNCH} \right], \text{[CID]}, \left[ \frac{\text{TM} = f}{\text{T1} = f, \text{T2} = f, \text{T3} = f} \right], \left[ \frac{\text{RM} = f}{\text{R1} = f, \text{R2} = f, \text{R3} = f} \right],$$

$$\text{[CONNECTOR} = \left[ \begin{matrix} \text{[ALL]} \\ \text{m} \end{matrix} \right], \left[ \begin{matrix} \text{[STRUCTURE]} \\ \text{FLUID} \\ \text{BOTH} \\ \text{NONE} \end{matrix} \right] \right] = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix} \right\}$$

### Examples:

```
DISPLACEMENT=5
DISPLACEMENTS (REAL) =ALL
DISPLACEMENT (SORT2, PUNCH, REAL) =ALL
DISPLACEMENT (SORT2, PRINT, PSDF, CRMS, RPUNCH) =20
DISPLACEMENT (PRINT, RALL, NORPRINT) =ALL
DISP (T1=1.-3, T3=1.-2) = ALL
DISP (TM=1.-3, PRINT, PLOT) = ALL
DISP (TM=1.-3, PRINT, PLOT, SORT2) = 20
DISP (CONN=23) =54
DISPLACEMENT (PLOT, PRINT, BOTH) = ALL
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of load, frequency, or time for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

Describer	Meaning
	* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests that the power spectral density function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.
ATOC	Requests that the autocorrelation function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.
CRMS	Requests that the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.
RALL	Request that all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 6.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Requests printing of output coordinate system ID in printed output file, (.f06).
TM	Translational magnitude filter. See Remark 8.
T1, T2, T3	Translational component filters. See Remark 8.
RM	Rotational magnitude filters. See Remark 8.
R1, R2, R3	Rotational component filters. See Remark 8.
F	Filter value (Real > 0.0). See Remark 8.

Describer	Meaning
CONNECTOR	<p>A set of CWELD or CFAST, elements are defined from which auxiliary grids will be determined for output post-processing for displacement display in the basic system. This command produces the following actions:</p> <p>The auxiliary point “grids” determined by the set m of connector elements specified on this entry will be appended to the set n defined on the right side of the DISP command.</p> <p>m is the identification of a connector element set defined by a previously appearing SET command.</p> <p>If the right side of the DISP command is defined as NONE, then no points will be output even if the user has defined the keyword CONN=ALL or CONN=m.</p> <p>If the right side of the DISP command is defined as ALL, then auxiliary point “grids” for all connectors will be generated regardless if CONN= is specified.</p> <p>If the user wishes to produce displacements for all the connector elements and have them appended to the set n defined on the right side of the DISP command, then to avoid having to define a previously appearing m SET command with all connectors listed, the user may define CONN=ALL.</p>
ALL	Displacements for all points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	Displacement for no points will be output.
STRUCTURE, FLUID, BOTH	Request eigenvector output data type for normal mode computation (SOL 103) with METHOD(COUPLED) case control. Structure displacement only, Fluid pressure only or both Structure displacement and Fluid pressure output separately. Default is NONE for both structure displacement and fluid pressure together. See Remark 10.

**Remarks:**

1. The defaults for SORT1 and SORT2 depend on the type of analysis:
  - a. SORT1 is the default in static analysis, frequency response, steady state heat transfer analysis, real and complex eigenvalue analysis, flutter analysis, and buckling analysis.
  - b. SORT2 is the default in transient response analysis (structural and heat transfer). SORT2 is not available for real eigenvalue (including buckling), complex eigenvalue, or flutter analysis. If SORT1 is selected in a transient solution for one or more of the commands ACCE, DISP, ENTH, FORC, HDOT, MPCF, OLOA, SPCF, STRA, STRE, and VELO, then the remaining

commands will also be output in SORT1 format. If SORT2 is selected in a static or frequency response solution for one or more of the commands ACCE, DISP, FORC, MPCF, OLOA, SPCF, STRA, STRE, and VELO, then the remaining commands will also be output in SORT2 format.

- c. XY plot requests forces SORT2 and overrides SORT1 requests!
  - d. If a RANDOM request occurs the output will be in SORT2 format, even if SORT1 is requested.
2. VECTOR and PRESSURE are alternate forms and are equivalent to DISPLACEMENT. In complex analysis, the pressure is ALWAYS magnitude-phase.
  3. DISPLACEMENT=NONE overrides an overall output request.
  4. The units of translation are the same as the units of length of the model. Rotations are in units of radians.
  5. Displacement results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
  6. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
  7. Note that the CID keyword affects only grid point related output, such as DISPlacement, VELOcity, ACCEleration, OLOAD, SPCForce and MPCForce. In addition, CID keyword needs to appear only once in a grid-related output request anywhere in the Case Control Section to turn on the printing algorithm.
  8. Displacement components may be selected to control filtering to reduce the amount of output produced. When magnitudes are selected, the component values are ignored. Only a single positive value for  $f$  can be supplied, and comparisons are performed in the global reference frame. Comparisons are performed after the SET intersection is performed against the domain. Selection of this option does not effect the MAXMIN(GRID) operations. Scalar comparisons are performed using the minimum of all supplied values for the filters.

For complex vectors encountered in frequency response analysis, the magnitudes TM and RM follow a derivation using a deterministic interpretation and are calculated as follows:

- For Grid Points

TM

Define

$$C1 = T1real^{**2} + T2real^{**2} + T3real^{**2}$$

$$C2 = T1imag^{**2} + T2imag^{**2} + T3imag^{**2}$$

$$C3 = T1real*T1imag + T2real*T2imag + T3real*T3imag$$

$$C4 = (C1 + C2)/2$$

$$C5 = (C1 - C2)/2$$

Then,

$$TM = C4 + SQRT (C5^{**2} + C3^{**2})$$

RM

The calculations are similar to the above, except that R1, R2 and R3 are used in place of T1, T2 and T3, respectively.

- For Scalar Points

In this case, TM and RM have the same meaning.

Define

$$C1 = T1real**2$$

$$C2 = T1imag**2$$

$$C3 = T1real*T1imag$$

$$C4 = (C1 + C2)/2$$

$$C5 = (C1 - C2)/2$$

Then,

$$TM \text{ (or RM)} = C4 + \text{SQRT} (C5**2 + C3**2)$$

9. When using filters, the compound usage of the verbs PRINT, PLOT is allowed. The entries in the printed output are the entries that exceed any threshold, while the remaining entries within the SET are marked as plot to allow for postprocessing operations. When SORT2 is selected, then PRINT, PLOT must be used to allow for table transpose operations to occur. When any entry in the SORT2 format is above the threshold, all values for time or frequency will be printed for the grid point.
10. Default eigenvector output format of coupled mode computation (SOL 103) is structure displacement and fluid pressure combined in one datablock per same modes DISP(PRINT) = ALL.

The eigenvector table has normal title in f06 file.

```
REAL EIGENVECTOR NO. 1
```

To get structure displacement and fluid pressure separately from coupled mode computation (with METHOD(coupled) in SOL 103), DISP(PRINT,STRUCTURE) = ALL is used for STRUCTURAL displacement only. The eigenvector table has title (STRUCTURE) in f06 file.

```
REAL EIGENVECTOR(STRUCTURE) NO. 1
```

DISP(PRINT,FLUID) = ALL is used for FLUID pressure only. The eigenvector table has the title (FLUID) in f06 file.

```
REAL EIGENVECTOR(FLUID) NO. 1
```

DISP(PRINT,BOTH) = ALL is used for both STRUCTURAL displacement and FLUID pressure but separately. Each eigenvector table includes its own title (STRUCTURE) or (FLUID) in f06 file.

```
REAL EIGENVECTOR(STRUCTURE) NO. 1
```

```
.....
```

```
REAL EIGENVECTOR(FLUID) NO. 1
```

```
.....
```



**DIVERG**

## Static Aeroelastic Divergence Request

Selects the divergence parameters in a static aeroelastic divergence problem.

**Format:**

DIVERG=*n*

**Example:**

DIVERG=70

Describer	Meaning
<i>n</i>	Set identification of a DIVERG Bulk Data entry (Integer > 0).

**Remark:**

1. Static aeroelastic divergence analysis can be performed only in SOLs 144, 200 and 400.

**DLOAD**

## Dynamic Load Set Selection

Selects a dynamic load or an acoustic source to be applied in a transient or frequency response problem.

**Format:**

DLOAD=n

**Example:**

DLOAD=7 3

Descriptor	Meaning
n	Set identification of a DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE Bulk Data entry (Integer > 0).

**Remarks:**

1. RLOAD1 and RLOAD2 may only be selected in a frequency response problem.
2. TLOAD1 and TLOAD2 may be selected in a transient or frequency response problem.
3. Either an RLOADi or TLOADi entry (but not both) must be selected in an aeroelastic response problem. If RLOADi is selected, a frequency response is calculated. If TLOADi is selected, the transient response is computed by Fourier transform. When there are only gust loads (GUST Bulk Data entry), the DLOAD selects a TLOADi or RLOADi Bulk Data entry with zero load, along with field 3 of the GUST command.
4. The DLOAD command will be ignored if specified for upstream superelements in dynamic analysis. To apply loads to upstream superelements, refer to the LOADSET Case Control command.
5. For nonlinear dynamic analysis with SOL 400, TEMPERATURE load can't be applied by DLOAD/TEMP/TLOADi. TEMP (case control) and TTEMP (Bulk data) should be used to apply the temperature load. However, for pure linear analysis, DLOAD/TEMP has to be used for temperature load.

**DRSPAN**

## Response Spanning Set Selection

Selects a set of DRESP1 entries for the current subcase that are to be used in a DRESP2 or DRESP3 response that spans subcases.

**Format:**

DRSPAN=*n*

**Example:**

DRSPAN=1 0

Describer	Meaning
<i>n</i>	Set identification of a previously appearing SET command (Integer > 0).

**Remarks:**

1. In SOL 200, DRESP2 or DRESP3 Bulk Data entries can invoke DRESP1 responses that span subcases if these DRESP1 responses have been identified using a DRSPAN Case Control command that references a SET request that identifies the DRESP1 Bulk Data entries.
2. Each DRESP1 identified must produce a scalar value.
3. The DRSPAN Case Control command must be at the subcase level, whereas the SET request can be specified above the subcase level.
4. DRESP2, or DRESP3 that SPANS subcases, cannot reference another DRESP2 and/or DRESP3.
5. DRSPAN must appear in every subcase in the file if the synthetic response is to span the subcases. The synthetic response via DRESP2 or DRESP3 must reference all DRESP1 IDs defined in SETs of DRSPAN. In DEQATN, all DRESP1 IDs should show up in the list of variables. For those DRESP1 IDs that are not needed, it can be dropped from the definition of equation(s).
6. To check the value of spanned response, set parameter P2 of DOPTPRM to a value equal to or larger than 8, see DOPTPRM for details. Note that verification requires performing hand calculation based on user-provided DEQATN for the spanned response.

## DSAPRT

## Design Sensitivity Output Parameters

Specifies design sensitivity output parameters.

### Format:

$$DSAPRT \left[ \begin{array}{c} \text{FORMATTED} \\ \text{UNFORMATTED} \\ \text{NOPRINT} \end{array} \right], \left[ \begin{array}{c} \text{NOEXPORT} \\ \text{EXPORT} \end{array} \right], \text{START} = i, \text{BY} = j, \text{END} = k = \left[ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right]$$

### Examples:

```
DSAPRT (FORMATTED, EXPORT)
DSAPRT (FORMATTED, START=FIRST, BY=3, END=LAST) =1 0 1
DSAPRT (UNFORMATTED, START=FIRST)
DSAPRT (UNFORMATTED, EXPORT)
DSAPRT (FORMATTED, END=4) =ALL
DSAPRT (UNFORMATTED, END=SENS) =ALL
DSAPRT (NOPRINT, EXPORT)
```

Descriptor	Meaning
FORMATTED	Output will be presented with headings and labels.
UNFORMATTED	Output will be printed as a matrix print (see description of the MATPRN module in the <a href="#">MSC Nastran DMAP Programmer's Guide</a> ).
NOPRINT	No output will be printed.
EXPORT	Output will be exported to an external binary file specified by PARAM,IUNIT.
NOEXPORT	Output will not be exported to an external binary file.
START=i	Specifies the first design cycle for output (Integer > 0 or Character: "FIRST" or "LAST"; Default = 1 or "FIRST").
BY=j	Specifies the design cycle interval for output. See Remark 2. (Integer ≥ 0; Default = 0).
END=k	Specifies the last design cycle for output. (Integer > 0 or Character: "FIRST", "LAST", or "SENS"; Default = "LAST")
ALL	All retained design responses (defined on DRESP1, DRESP2 and DRESP3 entries) will be output.
n	Set identification of a previously appearing SET command. Only sensitivities of retained responses with identification numbers that appear on this SET command will be output (Integer > 0).

**Remarks:**

1. Only one DSAPRT may appear in the Case Control Section and it must occur with or above the first SUBCASE command.
2. Sensitivity data will be output at design cycles  $i$ ,  $i+j$ ,  $i+2j$ , ...,  $k$ . Note that the  $BY=0$  implies no sensitivity results will be output at the intermediate design cycles.
3.  $END=SENS$  requests design sensitivity analysis, and no optimization will be performed.
4. If both DSAPRT and  $PARAM,OPTEXTIT, 4, -4$ , or  $7$  are specified, then DSAPRT overrides  $PARAM,OPTEXTIT, 4, -4$ , or  $7$ .  $PARAM,OPTEXTIT$  values and the equivalent DSAPRT commands are described as follows:

OPTEXTIT	Equivalent DSAPRT Command
4	DSAPRT(UNFORMATTED, END=SENS)
-4	DSAPRT(NOPRINT, EXPORT, END=SENS)
7	DSAPRT(UNFORMATTED, START=LAST)

5. The  $n$  and NONE options are not supported for UNFORMATTED output. Only the UNFORMATTED option is supported for EXPORT.
6.  $PARAM,DSZERO$  can be used to set a threshold for the absolute value of the formatted sensitivity prints.
7. Design Sensitivity analysis is never performed following a discrete design optimization. Therefore, no sensitivity output will be produced with  $DSAPRT(END=LAST)$  when discrete optimization is performed at the end of a job.
8. Formatted sensitivity data can also be written into Comma Separated Values(or CSV) file with following steps
  - a.  $DSAPRT(formatted,..)$  request in case control. Note that 'formatted' is a default option.
  - b.  $PARAM,XYUNIT,52$  in bulk data. Unit 52 is simply chosen as an example.
  - c. file assignment statement, such as  
`ASSIGN USERFILE='jobname.csv' FORM=formatted STATUS=new UNIT=52`

## DSYM

## Dihedral Symmetry Option in Cyclic Symmetry

Provides for either one or two planes of overall symmetry in DIH-type cyclic symmetry problems.

### Format:

$$DSYM = \left\{ \begin{array}{c} S \\ A \\ SS \\ SA \\ AS \\ AA \end{array} \right\}$$

### Example:

DSYM=AS

Describer	Meaning
S, A	The problem is assumed to be symmetrical (S) or antisymmetrical (A) with respect to the plane containing side 1 of segment 1.
SS, SA, AS, AA	The problem is assumed to be symmetrical (or antisymmetrical) with respect to the plane containing side 1 of segment 1 (denoted by the first symbol), and also with respect to a plane perpendicular to side 1 (denoted by the second symbol).

**DYSTIFF**

## Dynamic Stiffness Output Request

Requests the form and type of dynamic stiffness output in SOL 108, 111, 200 and SOL 400 only.

$$DYSTIFF \left[ \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT,PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] \right) \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

**Examples:**

DYSTIFF = ALL

DYSTIFF (REAL, PUNCH, PRINT)=17

DYSTIFF = 25

DYSTIFF (SORT2, PRINT)=20

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each frequency.
SORT2	Output will be presented as a tabular listing of frequency for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Dynamic Stiffness at all points will be output. See remark 2.
n	Set identification of a previously appearing SET command. Only dynamic stiffness of points with identification numbers that appear on this SET command will be output. (Integer >0).
NONE	No dynamic Stiffness will be output (Default).

### Remarks:

1. DYSTIFF is available for SOL 108, 111 and SOL 200 and 400 with ANALYSIS=DFREQ and MFREQ. Note that DYSTIFF is available for output only and cannot be utilized as the design response in SOL 200.
2. See Remark 1. under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. DYSTIFF=NONE suppresses the generation of dynamic stiffness output.
4. Dynamic Stiffness,  $d_i$  at a grid point is computed as,  $d_i=f_i/u_i$  where  $f_i$  is the load applied and  $u_i$  is the complex displacement. The dynamic stiffness will be output for the grids having load applied on it.
5. Refrain from using ALL which may produce voluminous output.
6. The dynamic stiffness output will be subjected to the mechanical load only.
7. Dynamic stiffness output is not supported for enforced motion analysis. The results will be zero for enforced motion analysis.



## ECHO Bulk Data Echo Request

Controls echo (i.e., printout) of the Bulk Data.

### Format:

$$\text{ECHO} = \left\{ \begin{array}{l} \text{[SORT]([EXCEPT]cdni,...)} \\ \text{UNSORT} \\ \text{BOTH} \\ \text{NONE} \end{array} \right\}, \text{PUNCH} \left[ \begin{array}{l} \text{SORT} \\ \text{BOTH} \\ \text{NEWBULK} \end{array} \right], \text{FILE} \left\{ \right.$$

### Examples:

```
ECHO=UNSORT
ECHO=BOTH
ECHO=PUNCH, SORT (MAT1, PARAM)
ECHO=SORT (EXCEPT DMI, DMIG)
ECHO=BOTH, PUNCH, FILE
```

(In the above examples See Remark 7.: the PUNCH keyword will always result in the PUNCH file containing the entire bulk data input. Explicit inclusion indicators (CDNI) such as (MAT1, PARAM) will always cause a F06 file to be written that contains only the (CDNI) data.)

Describer	Meaning
SORT	The sorted (arranged in alphanumeric order) Bulk Data will be printed.
EXCEPT	Excludes cdni Bulk Data entries from sorted echo printout. See Remark 6.
cdni,...	Defines Bulk Data entry names to be included, or excluded by EXCEPT, in the sorted echo printout. The PUNCH (.pch) file is not affected by cdni.
UNSORT	The unsorted Bulk Data will be printed. If SORT is also not specified, the sorted Bulk Data will not be printed.
BOTH	Both sorted and unsorted Bulk Data will be printed. This is equivalent to ECHO=SORT, UNSORT.
NONE	Neither sorted nor unsorted Bulk Data will be printed.
PUNCH	The entire Bulk Data will be written to the punch (.pch) file. See Remark 7. for options.
FILE	The entire Bulk Data echo will be written to the separate file with a default suffix of .BECHO in *.f06 form. A user-defined filename must be specified in the ASSIGN statement.
NEWBULK	In SOL 200, a complete unsorted Bulk Data file is written to the PUNCH file with updated design model entries.

### Remarks:

1. If no ECHO command appears, sorted Bulk Data will be printed.
2. Comments will appear at the front of the sorted file if ECHO=PUNCH.
3. Portions of the unsorted Bulk Data can be selectively echoed by including the commands ECHOON and ECHOOFF at various places within the Bulk Data. ECHOOFF stops the unsorted echo until an ECHOON commands is encountered. Many such pairs of commands may be used. The ECHOON and ECHOOFF command may be used in the Executive and Case Control Sections; however, ECHOOFF should not be the first entry and continuation entries are not handled correctly.
4. If the SORT (cdni,...) describer is specified in a restart in SOLs 101 through 200, then the continuation entries will not be printed.
5. If the SORT (cdni,...) describer is used, then it must appear as the last describer, as in the preceding example.
6. If the EXCEPT describer is specified, then it must be specified before all cdni. All Bulk Data entry types will be listed except those given for cdn1, cdn2, etc. If EXCEPT is not specified, then only those Bulk Data entry types listed under cdn1, cdn2, etc. will be listed.
7. PUNCH without options or PUNCH(SORT) produces a sorted listing of the entire bulk data input in the punch file. PUNCH(NEWBULK) produces unsorted bulk data input with updated design model entries in the punch file. PUNCH(BOTH) combines these two outputs.

**EDE****Element Energy Loss Per Cycle Output Request**

Requests the output of the energy loss per cycle in selected elements.

**Format:**

$$\text{EDE} \left[ \left( \begin{array}{c} \text{PRINT} \\ \text{PLOT} \end{array} \right), \begin{array}{c} \text{PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] \left[ \text{THRESH} = p \right] \left[ \text{NOPERCENT} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

EDE=ALL

EDE (PUNCH, THRESH=.0001) = 19

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

AVERAGE	Requests average energy (in frequency response analysis only) (Default).
AMPLITUDE	Requests amplitude of energy (in frequency response analysis only).
PEAK	Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Energies for elements having an energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).
NOPERCENT	Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will appear as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized.
ALL	Energy for all elements will be computed.

Describer	Meaning
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EDE command, or above all subcases (Integer > 0).
NONE	Element energy loss will not be output.

### Remarks:

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by AVERAGE:

$$E_o = \pi\omega(\{u_r\}^T[B_e]\{u_r\} + \{u_i\}^T[B_e]\{u_i\})$$

AMPLITUDE:

$$E_a = \pi\omega\sqrt{(\{u_r\}^T[B_e]\{u_r\} - \{u_i\}^T[B_e]\{u_i\})^2 + (2\{u_r\}^T[B_e]\{u_i\})^2}$$

PEAK:

$$E_{peak} = E_o + E_a$$

where:

- E = elemental energy.
- $\{u_r\}$  = displacement (real part).
- $\{u_i\}$  = displacement (imaginary part).
- $[B_e]$  = elemental damping.

5. In SOLs 111 and 112, EDE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Only damping from the viscous dampers (e.g., CVISC, CDAMPi, etc.) are included. Structural damping is not included in the calculation.
7. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
8. If there are any frequency dependent CBUSH elements present, and PARAM, BUSHNM, NO; then all CBUSH elements will be excluded from the element energy loss output.

9. For transient response, EDE is computed in a quasi-static manner at each time step:  $\frac{1}{2} \{v\}^T [B_e] \{u\}$ ; where  $B_e$  is elemental damping and  $v$  and  $u$  are velocity and displacement respectively.

## EKE

## Element Kinetic Energy Output Request

Requests the output of the kinetic energy in selected elements.

### Format:

$$EKE \left( \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{l} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] \left[ \text{THRESH} = p \right] \left[ \text{NOPERCENT} \right] \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

### Examples:

```
EKE=ALL
EKE (PUNCH, THRESH=.0001) =19
```

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

AVERAGE	Requests average energy (in frequency response analysis only) (Default).
AMPLITUDE	Requests amplitude of energy (in frequency response analysis only).
PEAK	Requests peak energy (for frequency response analysis only). PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Kinetic energies for elements having a energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).
NOPERCENT	Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will appear as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized.
ALL	Kinetic energy for all elements will be computed.

Describer	Meaning
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the EKE command, or above all subcases (Integer > 0).
NONE	Kinetic energy values will not be output.

**Remarks:**

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy values may be computed in one of three ways as selected by

AVERAGE:

$$E_o = \frac{1}{4}(\{v_r\}^T [M_e] \{v_r\} + \{v_i\}^T [M_e] \{v_i\})$$

AMPLITUDE:

$$E_a = \frac{1}{4} \sqrt{(\{v_r\}^T [M_e] \{v_r\} - \{v_i\}^T [M_e] \{v_i\})^2 + (2\{v_r\}^T [M_e] \{v_i\})^2}$$

PEAK:

$$E_{peak} = E_o + E_a$$

where:

E = elemental energy.

{v<sub>r</sub>} = velocity (real part).

{v<sub>i</sub>} = velocity (imaginary part).

[M<sub>e</sub>] = elemental mass.

5. In SOLs 111 and 112, EKE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
7. EKE output request is available in SOL 400 multi-disciplinary and linear perturbation analyses using MODES and MCEIG types of analysis.

8. If there are any frequency dependent CBUSH elements present, and PARAM, BUSHNM, NO; then all CBUSH elements will be excluded from the element kinetic energy output.
9. For transient response, EKE is computed in a quasi-static manner at each time step:  $\frac{1}{2} \{v\}^T [M_e] \{v\}$ ; where  $M_e$  is elemental mass and  $v$  is velocity.



## ELAFORCES

### Elastic Forces Output Request

Requests the form and type of elastic forces output.

#### Format:

$$\text{ELAFORCES} \left( \left( \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right), \left( \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

#### Examples:

```
ELAFORCE=ALL
ELAF (REAL, PUNCH, PRINT)=17
ELAFORCE=25
ELAFORCE (SORT2, PRINT)=20
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.

PRINT or (blank)	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Elastic forces for all points will be output. See Remarks 2.

Describer	Meaning
NONE	No elastic force will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer >0).

**Remarks:**

1. Refrain from using ALL in transient analysis which may produce voluminous output.
2. See Remark 1. under [DISPLACEMENT \(Case\)](#), for a discussion of SORT1 and SORT2.
3. ELAFORCE=NONE suppresses the generation of elastic forces output.
4. Although ELAFOR selects the desired grid IDs, it is to be noted that the connections should be included in the computation. Hence,  $F_i$  of a point the requested set can be also expressed as

$$F_i = \sum_{j=1}^n K_{ij}u_j$$

5. ELAFORCE is available for SOL 108, 109, 111, 112 and SOL 200 with ANALYSIS=DFREQ, MFREQ and MTRAN. Note that ELAFORCE is available for output only and can not be utilized as design response.
6. ELAFORCE does not support external SE via OTM due to limitation.
7. ELAFORCE does not support DOMAINSOLVER with PARTOPT=grid.

## ELSDCON

## Element Stress Discontinuity Output Request

Requests mesh stress discontinuities based on element stresses.

### Format:

$$\text{ELSDCON} \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Examples:

ELSDCON=ALL  
ELSDCON=19

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Stress discontinuity requests for all SURFACE and VOLUME Case Control commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request (Integer >0).
NONE	No element stress discontinuity output.

### Remarks:

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME Case Control commands.

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME Case Control commands. Also, the GPSTRESS command must be present for printed output and the STRFIELD command for postprocessing output using the .xdb file (PARAM,POST,0) for the same surfaces and volumes.

## ELSENS

## Element Sensitivity Output for SOL 108/111

Select SOLUTION frequencies and RESPONSE DOFs for the generation element sensitivity.

### Format:

ELSENS  $\left( \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix}, \begin{bmatrix} \text{REAL or IMAG} \\ \text{PHASE} \end{bmatrix}, [\text{THRESH} = P], \text{RESPONSE} = r,$

$\left[ \text{SOLUTION} = \begin{Bmatrix} \text{ALL} \\ \text{self} \end{Bmatrix} \right],$

$\left[ \text{MASS, STIFF, DYNAMIC, SQSTIFF, SQDYNA} \right] = \begin{Bmatrix} \text{ALL} \\ n \\ \text{NONE} \end{Bmatrix}$

### Example

```
SET 81 = 100.0, 120.0
SET 91 = 11240/T3, 4001/T1
SET 96 = 15920 THRU 15950
$
ELSENS (RESPONSE=91, SOLUTION=81, MASS, STIFF, DYNAMIC) = 96
```

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

THRESH            The magnitude of element sensitivity less than p will be suppressed in all output files: print, punch, plot, .op2, and .xdb. (Default = 0.0).

RESPONSE        Adjoint load response will be computed for unit load applied at grid point components in SET r.

Describer	Meaning
SOLUTION	Frequency responses at these forcing frequencies, defined in setf, will be used for element sensitivity computation. (Default=all forcing frequencies)
MASS	Sensitivity with element mass matrices will be computed and output.
STIFF	Sensitivity with element stiffness matrices will be computed and output.
DYNAMIC	Sensitivity with element stiffness and mass matrices will be computed and output.
SQMASS	Squared sensitivity with element mass matrices will be computed and output.
SQSTIF	Squared sensitivity with element stiffness matrices will be computed and output.
SQDYNA	Squared sensitivity with element stiffness and mass matrices will be computed and output.
ALL	Sensitivities for all elements will be calculated.
n	Set identification number. Sensitivity for all elements specified on the SET n command will be calculated. The SET n command must be specified in the same subcase as the ELSENS command, or above all subcases ( Integer > 0 ). The IDs in set n must be EID (element ID).
NONE	Elemental sensitivity will not be output.

### Remarks:

1. Set r for RESPONSE has no default.
2. The equations various options of ELSENS

$$\text{ELSENS(MASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r]$$

$$\text{ELSENS(STIFF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSEND(DYNAMIC)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r]$$

$$\text{ELSENS(SQMASS)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [A_{\text{setf}}]^* [M_{\text{elem}}]^* [A_r]^*$$

$$\text{ELSENS(SQSTIF)} = [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + [U_{\text{setf}}]^* [K_{\text{elem}}]^* [U_r]^*$$

$$\text{ELSEND(DYNAMIC)} = [A_{\text{setf}}]^t [M_{\text{elem}}] [A_r] + [U_{\text{setf}}]^t [K_{\text{elem}}] [U_r] + [A_{\text{setf}}]^* [M_{\text{elem}}]^* [A_r]^* + [U_{\text{setf}}]^* [K_{\text{elem}}]^* [U_r]^*$$

where  $[U_{\text{setf}}]$  is the displacement of SOLUTION

$[A_{\text{setf}}]$  is the acceleration of SOLUTION

$[U_r]$  is the displacement of RESPONSE

$[A_r]$  is the acceleration of RESPONSE

$[K_{\text{elem}}]$  is element stiffness matrix

$[M_{\text{elem}}]$  is element mass matrix

superscript \* means complex conjugate of the term.

**ELSUM** Element Summary Output Request

Requests that a summary of element properties grouped by element type and/or element property type are to be printed.

**Format:**

$$\text{ELSUM}([\text{EID}, \text{PID}, \text{BOTH}, \text{EIDSUM}, \text{PIDSUM}, \text{NSMCONT}, \text{SUMMARY}]) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

**Examples:**

```
ELSUM = 9
ELSUM (PID) = ALL
```

Describer	Meaning
EID	Element summary output is grouped by element type.
PID	Element summary output is grouped by element property type.
BOTH	Both EID and PID groupings are produced.
EIDSUM	Only a summary of the mass totals for the EID grouping is produced.
PIDSUM	Only a summary of the mass totals for the PID grouping is produced.
NSMCONT	Nonstructural mass contributions from NSM, NSM1, NSML, and NSML1 Bulk Data entries are identified.
SUMMARY	Only a summary of the mass totals is produced.
ALL	Element summary output for all elements
n	Set identification of a previously appearing SET command. Produces output for only those elements whose identification numbers appear in the list of this SET command.
NONE	No element summary output is produced.

**Remarks:**

1. The ELSUM Case Control command produces a summary of properties for elements. The properties include element ID, material ID, length or thickness, area, volume, structural mass, nonstructural mass, total mass, and the product of total mass \* WTMASS. Total mass is the sum of the structural and nonstructural masses.



2. Certain element types produce only partial data. For these element types, no mass data is produced, and mass totals will not include any contributions from these element types. Mass data is computed for the following element types: CBAR, CBEAM, CBEND, CHEXA, CMASSi, CONM1, CONM2, CONROD, CPENTA, CQUAD4, CQUAD8, CQUADR, CRAC2D, CRAC3D, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, CTRIAX6, and CTUBE. The mass of elements with mid side nodes is approximated using only the geometry of the corner nodes. To get an accurate total mass, use the output from the Grid Point Weight Generator (PARAM,GRDPNT).
3. EIDSUM takes precedence over EID if both are present. Likewise, PIDSUM takes precedence over PID.
4. The ELSUM Case Control command is ignored in heat transfer solution sequences.
5. The NSMCONT descriptor produces various amounts of output depending upon whether the summary option is selected (SUMMARY, PIDSUM or EIDSUM requested). If NSMCONT is combined with PID, a table is produced that identifies the contribution of each NSM type Bulk Data entry to the total element nonstructural mass. If SUMMARY is included with PID and NSMCONT, an additional table is produced that identifies the mass contributions for each property type by property ID.
6. ELSUM output is only available for the PRINT option, not the PUNCH or PLOT options used in other commands.
7. ELSUM does not support advanced nonlinear elements.

## ENDSTEP

Specifies Final Analysis Step

Specifies final analysis step for SOL 700.

### Format:

ENDSTEP = Value

### Example:

ENDSTEP = 10000

Describer	Meaning
Value	Steps to end the simulation (Integer, Default = 9999999).

**ENDTIME**

Specifies Final Analysis Time

Specifies final analysis time for SOL 700.

**Format:**

ENDTIME = Value

**Example:**

ENDTIME = 0.01

Describer	Meaning
Value	Time in the applicable units for the model (usually seconds) (Real; Default = 1e+20).

## ENDMODULE

Module Bulk Data section delimiter

Indicates (delimits) the end of a Module's Bulk Data section.

### Format:

```
ENDMODULE
```

### Example:

```
ENDMODULE  
END MODULE
```

### Remarks:

1. A BEGIN MODULE command must appear somewhere before ENDMODULE.
2. The ENDDATA command may be used to terminate the last Module's Bulk Data section.

**ENTHALPY**

## Heat Transfer Enthalpy Output Request

Requests form of enthalpy vector output in transient heat transfer analysis (SOL 159).

**Format:**

$$\text{ENTHALPY} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

**Example:**

ENTHALPY=5

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Enthalpy for all grid points will be output.
n	Set identification of previously appearing SET command. Only enthalpies of grid points with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	Enthalpy for no grid points will be output.

**Remark:**

1. ENTHALPY=NONE is used to override a previous ENTHALPY = n or ENTHALPY = ALL command.

## EQUILIBRIUM

## Equilibrium Force Output Request

Specifies options for equilibrium force balance output of applied loads, single point constraint forces and forces due to multi-point constraints and rigid elements.

### Format:

$$\text{EQUILIBRIUM} \left[ \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{gid} \\ \text{NONE} \end{array} \right\}$$

### Examples:

```
EQUILIBRIUM
EQUILIBRIUM = 501
```

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

YES	Requests moment summation referenced to origin of basic coordinate system.
gid	Requests moment summation referenced to basic system location specified by the coordinates of grid point gid.
NONE	Equilibrium force balance output will not be generated.

### Remarks:

1. The EQUILIBRIUM Case Control command produces a summary of the applied loads, single point forces of constraint (SPC), and multipoint/rigid body element forces of constraint (MPC), as well as a summation of these quantities. In order for the summation to represent all of the forces in the problem, these forces must be available and, therefore, the specification of an EQUILIBRIUM Case Control command causes the program to automatically compute the SPC and MPC forces. However,

if desired, the associated Case Control commands should request output. The single point forces of constraint are requested by the presence of an SPCFORCE command, and the multipoint/RBE constraint forces are requested by an MPCFORCE command. Applied loads are automatically generated by the presence of the LOAD selection Case Control command.

2. Results are always output in the basic coordinate system.
3. The EQUILIBRIUM Case Control command is applicable to Linear Static analysis (SOL101) only, and does not produce output if any superelements are present.

## ERP

### Equivalent Radiated Power Panel Participation Factor Output Request

Requests the form and type of ERP panel participation factor output for frequency or transient analysis.

#### Format:

$$\begin{aligned}
 & \text{ERP} \left[ \begin{array}{l} \text{[SORT2]} \\ \text{[SORT1]} \end{array} \right], \left[ \begin{array}{l} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{array} \right] \left[ \text{SOLUTION} = \begin{array}{l} \text{ALL} \\ \text{setf} \\ \text{sett} \end{array} \right], \\
 & \left[ \text{MPF} = \begin{array}{l} \text{ALL} \\ \text{nlm} \\ \text{NONE} \end{array} \right], [\text{MPFSORT} = \text{sorttype}], [\text{ELEMENT}], \\
 & \left[ \text{KEY} = \begin{array}{l} \text{frequency} \\ \text{fraction} \\ \text{time} \end{array} \right], \left[ \text{FILTER} = \begin{array}{l} \text{0.01} \\ \text{real\_value} \end{array} \right], \\
 & \left[ \text{ERPRHO} = \begin{array}{l} \text{1.0} \\ \text{real\_value} \end{array} \right], \left[ \text{ERPC} = \begin{array}{l} \text{1.0} \\ \text{real\_value} \end{array} \right] \\
 & \left[ \text{RHOC} = \begin{array}{l} \text{1.0} \\ \text{real\_value} \end{array} \right], \left[ \text{ERPRLF} = \begin{array}{l} \text{1.0} \\ \text{real\_value} \end{array} \right] \\
 & \left[ \text{ERPREFDB} = \begin{array}{l} \text{1.0} \\ \text{real\_value} \end{array} \right], [\text{CSV} = \text{unit}] = \begin{array}{l} \text{ALL} \\ \text{setp} \\ \text{NONE} \end{array}
 \end{aligned}$$

#### Examples:

```

SET 17 = 10.,20.,30.,40.,80.,100. $ A list of frequencies
SET 25 = ROOF, DOORLF $ A list of ERP Panel names
                    $ from a ERPPNL Bulk Entry
ERP ( PRINT,PUNCH,SOLUTION=17,KEY=frac ) = 25
    
```

Describer	Meaning
SORT1	Output is presented as a tabular listing of ERP panels for each frequency.
SORT2	Output is presented as a tabular listing of frequency for each ERP panel.



Describer	Meaning
-----------	---------

PRINT or (blank)	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PUNCH	X		X*
PLOT		X	X*
			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

SOLUTION	Keyword to select frequencies or time.
ALL	If associated with SOLUTION, all frequencies or times are selected. If associated with <i>setp</i> , all ERPPNL entries are selected.
setf	Identifier of Case Control SET command defining frequencies.
sett	Identifier of Case Control SET command defining times.
MPF	Requests the output of Modal Participation Factors for ERP. See Remark 9.
ALL	All structural modes for which MPF will be computed
nIm	Number of lowest structural modes for which MPF will be computed
NONE	MPF will not be computed. (Default)
MPFSORT	Keyword selecting the sort type. Default is sorting by increasing natural mode number. See Remark 10.
Sorttype	Sort options: ABSA output will be sorted by absolute value in ascending order. ABSD output will be sorted by absolute value in descending order. ALGA output will be sorted by algebraic value in ascending order. ALGD output will be sorted by algebraic value in descending order
ELEMENT	Keyword to request element-by-element ERP output
KEY	Keyword selecting the output item used to sort the printed output. The default produces output sorted on either frequency (SORT2) or ERP panel name (SORT1).  KEY= <i>fraction</i> produces output sorted in descending order of the fractional ERP value of maximum ERP across all frequencies or time steps of a panel.  KEY= <i>time</i> produces output sorted on either time (SORT2) or ERP panel name (SORT1). In transient the default KEY=frequency will automatically become KEY=time.

Describer	Meaning
FILTER	Keyword specifying the value of a filter to be applied to the printed output only. ERP values are printed only if the fractional ERP value of maximum ERP across all frequencies or time steps of a panel exceeds the filter value.
ERPRHO	Fluid density for Equivalent Radiated Power (ERP) analysis. This item is actually an MSC Nastran parameter.
ERPC	Phase speed of the fluid for Equivalent Radiated Power (ERP) analysis. This item is actually an Nastran parameter.
ERPRLF	Radiation loss factor. In frequency the scale factor $C = \text{ERPRLF} * (\frac{1}{2}\text{ERPRHO} * \text{ERPC})$ . In transient the scale factor $C = \text{ERPRLF} * (\text{ERPRHO} * \text{ERPC})$ .
RHOCP	Scale factor used in dB computation. This item is actually an MSC Nastran parameter.
ERPREFDB	Scale factor used in dB computation. This item is actually an MSC Nastran parameter.  The dB calculation is $\text{ERPdB} = 10 \log \left( \text{RHOCP} \cdot \frac{\text{ERP}}{\text{ERPREFDB}} \right)$ .
CSV	Results will be written to a .csv file.  unit      Unit of the .csv file as used on the required ASSIGN statement.  setp      Identifier of case control SET command defining NAMEi entries from an ERPPNL Bulk Data entry defining panels.  NONE      No ERP output is produced.

### Remarks:

- ERP is required to produce any ERP output and ERP request must appear either above SUBCASE level or within 1st SUBCASE. Subsequent SUBCASEs, from 2nd onwards, may or may not have ERP request.
- Output is generated in SORT2 by default. Unlike other Case Control requesting SORT2 format, the ERP command does not force all other output into SORT2 format.
- FILTER has no effect on PUNCHED, CSV or OP2 output.
- In addition to individual panel output a summary named ALLPANEL is produced. If there are multiple subcases, the panel name is formed from the serial subcase number (1-nsubc) and the characters 'ALLP' as in ALLP0002 unless the ERP command request output for ALL panels across the Subcases. In this case, the summary panel name ALLPANEL is retained.
- Selectable frequencies are dependent on the presence of an OFREQ Case Control command.
- ERPRHO, ERPC, ERPRLF, RHOCP, and ERPREFDB are actually PARAM,name,value entries.
- The filter process avoids printing ERP for cases where ERP/ERPMAX is less than the FILTER value. ERPMAX is the maximum ERP value across all frequencies for a panel.
- If output to a .csv file is requested, the file must be assigned with logical key USERFILE and FORM=FORMATTED, e.g.,

ASSIGN USERFILE = myfile.csv UNIT=50 FORM=FORMATTED STATUS=NEW

9. The MPF of ERP will be computed with the same parameters, such as SOLUTION, csv, setp and etc.
10. MPFs for ERP are sorted by increasing order of natural frequencies unless the MPFSORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. Since ERP is always positive, ABSA/ALGA and ABSD/ALGD will produce same sorting results.
11. Element ERP includes elements defined via ERPPNL and SET3. For 3D element types, such as HEXA, PENTA and TETRA, element ERP includes those elements at the outer surface of the model. For an element with multiple faces exposed, the element ERP is the summation of ERP of all exposed faces.
12. MPF for ERP and element ERP may increase the volume of output significantly.

## ESE Element Strain Energy Output Request

Requests the output of the strain energy in selected elements.

### Format:

$$\text{ESE} \left[ \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \left[ \begin{array}{l} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right] [\text{THRESH} = p][\text{NOPERCENT}] \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \end{array} \right\}$$

### Examples:

```
ESE=ALL
ESE (PUNCH, THRESH=.0001)=19
```

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

AVERAGE	Requests average energy in frequency response analysis only.
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
THRESH	Energies for elements having an energy value of less than p% will be suppressed in all output files: print, punch, plot, .op2, and .xdb. THRESH overrides the value of TINY described in Remark 1. (Default = 0.001).
NOPERCENT	Do not compute TOTAL ENERGY OF ALL ELEMENTS IN PROBLEM and PERCENT OF TOTAL. The values will appear as zeros. Sparse data recovery and NOPERCENT can result in significant reduction in computing resources. But if RTYPE=TOTSE appears on a DRESP1 entry then full data recovery is necessary and no savings will be realized.
ALL	Energy values for all elements will be computed.

Describer	Meaning
n	Set identification number. Energy for all elements specified on the SET n command will be computed. The SET n command must be specified in the same subcase as the ESE command, or above all subcases (Integer > 0).
NONE	Element strain energy will not be output.

**Remarks:**

1. If THRESH = p is not specified, then p defaults to the values specified by user parameter TINY.
2. The energy calculations include the contribution of initial thermal strain.
3. Energy density (element strain energy divided by element volume) is also computed in some solution sequences. It can be suppressed by use of PARAM,EST,-1.
4. For frequency response analysis, the energy may be computed in one of three ways as selected by AVERAGE:

$$E_o = \frac{1}{4}(\{u_r\}^T [K_e] \{u_r\} + \{u_i\}^T [K_e] \{u_i\})$$

AMPLITUDE:

$$E_a = \frac{1}{4} \sqrt{(\{u_r\}^T [K_e] \{u_r\} - \{u_i\}^T [K_e] \{u_i\})^2 + (2\{u_r\}^T [K_e] \{u_i\})^2}$$

PEAK:

$$E_{peak} = E_o + E_a$$

where:

- E = elemental energy.
- $\{u_r\}$  = displacement (real part).
- $\{u_i\}$  = displacement (imaginary part).
- $[K_e]$  = elemental stiffness.

5. In SOLs 111 and 112, ESE is not available if both PARAM,DDRMM,0 and PARAM,SPARSEDR,NO are specified.
6. Element data recovery for thermal loads is not currently implemented in dynamics.

7. Element strain energy is available for nonlinear static analysis (SOL 106). However, in a normal modes analysis in SOL 106 with PARAM,NMLOOP (or ANALYSIS=MODES) or a PARAM NMLOOP restart into SOL 103, energies are computed for elements with linear properties only. ESE output request is also available in SOL 400 linear and nonlinear static analyses as well as multi-disciplinary and linear perturbation analyses using MODES and MCEIG types of analysis. All other nonlinear solution sequences do not support element strain energy output.
8. The strain energy for nonlinear elements is calculated by integrating the specific energy rate, the inner product of strain rate, and stress over element volume and time.

$$E = \int_0^t \int_V \dot{\varepsilon}^T \sigma dV dt \quad (5-10)$$

where:

- $\sigma$  = stress tensor
- $\dot{\varepsilon}$  = tensor of the strain rate
- $V$  = element volume
- $t$  = actual time in the load history

Loads from temperature changes are included in Eq. (5-10). If we assume a linear variation of temperatures from subcase to subcase, then the strain energy in Eq. (5-10) for the special case of linear material and geometry becomes

$$E = \frac{1}{2} u^T K_e u - \frac{1}{2} u^T P_{et} \quad (5-11)$$

where  $P_{et}$  is the element load vector for temperature differences.

For linear elements, the default definition of element strain energy is

$$E = \frac{1}{2} u^T K_e u - u^T P_{et} \quad (5-12)$$

where  $P_{et}$  is the element load vector for temperature differences and element deformation.

In Eq. (5-12), the temperatures are assumed to be constant within a subcase. The default definition of the strain energy for linear elements differs from the definition for nonlinear elements by a factor of 1/2 in the temperature loads. To request the strain energy for linear elements using Eq. (5-11), set the parameter XFLAG to 2; the default value for XFLAG is 0, which uses Eq. (5-12) for the strain energy of linear elements.

9. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.

10. If there are any frequency dependent CBUSH elements present, and PARAM, BUSHNM, NO; then all CBUSH elements will be excluded from the element strain energy output.
11. For transient response, ESE is computed in a quasi-static manner at each time step:  $\frac{1}{2} u^T K_e u$ ; where  $K_e$  is elemental stiffness and u is displacement.

## EXPORTLD

Saves a Load Vector on the Database for Subsequent Reuse

Saves the load vector for the current subcase on the database.

### Format:

$$\text{EXPORTLD}([\text{LOADID} = \text{lid}][\text{LOADNAME} = \text{ldname}]) = \left\{ \begin{array}{c} n \\ \text{ALL} \end{array} \right\}$$

### Examples:

```
EXPO (LOADNAME=LANDGEAR)
EXPORTLD=10
```

Describer	Meaning
LOADID=lid	User assignable value for the LOADID describer. Used to uniquely identify a saved load vector for later retrieval (using DBLOCATE, for example). The default value is the subcase ID.
LOADNAME=ldname	User-assignable value (of up to eight characters) for the LOADNAME describer. Used to uniquely identify a saved load vector for later retrieval. The default value is blank.
N	Results for grid point components in SET n will be exported.
ALL	Results for ALL grid point components will be exported. Default is ALL.

### Remarks:

1. Each load vector is stored individually as a one column matrix that is qualified by LOADID and LOADNAME. The combination of both the LOADID and the LOADNAME describers should uniquely identify the load vector to avoid overwriting (or possibly triggering the output twice rule an existing load vector).
2. If placed above the SUBCASE entry, then the load vectors for all subcases are saved.
3. The load vectors are qualified by LOADID and LOADNAME for selection using the WHERE clause on FMS commands that support it.
4. The following table shows the value of the LOADID and LOADNAME describers that are assigned for various EXPORTLD requests:



Example	Results
EXPORTLD	LOADID=subcase ID, LOADNAME=' ' - default value applied.
EXPORTLD(LOADID=12)	LOADID=12, LOADNAME=' '
EXPORTLD(LOADID=1,LOADNAME=FORCE12)	LOADID=1, LOADNAME='FORCE12'
EXPORTLD(LOADNAME=ALLCASES)	LOADID=subcase ID, LOADNAME='ALLCASES'

5. The load vector is typically imported into a run using the FMS DBLOCATE statement. The imported load is referenced by using its LOADID value on a LOAD Case Control command or Bulk Data entry. For example:

```

ASSIGN loads1='run1.MASTER'
DBLOCATE datablk=(EXTLD) WHERE(LOADNAME='ALLCASES'),
CONVERT (LOADID=LOADID+1000) LOGICAL=loads1
...
CEND
LOADS=1001 $ Select external load with LOADID=1001, imported from
previous run.

```

## EXTDRIN

### External Superelement Data Recovery Restart Run Request

Requests the job to perform an external superelement data recovery restart. Also specifies the storage media of the boundary solution data.

**Format:**

```
EXTDRIN (
    DMIGOP2 [= unitop2]
    MATRIXOP4 or MATOP4 [= unitop4]
    MATRIX or MATDB
    DMIGDB
)
```

**Examples:**

```
EXTDRIN
EXTDRIN (DMIGOP2=45)
EXTDRIN (MATOP4)
```

Descriptor	Meaning
DMIGOP2 = <i>unitop2</i>	Retrieve the boundary solution data stored in DMIG Bulk Data internal format on an .op2 file whose Fortran unit number is given by unitop2 (Integer > 0, Default=31).
MATRIXOP4 or MATOP4 = <i>unitop4</i>	Retrieve the boundary solution data on an .op4 file whose Fortran unit number is given by unitop4. (Integer > 0, Default=31).
MATRIXDB or MATDB	Retrieve the boundary solution data stored in the standard matrix format from the database.
DMIGDB	Retrieve the boundary solution data stored in the DMIG Bulk Data internal format from the database.

**Remarks:**

- EXTDRIN is intended for step three in external superelement analysis. External superelement data recovery is accomplished by restarting from the data base created in step one (external superelement creation run) and attaching the boundary solution data from step two (assembly run). The boundary solution data must have been stored in step two via the EXTDROUT Case Control command or user parameter EXTDROUT.
- External superelement data recovery restarts are limited to SOLs 101, 103, 107 through 112, and 400.
- EXTDRIN must be specified above all subcases.
- User parameters EXTDR and EXTDRUNT are the old method for requesting an external superelement data recovery restart. EXTDRIN Case Control command takes precedence over the EXTDR and EXTDRUNT user parameters.
- If data recovery is desired for the external component in SOLs 101, 103, 107 through 112 and 400, there are four methods to transmit the displacements of the reduced model, in Step 2, to the external full model. The method is selected by the EXTDROUT Case Control command in Step 2. The options are as follows:

- a. `EXTDROUT(MATRIXDB[=dbext])`. The displacements of the reduced component model are stored directly on the database. The sequencing of the displacement degrees-of-freedom corresponds to the sequencing in the reduced model. Keyword `dbext` exports the boundary solution data to a new DBset initialized by the “INIT dbext” statement in the File Management section of the input.
- b. `EXTDROUT(DMIGDB[=dbext])`. The displacements of the reduced model are stored on the database in a format which allows automatic connection to the reduced component model if the reduction grid points and scalar points are the same grid points and scalar points used in the analysis model. Keyword `dbext` exports the boundary solution data to a new DBset initialized by the “INIT dbext” statement in the File Management section of the input.
- c. `EXTDROUT(DMIGOP2=unit)`. The same as `EXTDROUT(DMIGDB)` except that the displacements of the reduced model are written in `OUTPUT2` format to a tape unit specified by the unit keyword (Default=31). The output unit can be assigned to a specific file by using an `ASSIGN OUTPUT2` command in the File Management Section.
- d. `EXTDROUT(MATOP4=unit)`. The same as `EXTDROUT(MATRIXDB)` except that the displacements of the reduced model are written in `OUTPUT4` format to a tape unit specified by the unit keyword (Default=31). The output unit can be assigned to a specific file by using an `ASSIGN OUTPUT4` command in the File Management Section.

Data recovery restart for the external superelement (Step 3) is available in SOLs 101, 103, 107 through 112, and 400 and is accomplished by using a restart procedure from the data base created in Step 1 and specifying the `EXTDRIN` Case Control command to import the solutions from Step 2. The method on inputting the reduced displacements into the component model depends on the method used to output the external component in Step 2. The setups for Steps 2 and 3 are:

EXTDROUT Option	Step-2 Assembly Job	Step-3 Data Recovery for External SE
MATRIXDB, or DMIGDB	<p>If <i>dbext</i> is specified in EXTDROUT then in FMS:</p> <p>INIT <i>dbext</i></p> <p>In Case Control:</p> <p>EXTDROUT (MATRIXDB (or DMIGDB) [=dbext])</p> <p>On command line run with <i>scr=no</i> assuming job name is called 'step2.dat'</p>	<p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN SEYYY='step2.MASTER' DBLOCATE DATABLEK=(EXTDB), WHERE (SEID=YYY), LOGICAL=SEYYY</pre> <p>In Case Control:</p> <p>EXTDRIN (MATRIXDB or DMIGDB)</p>
DMIGOP2	<p>In FMS:</p> <pre>ASSIGN OUTPUT2='extsedr.op2', UNIT=45</pre> <p>In Case Control:</p> <p>EXTDROUT (DMIGOP2=45)</p>	<p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN INPUTT2='extsedr.op2', UNIT=45</pre> <p>In Case Control:</p> <p>EXTDRIN (DMIGOP2=45)</p>
MATOP4	<p>In FMS:</p> <pre>ASSIGN OUTPUT4='extsedr.op4', UNIT=45</pre> <p>In Case Control:</p> <p>EXTDROUT (MATOP4=45)</p>	<p>In FMS:</p> <pre>ASSIGN SEXX='step1.MASTER' RESTART LOGICAL=SEXX ASSIGN INPUTT4='extsedr.op4', UNIT=45</pre> <p>In Case Control:</p> <p>EXTDRIN (MATOP4=45)</p>

- For SOL 101, the Step 3 Case Control structure must match the system model subcase structure in the numbers of loading conditions. The loading used in step one to generate the loads transmitted to the analysis model must also be specified in this step. If the analysis model had more loading conditions than the component model, then the loadings defined in Step 1 must be specified first.
- For SOLs 103 and 107 through 112, the Step 3 Case Control structure must match the analysis model subcase structure in the number of eigenvalue extractions, *FREQ/DLOAD* or *TSTEP/DLOAD* subcases.

8. SOL 400 with multidisciplinary and linear perturbation analyses will produce multiple boundary solutions in Step 2; i.e., a boundary solution for each SUBCASE with a unique ANALYSIS command. Hence in SOL 400 for Step 3 as soon as the boundary solutions have been imported, a summary of the boundary solutions will be printed in the f06. Here is a sample:

Assembly Run solutions					
Solution ID	Analysis type	Subcase	Step	Super-element	Title/Subtitle/Label
1	STATICS	2	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED STATICS
		5	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED STATICS FOR BUCKLING ANALYSIS
2	MODES	7	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED NORMAL MODES - SOLVE ALL EIGENVALUES
3	BUCKLE	6	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED BUCKLING - SOLVE ALL EIGENVALUES
4	MODES	3	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL FREQUENCY - FOUR FREQUENCIES AND DELTA-F=.001 AND F1=1.04
5	MFREQ	3	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL FREQUENCY - FOUR FREQUENCIES AND DELTA-F=.001 AND F1=1.04
6	MODES	1	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001
7	MTRAN	1	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL TRANSIENT - FIFTY TIME STEPS AND DELTA-T=.001
8	MCEIG	4	0	100	MODEL T-6A (NTA) SEPARATION LOADS ANALYSIS, 270 KEAS HALF AIRCRAFT MODEL WITH PYLON & HMP,CNTRLS LOCKED MODAL COMPLEX - SOLVE FOR FIRST FIVE EIGENVALUES

The “Solution ID” column contains an arbitrary identification number which may be specified on the SOLUTION Case Control command in Step 3 whose usage will be described below. The “Analysis Type”, “Subcase”, and “Step” columns are the same as the ANALYSIS, SUBCASE, and STEP commands specified on Step 2. The “Superelement” column pertains the external superelement ID processed in Step 2. The “Title/Subtitle/Label” is taken from the TITLE, SUBTITLE, and LABEL commands specified in Step 2.

In Step 3, as with any restart, the Case Control must contain the same loads and boundary condition (LBC) commands specified in Step 1 like LOAD, SPC, MPC, METHOD, etc. If there is only one subcase specified in Step 3 then, by default, SOL 400 will perform external superelement data recovery for all solutions above. Here is a sample Step 3 Case Control setup:

```
EXTDRIN . . .
SPC=1
MPC=3
LOAD=5
$
DISP=ALL
STRESS=ALL
```

The displacement and element stress request for all solutions listed in the table above will be computed.

If, however, data recovery is desired at only a few solutions then a SUBCASE and SOLUTION command pair is required to obtain data recovery at each of the desired solutions. The subcases may be specified in any order and the SOLUTION command specifies the desired “Solution ID” from the table above. For example,

```
EXTDRIN . . .
SPC=1
MPC=3
SUBCASE 1
    SOLUTION=5
    STRESS=ALL
SUBCASE 2
    SOLUTION=3
    DISP=ALL
```

Alternatively, the ANALYSIS command may be used instead of the SOLUTION command. For example,

```
EXTDRIN . . .
SPC=1
MPC=3
SUBCASE 1
    ANALYSIS=MFREQ
    STRESS=ALL
SUBCASE 2
    ANALYSIS=BUCKLE
    DISP=ALL
```

If there is more than one solution associated with a particular analysis type, then results will be computed for all “Solution IDs” corresponding to that analysis type.

9. In Step 2, more than one external superelement may be processed by EXTDROUT. The following must be noted if DMIGOP2 or MATOP4 is used.
  - a. In SOLs 101, 103, and 107 through 112, the user must specify separate EXTDROUT as shown in the example below:

```
SUBCASE 1
    SUPER=10
    EXTDROUT (DMIGOP2=45)
SUBCASE 2
    SUPER=20
    EXTDROUT (DMIGOP2=46)
```

- b. In SOL 400, the user may specify separate EXTDROUT as described above or the same EXTDROUT command for all external superelements. If the user specifies the same EXTDROUT, then in the 3rd step a fatal message will be issued as shown below:

Assembly Run Solutions					
Solution ID	Analysis type	Subcase	Step	Super-element	Title/Subtitle/Label
1	MODES	1	0	10	MODAL, 3-STEP EXTERNAL SE, DMIGDB, ASSEMBLY
2	MODES	1	0	20	MODAL, 3-STEP EXTERNAL SE, DMIGDB, ASSEMBLY

\*\*\* USER FATAL MESSAGE 22907 (EXTDRSUM)



The attached boundary solutions %1 contains solutions for more than one external superelement and the desired superelement ID has not been selected.

**USER ACTION:** If the EXTDROUT Case Control command (or PARAM,EXTDROUT) specified DMIGOP2 or MATOP4 then you must specify the superelement ID on the SUPER Case Control command to select the appropriate boundary solution.

If EXTDROUT Case Control command (or PARAM,EXTDROUT) specified MATRIXDB or DMIGDB then you must specify the superelement ID

a) On the SUPER Case Control command to select the appropriate boundary solution.  
or  
b) In the WHERE(SEID=superelement\_ID) clause on the DBLOCATE statement of the boundary solutions database to select the appropriate boundary solution.

The summary shows that the solution file contains solutions for superelements 10 and 20 and the message indicates that only one solution may be processed. So the user must specify the SUPER command to select the desired superelement; e.g.,

```
SUPER=10
EXTDRIN (DMIGOP2=45)
```

Alternatively for EXTDRIN(DMIGDB or MATRIXDB) only, the user may specify WHERE(SEID=10) on the DBLOCATE statement of the boundary solution database.

10. The following examples continue from the examples described in Remark 18 under the EXTSEOUT Case Control command description.

#### Example 1. MATDB / MATRIXDB or DMIGDB Option

- Assembly Job (assume input file is named “assembly.dat”)
  - File Management Section (FMS) Requirement only if it is desired to store the boundary solutions on a dbset separate from DBALL

```
INIT EXTDRDB
```

- Case Control Requirement for MATDB / MATRIXDB Option.

Store boundary solutions on the DBALL dbset

```
EXTDROUT
```

Store boundary solutions on the EXTDRDB dbset

```
EXTDROUT ( MATRIXDB = EXTDRDB )
```

- Case Control Requirement for DMIGDB Option

Store boundary solutions on the DBALL dbset

```
EXTDROUT ( DMIGDB )
```

Store boundary solutions on the EXTDRDB dbset

```
EXTDROUT ( DMIGDB = EXTDRDB )
```

For all options above, “scr = no” must be specified on the Nastran command line to ensure that the databases are saved at the end of the jobs.

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)
  - File Management Section (FMS) Requirement

### Example 1. MATDB / MATRIXDB or DMIGDB Option

```
ASSIGN SE10='extse10.MASTER'
RESTART LOGI=SE10
ASSIGN ASMB='assembly.MASTER'
DBLOC DATABLK=(EXTDB) WHERE(SEID=10) LOGI=SE10
```

- Case Control Requirement for MATDB / MATRIXDB Option

```
EXTDRIN
```

- Case Control Requirement for DMIGDB Option

```
EXTDRIN ( DMIGDB )
```

### Example 2. DMIGOP2 Option

- Assembly Job

- Setup for SOLs 101, 103, 107 through 112, and 400 each superelement boundary solution is stored on a separate Fortran unit

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT2='extse10dr.op2' UNIT=25 DELETE
ASSIGN OUTPUT2='extse20dr.op2' UNIT=26 DELETE
ASSIGN OUTPUT2='extse30dr.op2' UNIT=27 DELETE
```

- Case Control Requirement

```
SUBCASE 1
SUPER = 10
EXTDROUT ( DMIGOP2 = 25 )
SUBCASE 2
SUPER = 20
EXTDROUT ( DMIGOP2 = 26 )
SUBCASE 3
SUPER = 30
EXTDROUT ( DMIGOP2 = 27 )
```

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT2='extalldr.op2' UNIT=25 DELETE
```

- Case Control Requirement

```
EXTDROUT ( DMIGOP2 = 25 )
```

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)

- Setup for SOLs 101, 103, 107 through 112, and 400



**Example 2. DMIGOP2 Option**

- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'
RESTART LOGICAL=SE10
ASSIGN INPUTT2='extse10dr.op2' UNIT=25
```

- Case Control Requirement

```
EXTDRIN ( DMIGOP2 = 25 )
```

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit

- File Management Section (FMS) Requirement

```
ASSIGN SE10='extse10.MASTER'
RESTART LOGICAL=SE10
ASSIGN INPUTT2='extalldr.op2' UNIT=25
```

- Case Control Requirement – SUPER command is required to select the correct superelement boundary solution

```
SUPER = 10
EXTDRIN ( DMIGOP2 = 25 )
```

**Example 3. MATOP4 Option**

- Assembly Job

- Setup for SOLs 101, 103, 107 through 112, and 400 each superelement boundary solution is stored on a separate Fortran unit.

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT4='extse10dr.op4' UNIT=25 DELETE
ASSIGN OUTPUT4='extse20dr.op4' UNIT=26 DELETE
ASSIGN OUTPUT4='extse30dr.op4' UNIT=27 DELETE
```

- Case Control Requirement

```
SUBCASE 1
SUPER = 10
EXTDROUT ( MATOP4 = 25 )
SUBCASE 2
SUPER = 20
EXTDROUT ( MATOP4 = 26 )
SUBCASE 3
SUPER = 30
EXTDROUT ( MATOP4 = 27 )
```

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit.

### Example 3. MATOP4 Option

- File Management Section (FMS) Requirement

ASSIGN OUTPUT4='extalldr.op4' UNIT=25 DELETE

- Case Control Requirement

EXTDROUT ( MATOP4 = 25 )

- External Superelement Data Recovery Restart Job (repeat following setup for each superelement)

- Setup for SOLs 101, 103, 107 through 112, and 400

- File Management Section (FMS) Requirement

ASSIGN SE10='extse10.MASTER'

RESTART LOGICAL=SE10

ASSIGN INPUTT4='extse10dr.op4' UNIT=25

- Case Control Requirement

EXTDRIN ( MATOP4 = 25 )

- Alternative setup for SOL 400 only – all superelement boundary solutions are stored on the same Fortran unit

- File Management Section (FMS) Requirement

ASSIGN SE10='extse10.MASTER'

RESTART LOGICAL=SE10

ASSIGN INPUTT4='extalldr.op4' UNIT=25

- Case Control Requirement – SUPER command is required to select the correct superelement boundary solution

SUPER = 10

EXTDRIN ( MATOP4 = 25 )

**EXTDROUT**

## External Superelement Boundary Solution Storage Request.

Requests the job to store external superelement boundary displacements and column labels; e.g., eigenvalues, forcing frequencies, time steps. Also specifies the storage media of the boundary solution data.

**Format:**

$$\text{EXTDROUT} \left[ \begin{array}{l} DMIGOP2 [= unitop2] \\ MATRIXOP4 \text{ or } MATOP4 [= unitop4] \\ \text{MATRIXDB or MATDB} [= dbext] \\ DMIGDB [= dbext] \end{array} \right] \text{ NOCASE}$$
**Examples:**

```
EXTDROUT
EXTDROUT (DMIGOP2=45)
EXTDROUT (MATOP4 NOCASE)
```

Describer	Meaning
DMIGOP2 = unitop2	Store the boundary solution data in DMIG Bulk Data internal format on an .op2 file whose Fortran unit number is given by unitop2 (Integer > 0, Default=31).
MATRIXOP4 or MATOP4 = unitop4	Store the boundary solution data on an .op4 file whose Fortran unit number is given by <i>unitop4</i> . (Integer ≠ 0, Default=31). <i>unitop4</i> is a non-zero integer with the following meanings:  <i>unitop4</i> > 0: Store in sparse format  <i>unitop4</i> < 0: Store in non-sparse format on Fortran unit number given by  unitop4
MATRIXDB or MATDB = dbext	Store the boundary solution data in the standard matrix format in the database. Dbext is the name of a new dbset on which to store the boundary data. Dbext must be allocated on an INIT FMS statement.
DMIGDB	Store the boundary solution data in the DMIG Bulk Data internal format in the database. Dbext is the name of a new dbset on which to store the boundary data. Dbext must be allocated on an INIT FMS statement.
NOCASE	NOCASEBy default in SOL 400 only, the TITLE/SUBTITLE/LABEL contents are exported along with boundary data. Specify NOCASE if you do not want these contents to be exported.

**Remarks:**

1. EXTDROUT is intended for step two (or assembly run) in external superelement analysis.
2. EXTDROUT is honored in SOLs 101, 103, 107 through 112, and 400 only.

3. In SOLs 101, 103, and 107 through 112, if there are multiple external superelements and DMIGOP2 or MATOP4 is requested then EXTDROUT must be specified in separate subcases for each external superelement along with the SUPER command and unitop4 or unitop2 must be unique for each external superelement. Separate EXTDROUT commands are not required in SOL 400. See Remarks 9. and 10. under the EXTDRIN Case Control command description.
4. User parameters EXTDROUT and EXTDRUNT are the old method for requesting the storage of boundary solution data. EXTDROUT Case Control command takes precedence over the EXTDROUT and EXTDRUNT user parameters.
5. See EXTDRIN's case control command description for a description of EXTDROUT and EXTDRIN usage.

## EXTSEOUT

## External Superelement Creation Specification

Request the creation of an external superelement in SOLs 101, 103, 107 through 112, or 400 only.

### Format:

$$\text{EXTSEOUT} \left[ \left[ \text{STIFFNESS} \text{ MASS} \text{ DAMPING} \text{ K4DAMP} \text{ LOADS} \text{ FSCOUPL}, \right. \right.$$

$$\left. \left. \text{ASMBULK} \left[ = \left\{ \begin{array}{l} \text{MAN} \\ \text{MANQ} \\ \text{AUTO} \end{array} \right\}, \text{EXTBULK}, \text{EXTID} = \textit{seid}, \left[ \begin{array}{l} \text{QSET} \\ \text{ASET} \end{array} \right], \right. \right.$$

$$\left. \left. \text{DMIGSFIX} = \left\{ \begin{array}{l} \textit{cccccc} \\ \text{EXTID} \end{array} \right\} \left[ \begin{array}{l} \text{MATDB (or MATRIXDB) [= dbext]} \\ \text{DMIGDB [= dbext]} \\ \text{DMIGOP2 = unitop2} \\ \text{DMIGPCH} \\ \text{MATOP4 (or MATRIXOP4) = unitop4} \end{array} \right] \right] \right.$$

### Example(s):

```

EXTSEOUT
EXTSEOUT (DMIGOP2=26)
EXTSEOUT (ASMBULK EXTID=200)
EXTSEOUT (ASMBULK EXTBULK EXTID=100)
EXTSEOUT (ASMBULK=AUTO EXTBULK EXTID=100)
EXTSEOUT (ASMBULK=MANQ EXTID=10 DMIGDB)
EXTSEOUT (ASMBULK EXTID=100 DMIGOP2=26)
EXTSEOUT (ASMBULK EXTID=100 DMIGPCH)
EXTSEOUT (ASMBULK EXTID=100 DMIGSFIX=XSE100 DMIGPCH)
EXTSEOUT (ASMBULK EXTID=200 DMIGSFIX=EXTID DMIGPCH)
EXTSEOUT (ASMBULK EXTID=100 MATOP4=26)
EXTSEOUT (ASMBULK EXTID=200 MATOP4=-25)

```

(See also Remarks 17. and 18.)

Describer	Meaning
STIFFNESS	Store the boundary stiffness matrix. See Remarks 1. and 2.
MASS	Store the boundary mass matrix. See Remark 1.
DAMPING	Store the boundary viscous damping matrix. See Remarks 1. and 2.

Describer	Meaning
K4DAMP	Store the boundary structural damping matrix. See Remark 1.
LOADS	Store the boundary static loads matrix. See Remark 1.
FSCOU	Store the boundary fluid-structure coupling matrix. See Remark 1.
ASMBULK or ASMBULK = MAN	Generate Bulk Data entries for use in a subsequent superelement assembly process and store them on the assembly punch file (.asm). This data, which is used in the main bulk data portion of a subsequent assembly job, includes an SEBULK entry that specifies MANUAL as the method for searching boundary points, and an SECONCT entry that defines connections for boundary grid and scalar points. See Remarks 3., 6., and 15.
ASMBULK = MANQ	Similar to the ASMBULK = MAN option, except that the generated SECONCT entry defines connections not only for boundary grid and scalar points, but also for Q-set points. This allows the user to have control over the Q-set points of the external superelement in the subsequent assembly job. See Remarks 3., 5., and 15.
ASMBULK = AUTO	Generate Bulk Data entries for use in a subsequent superelement assembly process, and store them on the assembly punch file (.asm). This data, which is to be used in the main bulk data portion of a subsequent assembly job, includes an SEBULK entry that specifies AUTO as the method for searching boundary points, and an SECONCT entry that defines connections for boundary scalar points. See Remarks 3., 6., and 15.
EXTBULK	Generate Bulk Data entries related to the external superelement and store them on the standard punch file (.pch). This data is used in the BEGIN SUPER portion of the bulk data of a subsequent assembly job. EXTBULK need not be specified if DMIGPCH or MATOP4 (see the following descriptions) is specified. See Remarks 3., 7., 8., and 15.  Note that, in general, the EXTBULK keyword is not required and is provided solely for user convenience. In the absence of EXTBULK and the associated output on the standard punch file resulting from it, the subsequent assembly job will retrieve the required data for the external superelement from the medium on which the boundary matrices are stored.
EXTID= <i>seid</i>	<i>seid</i> (integer > 0) is the superelement ID to be used on the SEBULK and SECONCT (if applicable) Bulk Data entries stored on the assembly punch file (.asm) if ASMBULK is specified, and in the BEGIN SUPER Bulk Data entry stored on the standard punch file (.pch) if EXTBULK or DMIGPCH is specified. See Remarks 3. - 9., 13., and 17.
DMIGSFIX = <i>cccccc</i>	<i>cccccc</i> is the suffix (up to six characters) to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks 10. - 13. See also Example 3 in Remark 18.

Describer	Meaning
DMIGSFIX = EXTID	The <i>seid</i> defined by the EXTID keyword is the suffix to be employed in the names of the DMIG matrices stored on the standard punch file (.pch) if the DMIGPCH option is specified. See Remarks 3., 10. - 13. See also Example 3 in Remark 18.
MATDB (or MATRIXDB) = dbext (Default)	Store the boundary matrices and other information on the database. Dbext is the name of a new dbset on which to store the boundary matrices. Dbext must be allocated on an INIT FMS statement. By default, matrices are stored on DBALL. See Example 1 in Remark 18. Dbext is required if 3-step external superelement data recovery will be performed.
DMIGDB= dbext	Similar to MATDB (or MATRIXDB) except that the boundary matrices are stored as DMIG Bulk Data entries on the database. Dbext is the name of a new dbset on which to store the boundary matrices. Dbext must be allocated on an INIT FMS statement. By default, matrices are stored on DBALL. See Example 1 in Remark 18. Dbext is required if 3-step external superelement data recovery will be performed.
DMIGOP2 = <i>unitop2</i>	Store the boundary matrices as DMIG Bulk Data entries on an .op2 file whose Fortran unit number is given by <i>unitop2</i> (Integer > 0). See Remark 16. See also Example 2 in Remark 18.
DMIGPCH	Store the boundary matrices as DMIG Bulk Data entries on the standard punch file (.pch). Note that, if boundary matrices include both structural and fluid grids, use DMIGOP2 instead. See Remarks 3. and 7. - 15. See also Example 3 in Remark 18.
MATOP4 = <i>unitop4</i> (or MATRIXOP4 = <i>unitop4</i> )	<p><i>unitop4</i> is a non-zero integer with the following meanings:</p> <p><i>unitop4</i> &gt; 0</p> <p>Store the boundary matrices in <i>sparse</i> format on an .op4 file whose Fortran unit number is given by <i>unitop4</i></p> <p><i>unitop4</i> &lt; 0</p> <p>Store the boundary matrices in <i>non-sparse</i> format on an .op4 file whose Fortran unit number is given by <math> unitop4 </math></p> <p>See Remarks 3., 7., 8., 15., and 16. See also Example 4 in Remark 18.</p>
ASET/QSET	Specifies how the external superelement's q-set dofs (dynamic degrees-of-freedom) are defined on the exported file. By default, all q-set dofs are written to QSET/QSET1 entries which means the superelement's modes will be "added" to but not synthesized into the assembly's modes. By specifying ASET, the q-set dofs are written to ASET/ASET1 entries and will be synthesized into the assembly's. The ASET/QSET switch does not affect DMIGPCH media option which always writes ASET/ASET1 entries.

**Remarks:**

1. If none of the descriptors STIFFNESS through FSCOUPL is specified, then all matrices are stored.
2. STIFFNESS and DAMPING may be abbreviated to STIF and DAMP, respectively.
3. EXTID with an *seid* value must be specified if one or more of ASMBULK, EXTBULK, DMIGPCH, or MATOP4 are specified.

If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, the value *seid* may not exceed 999999 since this value becomes part of the names given to the DMIG matrices generated on the standard punch file (.pch). See Remark 13., and Example 3 in Remark 18.

If PARAM,AUTOQSET,YES is specified to used generate the Q-set degrees of freedom (DOFs) (generalized coordinates), the value *seid* may not exceed 999 since this value becomes part of the automatically generated IDs of the SPOINTs representing the Q-set DOFs. See explanation in Item c under Remark 17.

4. If ASMBULK is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK  seid ... (specifies MANUAL as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary grid and scalar
points)
```

```
GRID entries for boundary and access grid points
SPOINT entries for boundary and access scalar points
CORD2x entries associated with the boundary and access GRID
entries
```

5. If ASMBULK = MANQ is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK  seid ... (specifies MANUAL as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary grid and
scalar points as well as for Q-set points)
GRID entries for boundary and access grid points
SPOINT entries for boundary and access scalar points as well as
for Q-set points
CORD2x entries associated with the boundary and access GRID
entries
```

6. If ASMBULK = AUTO is specified, the following Bulk Data entries are generated and stored on the assembly punch file (.asm):

```
SEBULK  seid ... (specifies AUTO as the method for searching
boundary points)
SECONCT seid ... (defines connections for boundary scalar
points)
```

```
GRID entries for access grid points
CORD2x entries associated with the access grid points
SPOINT entries for boundary and access scalar points
```



7. If DMIGPCH or MATOP4 is specified, then EXTBULK need not be specified. Also, the model cannot have rotors (ROTOR, ROTORG, ROTORSE, and ROTORAX Bulk Data entries).
8. If DMIGPCH is not specified, but EXTBULK or MATOP4 is specified, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

```
BEGIN BULK seid
GRID entries for boundary points
GRID entries for interior points referenced by PLOTEL entries
SPOINT entries for boundary scalar points as well as for Q-set
points
CORD2x entries associated with the above GRID entries
EXTRN
ASET
QSET/QSET1
PLOTEL
```

9. If DMIGPCH is specified, the following Bulk Data entries are generated and stored on the standard punch file (.pch):

```
BEGIN SUPER seid
GRID entries for boundary points
SPOINT entries for boundary scalar points as well as for Q-set
points
CORD2x entries associated with the boundary GRID entries
ASET/ASET1
PLOTEL entries referencing the boundary GRID entries
DMIG entries for the requested boundary matrices
```

10. The DMIGSFIX keyword is ignored if DMIGPCH is not specified.
11. If DMIGPCH is specified without the DMIGSFIX keyword, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

```
KAAX (boundary stiffness matrix)
MAAX (boundary mass matrix)
BAAX (boundary viscous damping matrix)
K4AAX (boundary structural damping matrix)
PAX (boundary load matrix)
AAX (boundary fluid-structure coupling matrix)
```

See Example 3 in Remark 18.

12. If the DMIGSFIX = *cccccc* form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:

```
Kcccccc (boundary stiffness matrix)
Mcccccc (boundary mass matrix)
Bcccccc (boundary viscous damping matrix)
K4cccccc (boundary structural damping matrix)
Pcccccc (boundary load matrix)
Acccccc (boundary fluid-structure coupling matrix)
```

See Example 3 in Remark 18.

13. If the DMIGSFIX = EXTID form is employed along with the DMIGPCH specification, then the boundary DMIG matrices generated and stored on the standard punch file (.pch) will have names of the following form:  

```
Kseid (boundary stiffness matrix)
Mseid (boundary mass matrix)
Bseid (boundary viscous damping matrix)
K4seid (boundary structural damping matrix)
Pseid (boundary load matrix)
Aseid (boundary fluid-structure coupling matrix)
```

where *seid* is the superelement ID specified by the EXTID keyword.  
See Example 3 in Remark 18.
14. If the DMIGPCH option is specified, the boundary DMIG matrices generated and stored on the standard punch file (.pch) may not be as accurate as the boundary matrices resulting from the other options (MATDB/MATRIXDB, or DMIGOP2 ,or MATOP4). Accordingly, this may result in decreased accuracy from the subsequent assembly job using these DMIG matrices.
15. The punch output resulting from EXTSEOUT usage is determined by ASMBULK, EXTBULK, DMIGPCH, and MATOP4 as follows:
  - No ASMBULK, EXTBULK, DMIGPCH or MATOP4  
No punch output.
  - ASMBULK, but no EXTBULK, or DMIGPCH, or MATOP4  
Punch output is generated and stored on the assembly punch file (.asm) as indicated in Remarks 4. through 6.
  - No ASMBULK, but EXTBULK, DMIGPCH, or MATOP4  
Punch output is generated and stored on the standard punch file (.pch) as indicated in Remarks 8. or 9. (as appropriate).
  - ASMBULK and EXTBULK, DMIGPCH or MATOP4  
Punch output consists of two distinct and separate parts. One part is generated and stored on the assembly punch file (.asm) as indicated in Remarks 4. through 6. The other part is generated and stored on the standard punch file (.pch) as indicated in Remarks 8. or 9. (as appropriate).
16. If DMIGOP2 = *unitop2* or MATOP4 = *unitop4* is specified, an appropriate ASSIGN OUTPUT2 or ASSIGN OUTPUT4 statement must be present in the File Management Section (FMS) for the specified Fortran unit numbers. See Examples 2 and 4 in Remark 18.
17. The creation of an external superelement using EXTSEOUT involves running a Nastran job, with the following additional data:
  - a. The data for the creation of the external superelement is specified by the EXTSEOUT Case Control command. It is best to specify this command above the subcase level. However, if the Nastran job itself involves superelements, then the EXTSEOUT command may also be specified within the subcase that is applicable to the residual.
  - b. The boundary points of the external superelement are specified by ASET/ASET1 Bulk Data entries.

- c. If the external superelement creation involves component mode reduction, then Q-set degrees of freedom (DOFs) (generalized coordinates) must be defined. This can be done either by automatically defining them via PARAM,AUTOQSET,YES or by explicitly specifying them via QSET/QSET1 Bulk Data entries. Note that for the latter case of explicit definition, only SPOINTs may be specified as Q-set points; no grid points may be specified. If this requirement is not met, the program terminates the execution with an appropriate fatal message.

If PARAM,AUTOQSET,YES is specified, SPOINTs are automatically generated internally to represent the Q-set DOFs. The IDs of these SPOINTs are of the form 9 $sssnnnn$  where  $sss$  is the superelement ID  $seid$  specified by the EXTID keyword, and  $nnnn$  is a mode number. Both  $sss$  and  $nnnn$  will have leading zeros inserted in them to ensure that  $sss$  is a three-digit number and  $nnnn$  is a four-digit number. Thus, for example, the Q-set DOF corresponding to the 8<sup>th</sup> mode of superelement ID 5 would be represented by an SPOINT with an automatically generated ID of 90050008, while the Q-set DOF corresponding to the 50<sup>th</sup> mode of superelement ID 25 would be represented by an SPOINT with an automatically generated ID of 90250050.

Because of the preceding numbering scheme, the superelement ID  $seid$  specified by the EXTID keyword must necessarily not exceed 999 when PARAM,AUTOQSET,YES is specified. The program terminates the job with a User Fatal Message if this condition is not met.

Further, because of the preceding numbering scheme, the user should ensure that, when PARAM,AUTOQSET,YES is specified, the IDs of the grid and scalar points of the external superelement do not conflict with the automatically generated SPOINT IDs of the Q-set DOFs. The program terminates the job with a User Fatal Message if this condition is not met.

- d. The fixity of the boundary DOFs for the component mode reduction may be specified using the BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 Bulk Data entries. (The default scenario assumes that all boundary DOFs are fixed for component mode reduction.)
- e. The output for the external superelement is generated in the assembly job. This output consists of displacements, velocities, accelerations, SPC forces, MPC forces and element stresses, strains and forces. However, in order for this output to be generated in the assembly job, the output requests must be specified in the external superelement creation run. Normally, the only output requests for the external superelement that are honored in the assembly job are those that are specified in the creation run. There is, however, one important exception to this: the output for the boundary and access grid and scalar points, as well as for all grid points associated with PLOT entries, can be obtained in the assembly job *even if there is no output request specified for these points in the creation run.*

If the creation run contains the load requests TEMPERATURE (LOAD) or DEFORM, then element stresses and element forces will not be available in the assembly run.

- f. If the assembly job involves the use of PARAM Bulk Data entries, then the following points should be noted:
- PARAM entries specified in the main bulk data portion of the input data apply *only to the residual and not to the external superelements.*
  - PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for an external superelement apply *only to that superelement.*

- The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all external superelements, is to specify such PARAM entries in the Case Control Section, not in the main bulk data. This is particularly relevant for PARAMs such as POST.
  - g. If EXTSEOUT is used in SOL 400 for a multidisciplinary or linear perturbation analysis and there are rigid elements present then RIGID=LINEAR must be specified.
18. The following examples illustrate details of job setups for the external superelement creation and the subsequent assembly process for various scenarios. These examples assume that there are three external superelement creation jobs, one each for external SE 10 (extse10.dat), SE 20 (extse20.dat) and SE 30 (extse30.dat), followed by an assembly job.

Example 1. MATDB / MATRIXDB or DMIGDB Option	
■	External SE Creation Jobs
•	File Management Section FMS Requirement only if 3rd step external superelement data recovery is desired
	SE 10: INIT EXTDB
	SE 20: INIT EXTDB
	SE 30: INIT EXTDB
	These INIT statements along with the DBEXT keyword below will create separate DBsets called “extse10.EXDTB”, “extse20.EXTDB”, and “extse30.EXTDB” and store the necessary data for the Assembly Job on them. All other data will be stored on DBALL.
•	Case Control Requirement for the MATDB / MATRIXDB option if 3 <sup>rd</sup> step external superelement data recovery is <b>not</b> desired.
	SE 10: EXTSEOUT (ASMBULK EXTID = 10)
	SE 20: EXTSEOUT (ASMBULK EXTID = 20)
	SE 30: EXTSEOUT (ASMBULK EXTID = 30)
	The EXTBULK keyword may be specified, but it is not necessary.
•	Case Control Requirement for the DMIGDB option if 3 <sup>rd</sup> step external superelement data recovery is <b>not</b> desired.
	SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB)
	SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGDB)
	SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGDB)
	For both options, scr = no should be specified on the Nastran job command line to ensure that the databases are saved at the end of the jobs.
•	Case Control Requirement for the MATDB / MATRIXDB option if 3 <sup>rd</sup> step external superelement data recovery is desired.
	SE 10: EXTSEOUT (ASMBULK EXTID = 10 MATDB=SE10)
	SE 20: EXTSEOUT (ASMBULK EXTID = 10 MATDB=SE20)

**Example 1. MATDB / MATRIXDB or DMIGDB Option (continued)**

SE 30: EXTSEOUT (ASMBULK EXTID = 10 MATDB=SE30)

- Case Control Requirement for the DMIGDB option if 3<sup>rd</sup> step external superelement data recovery is desired.

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB=SE10)

SE 20: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB=SE20)

SE 30: EXTSEOUT (ASMBULK EXTID = 10 DMIGDB=SE30)

- Assembly Job

- File Management Section (FMS) Requirement

(Note: If DBEXT was specified on the EXTSEOUT command in the External SE Creation Jobs then the extse10.DBALL, etc. files may be hidden (moved or renamed) from the Assembly Job for purposes of security or privacy.)

```
ASSIGN dbname10='extse10.MASTER'
```

```
ASSIGN dbname20='extse20.MASTER'
```

```
ASSIGN dbname30='extse30.MASTER'
```

```
DBLOCATE DATABLE=(EXTDB, EXTROTDB) CONVERT(SEID=10)
```

```
LOGICAL=dbname10
```

```
DBLOCATE DATABLE=(EXTDB,EXTROTDB) CONVERT(SEID=20)
```

```
LOGICAL=dbname20
```

```
DBLOCATE DATABLE=(EXTDB, EXTROTDB) CONVERT(SEID=30)
```

```
LOGICAL=dbname30
```

(Note: All of the data blocks stored on the databases for the external SEs have the same common name of EXTDB or EXTROTDB for external SEs with rotors.)

- Case Control Requirement

If 3<sup>rd</sup> step data recovery is desired for the external superelement, specify EXTDROUT Case Control command in a separate subcase for each superelement. See also the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the Main Bulk Data.

```
INCLUDE 'extse10.asm'
```

```
INCLUDE 'extse20.asm'
```

```
INCLUDE 'extse30.asm'
```

### Example 2. DMIGOP2 Option

- External SE Creation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2='extse10\_op2' UNIT=25 DELETE

ASSIGN OUTPUT2='extse20\_op2' UNIT=26 DELETE

ASSIGN OUTPUT2='extse30\_op2' UNIT=27 DELETE

- Case Control Requirement

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGOP2 = 25)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGOP2 = 26)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGOP2 = 27)

The EXTBULK keyword may be specified, but it is not necessary.

scr = yes may be specified on the Nastran command line unless a 3rd step external superelement data recovery restart is desired.

- Assembly Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2='extse10\_op2' UNIT=25

ASSIGN INPUTT2='extse20\_op2' UNIT=26

ASSIGN INPUTT2='extse30\_op2' UNIT=27

- Case Control Requirement

If 3rd step data recovery is desired for the external superelement, specify EXTDROUT Case Control command in a separate subcase for each superelement. See also the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

INCLUDE 'extse10.asm'

INCLUDE 'extse20.asm'

INCLUDE 'extse30.asm'

### Example 3. DMIGPCH Option

- External SE Creation Jobs

- Case Control Requirement

SE 10: EXTSEOUT (ASMBULK EXTID = 10 DMIGPCH)

SE 20: EXTSEOUT (ASMBULK EXTID = 20 DMIGPCH,

**Example 3. DMIGPCH Option (continued)**

DMIGSFIX = XSE20)

SE 30: EXTSEOUT (ASMBULK EXTID = 30 DMIGPCH,

DMIGSFIX = EXTID)

scr = yes may be specified on the Nastran command line since there is no need for the databases to be saved at the end of the jobs.

- Assembly Job

- Case Control Requirement

K2GG = (KAAX, KXSE20, K30)

M2GG = (MAAX, MXSE20, M30)

B2GG = (BAAX, BXSE20, B30)

K42GG = (K4AAX, K4XSE20, K430)

P2G = (PAX, PXSE20, P30)

A2GG = (AAX, AXSE20, A30)

- Case Control Requirement

If 3rd step data recovery is desired for the external superelement, specify EXTDROUT Case Control command along with the commands above in a separate subcase for each superelement. See also the examples under Remark 10 of the EXTDRAIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

INCLUDE 'extse10.asm'

INCLUDE 'extse20.asm'

INCLUDE 'extse30.asm'

The following INCLUDEs are also required. They must be grouped together and specified at the very end of the main bulk data (just before the ENDDATA delimiter).

INCLUDE 'extse10.pch'

INCLUDE 'extse20.pch'

INCLUDE 'extse30.pch'

### Example 4. MATOP4 Option

#### ■ External SE Creation Jobs

- File Management Section (FMS) Requirement

```
ASSIGN OUTPUT4='extse10_op4' UNIT=25 DELETE
```

```
ASSIGN OUTPUT4='extse20_op4' UNIT=26 DELETE
```

```
ASSIGN OUTPUT4='extse30_op4' UNIT=27 DELETE
```

- Case Control Requirement

```
SE 10:  EXTSEOUT (ASMBULK EXTID = 10 MATOP4 = 25)
```

```
SE 20:  EXTSEOUT (ASMBULK EXTID = 20 MATOP4 = 26)
```

```
SE 30:  EXTSEOUT (ASMBULK EXTID = 30 MATOP4 = -27)
```

Note: Boundary matrices will be stored in *sparse* format on .op4 files with Fortran unit numbers 25 and 26 and in *non-sparse* format on the .op4 file with Fortran unit number 27.

scr = yes may be specified on the Nastran command line unless a 3rd step external superelement data recovery restart is desired.

#### ■ Assembly Job

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT4='extse10_op4' UNIT=25
```

```
ASSIGN INPUTT4='extse20_op4' UNIT=26
```

```
ASSIGN INPUTT4='extse30_op4' UNIT=27
```

- Case Control Requirement

If 3rd step data recovery is desired for the external superelement, specify EXTDROUT Case Control command in a separate subcase for each superelement. See also the examples under Remark 10 of the EXTDRIN Case Control command for additional File Management and Case Control requirements for the Assembly Job.

- Bulk Data Requirement

The following INCLUDEs are required. They may be specified anywhere in the main bulk data.

```
INCLUDE 'extse10.asm'
```

```
INCLUDE 'extse20.asm'
```

```
INCLUDE 'extse30.asm'
```

The following INCLUDEs are also required. They must be grouped together and specified at the very end of the main bulk data (just before the ENDDATA delimiter).

```
INCLUDE 'extse10.pch'
```

```
INCLUDE 'extse20.pch'
```

```
INCLUDE 'extse30.pch'
```



19. If EXTBULK is specified, then METADATA entries will be copied to the .pch file.
20. Data recovery for the external superelement is possible via two methods: two-step and three-step.
  - a. In the two-step method, EXTSEOUT automatically generates output transformation matrices (OTMs) based on STRESS, FORCE, DISPLACEMENT, SPCFORCE, MPCFORCE, STRAIN, and MONITOR Case Control commands specified in the first (creation) run. Then in the second (assembly) run, data recovery is performed on the external superelement. This data recovery is limited to certain element types and specialized outputs. For example, grid point forces and stresses, corner stresses, strain and kinetic energies, and composite stresses are not supported in the OTMs. If Modules are present in the first run, then OTMs are generated for those grid points and elements in Module 0 only.
  - b. The three-step method requires a restart of the first run to perform data recovery on the external superelement. This method supports all data recovery options in the two-step method as well as grid point forces and stresses, corner stresses, strain and kinetic energies, and composite stresses. The method is explained in the description of the EXTDRIN Case Control command. If Modules are present in the first run, this method allows for data recovery in any Module.
  - c. Both methods are described in more detail in the *MSC Nastran Superelements and Modules User's Guide*.
  - d. For dynamically reduced external superelements, the displacement OTMs created with dynamic reduction include both the dynamic effects of the o-set and also the fixed boundary effects of the o-set due to interior loads. With the two-step method, only one of these effects should be included in a SOL 101, 106 and 400 (with ANALYSIS=STATICS) assembly. Therefore, if there are interior loads in the external superelement and the two-step method is being used, attaching the dynamically reduced external superelements in a static analysis of the assembly is not recommended.
21. Superelement can be used in conjunction with residual structures in fluid-structure interaction analysis / acoustic analysis, with the following restrictions:
  - a. If the superelement contains both structural and fluid degrees-of-freedom (dof), then the residual structure can only have either structural dof or fluid dof - it cannot have both.
  - b. Similarly, if the residual structure has both structural and fluid dof, then the superelement should only have either structural dof or fluid dof - it cannot have both.

## FATIGUE

### Fatigue Output Request

Requests one or more fatigue analyses for use in pseudo-static (SOL 101), modal (SOL 103), modal transient (SOL 112), and frequency response (SOL 108 and SOL 111) runs.

#### Format (SOL 101, 103, 112, 200 with ANALYSIS=STATICS):

$$\text{FATIGUE} \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \text{FORMAT} = [\text{CODE}], \left[ \begin{array}{l} \text{BULK} \\ \text{SET} \end{array} \right], \left[ \text{STATICS} \right], \text{STROUT} = [\text{CODE}] \right) = \left\{ \begin{array}{l} \text{n} \end{array} \right\}$$

#### Format (SOL 108, 111, 200 with ANALYSIS=DFREQ or MFREQ) (See Remark 12.):

$$\text{FATIGUE} \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \text{FORMAT} = [\text{CODE}], \left[ \text{BULK} \right], \left[ \begin{array}{l} \text{FREQ} \\ \text{DFREQ} \\ \text{MFREQ} \end{array} \right] \right) = \left\{ \begin{array}{l} \text{n} \end{array} \right\}$$

#### Examples:

FATIGUE=100

SET 99 = 100, 200

FATIGUE (SET) = 99

Descriptor	Meaning
SORT1	Printed output will be presented as a tabular listing of elements for each fatigue analysis or event. In the case of duty cycle, where multiple events are output (EVNTOUT=1 on FTGSEQ bulk data), each event is presented as a separate fatigue analysis. Output written to the OEFTG data blocks is always in SORT1 format.
SORT2	Printed output will be presented as a tabular listing of events for each element, node, or angle. This is only applicable for duty cycle (loading sequences with more than one event - EVNTOUT=1 on FTGSEQ bulk data). If there is only one event, the output is identical to SORT1. Output written to the OEFTG data blocks is always in SORT1 format.

Describer	Meaning
-----------	---------

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

See Remark 15. regarding data blocks produced by the FATIGUE case control.

FORMAT	Request that fatigue be output in specific file formats. See Remark 6. and 11.
STROUT	Request that certain stress or strain output be included. See Remarks 13. and 14.
CODE	Codes for specific file format or stress output. See Remark 6., 11., and 13.
BULK	Specifies that the <i>n</i> refers directly to bulk data IDs of FTGSEQ/ FTGLOAD, FTGPARM and FTGDEF entries of the same ID (default) to process.
SET	Specifies that the <i>n</i> refers to a previously appearing SET ID (SOL 101, 103, & 112 only) containing the IDs of FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries to process. This is a mechanism to request multiple fatigue output requests in a single analysis run.
STATICS	<b>SOL 200 only.</b> Specifies the FATIGUE case control is for ANALYSIS=STATICS subcases in a SOL 200 optimization run. This is the default if not specified. See Remark 9.
FREQ DFREQ MFREQ	<b>SOL 200 only.</b> Either of these specifies the FATIGUE case control is for frequency response (ANALYSIS=MFREQ or DFREQ) subcases in a SOL 200 optimization run. This is not the default and must be specified if the FATIGUE case control is referenced on a DRESP1 of type FRFTG. See Remark 9.
n	ID of FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries of the same ID (BULK option) to process, or  ID of a previously occurring SET case control entry (SET option) containing the IDs of the FTGSEQ/FTGLOAD, FTGPARM, FTGDEF bulk data entries to process (SOL 101, 103, & 112 only).

### Remarks:

1. A single FATIGUE case control is required to perform one (SOL 108/111) or more (SOL 101/103/112) fatigue analyses and must be present above the SUBCASE level. If not present, no fatigue analysis will occur regardless of the presence of other bulk data related to fatigue analysis.
2. It is not necessary to include any case control STRESS and/or STRAIN output requests in order for the FATIGUE output request to obtain the necessary stresses or strains for the fatigue calculation.
3. For a single fatigue analysis, BULK=n points to a FTGDEF, a FTGPARM, and a FTGSEQ (or FTGLOAD) entry, each with ID=n.
4. For multiple fatigue analyses, SET=n points to a previously appearing SET case control and each member of the SET is the ID of a FTGDEF, a FTGPARM, and a FTGSEQ (or FTGLOAD) bulk data entry with that ID (SOL 101, 103, or 112 only).
5. A fatigue analysis must have, at a minimum, loading and material data defined on either FTGSEQ or FTGLOAD and MATFTG bulk data entries, respectively, for a fatigue analysis to be valid. FTGPARM and FTGDEF entries can be absent, in which case, defaults will be used.
6. The following additional, optional, fatigue output file formats may be requested. The given codes must be summed if multiple files are requested. Example: CSV file and FEF file CODE would be 64+128=192. Note the following:
  - For multi-layered results, such as shells that compute top and bottom results, a separate file for each is generated for FEF formats with `_top` (or `_t`) or `_bottom` (or `_b`) designations in the name. For FEF file output, if only a single file with worst case layer results is desired, add 256 or 10,000 to the CODE. In this case no additional designation is put in the file name or `_w` is used. Using the above example would give 448 or 10192. Either mechanism works. Other designations that can appear in the filenames are `_N` or `_E` for element nodal or element center, respectively, and `_EVi` for event *i*, or `_EVS` for event summary only.
  - Note that CSV file output only ever creates a single file as both layers are reported in the single file.
  - Some output is not available for duty cycle jobs if EVNTOUT>0 on the FTGSEQ bulk data entry. Check messages if the `£06` and `log` files if the requested output files are not being generated.

File Format	CODE for SOLs 101/103/112	CODE for SOLs 108/111
No additional output	0	0
CSV File (Comma Separated - Excel File)*	64	64
FEF (Patran Results File)†	128	128

\* For random vibration fatigue using SOLs 108 or 111, the CSV file output request is also dependent on the LOGLVL field setting on the FTGPARM entry. By default a `<jid>PSD.csv` file is always generated, which at a minimum contains the input PSD, stress transfer function (TF) and result PSD for the critical entity.

†SOL 101, 103, 112 produce FEF files of type 'PATRAN 2 els' or 'PATRAN 2 nod' with .fef extension; SOL 108, 111 produce FEF files of type 'PATRAN 2 els' with .els\_fef extension. Also see Remark 11.

7. RESTARTs are possible with FATIGUE output request as long as the stress/strain state of the entities requested does not change. Most parameters on the FTGPARM (LOC excluded as it requires a different set of stresses/strains) can be changed on a RESTART. Different loading (FTGSEQ) and materials (MATFTG) can also be investigated using the RESTART capability (SOLs 101, 103, & 112 only).
8. Standard fatigue output is:

Stress-Life (SN) Analysis	Strain-Life (eN) Analysis
LIFE (Repeats*)	LIFE (Repeats)
LOG of LIFE (Repeats)	LOG of LIFE (Repeats)
LIFE (user units)†	LIFE (user units)
LOG of LIFE (user units)	LOG of LIFE (user units)
DAMAGE	DAMAGE
LOG of DAMAGE	LOG of DAMAGE
MAX STRESS‡	MAX STRESS or STRAIN **
MIN STRESS	MIN STRESS or STRAIN

\*Or Seconds for SOL 108 and SOL 111

†User units are fatigue equivalent units as defined on the FTGSEQ or FTGLOAD entry and other output is available depending on the settings of the FTGPARM entry. Please see those entries for more detail.

‡For SOLs 108 and 111 Maximum Stress = Mean Stress + MAXPEAK\*RMS Stress, Minimum Stress = Mean Stress - MAXPEAK\*RMS Stress. MAXPEAK is defined on FTGPARM entry and is defaulted to 3.0.

\*\*Depends on FTGPARM line used: STRESS or STRAIN.

9. STATICS, FREQ, DFREQ, MFREQ options are utilized in SOL 200 only. Fatigue and design sensitivity analysis will be performed for the designated analysis type only. If STATICS is specified (default), FATIGUE case control must be associated with subcases containing ANALYSIS=STATIC for a pseudo-static fatigue analysis and the fatigue DRESP1 responses/constraints specified must be of RTYPE=FATIGUE. If one of FREQ, DFREQ, MFREQ is specified, FATIGUE case control must be associated with subcases containing ANALYSIS=MFREQ or DFREQ for random vibration fatigue analysis and the fatigue DRESP1 response/constraints specified must be of RTYPE=FRFTG.
10. For SOL 108/111 with FATIGUE analysis, PARAM,SPARSEDR,no and PARAM,DDRMM,-1 both must be present. Otherwise, FATIGUE analysis will be skipped. Or, PARAM,SPARSEDR,yes, which is default, can be used instead.

11. The FEF file output request produces PATRAN 2 style elemental or nodal ASCII results files that can be imported directly into Patran for post processing. In order to import these files into Patran a template file is also required in order to tell Patran how to map the columns of data in the files to proper, meaningful labels. The analysis job produces both the FEF file and the corresponding template files. The template file has the extension `.tml` or `.res_tmpl` with the same base name as the FEF file. Jobs run with `LOC=ELEM` or `NODE` on the `FTGPARM` entry create elemental based FEF files or all SOL sequences. Jobs run with `LOC=NODA` produce nodal based FEF files for **SOLs 101, 103, and 112** and produce elemental based FEF files for **SOLs 108 and 111**. In Patran these files are imported under the **File|Import** menu with the **Object** set to **Results** and the **Format** set to either **PATRAN 2 els...** or **PATRAN 2 nod...** for elemental or nodal based FEF results files, respectively. The **Element Node Results** switch must be set to **Connectivity Based** for proper import of elemental based results.

You must use the job produced template files to import the FEF files. However, if you routinely postprocess in Patran you may wish to copy the appropriate template file to the standard template file location provided with the Patran installation. The advantage of putting it where the standard template files reside is that you may select it directly from the list without navigating the file browser to the location of the job produced template files. The disadvantage is that if you use customized fatigue equivalent units, the labels may change and not match.

12. When a **FATIGUE** case control is present to perform random vibration fatigue analysis using **SOLs 108 and 111**, only one **FREQUENCY** case control set is allowed for all transfer function **SUBCASEs** and should be above the **SUBCASE** level or only present in 1st **SUBCASE**. The **FREQUENCY** set cannot change from **SUBCASE** to **SUBCASE** or a fatal error is issued.
13. **SOL 101, 103, or 112 only**: The following additional, optional, stress or stain output may be requested. The given codes must be summed if multiple requests are required. Example: `CODE` would be `1+2=3` for fatigue stress and maximum stress range vector results. The presence of this output request overrides the `STROUT` entry on the `FTGPARM` entry. If `PLOT` is specified in the **FATIGUE** case control, no output is printed and only available in the `OUTPUT2` or `HDF5` files.

Additional Stress/Strain Output	CODE
No additional output	0
<b>Fatigue Stress/Strain</b> - these are the physical (or modal) stress/strain tensors passed to the fatigue analysis for each requested entity of the fatigue analysis for each SUBCASE (or mode). Valid for SOL 101, 103, and 112 only.	1
<b>Maximum Stress/Strain Range Magnitude or Vector</b> - this is the stress/strain maximum range magnitude (for all scalar COMB values of the FTGPARM entry) or vector (for 2D critical plane* analysis, COMB=CRITICAL) for all requested entities at the critical angle for SOLs 101, 103, and 112 only, as returned from the fatigue analysis. When LOC=ELEM or NODE on the FTGPARM entry, the computed vector is in the basic coordinate system. For LOC=NODA, the computed vector is in the SURFACE system as defined by OUTPUT field on the NAVG line of the FTGPARM entry. For scalar COMB values, a vector is not computed and only the x-component of the vector is provided. The other two components are always zero in this case.	2
<b>Stress/Strain Scalar Response Time History</b> - this is the computed scalar response time history (for all COMBs on the FTGPARM entry) at all time points as returned from the fatigue analysis for SOLs 101, 103, and 112 only. This is the actual response from which rainflow ranges and damage are computed. While individual events are processed, the ALL event is not processed when METHOD=0 on FTGPARM. Also LAYER=1 or 2 on the FTGPARM entry is honored for STROUT=4, but LAYER=0 prints values for both top and bottom. To limit output and ensure decent performance maxENTS on the FTGDEF entry is defaulted to 100 and if the number of entities exceeds this, the job stops. Setting maxENTS to a high number is not recommended due to performance issues.	4
<b>Stress/Strain Tensor Time History</b> - this is the computed tensor time history response at all time points for all requested entities used by the fatigue analysis for SOL 112 only.	8

\* COMB = CRITICAL on the FTGPARM entry.

- STROUT is not honored in manual restarts using SOL FTGRSTRT (automatic restarts are OK). STROUT=4 may use additional memory space on your system. For STROUT=4, it is also highly recommended to limit the number of requested entities on the FTGDEF bulk data entry. If the job stops because too many entities are requested, you can set maxENTS on the FTGDEF entry. STROUT=4 has the potential to produce enormous amounts of response data and external files, which can severely affect performance and disk space. STROUT=4 produces temporary CSV response files of each entity. These CSV files are deleted after the job completes unless scr=no is used when submitting the job. When METHOD=0 on the FTGSEQ entry, STROUT=4 only produces responses for each event. Only when METHOD=1 or 2 will STROUT=4 produce responses for all events combined or if there is only one event.

15. Output Data Blocks: The following table indicates the output data blocks produced by the FATIGUE case control and which parameters trigger creation of those data blocks. See the DMAP Programmers Guide for a description of these data blocks.

Data Block	Comment
OEFTG	<b>SOL 101, 103, 112 only.</b> This is the standard output data block for that is produced simply by the presence of a FATIGUE case control containing fatigue life/damage and related data for time-based fatigue analysis. Data from STROUT=2 is also written to this data block.
OEFTGM	<b>SOL 101, 103, 112 only.</b> This is the multiaxial/biaxial output data block that is produced by the presence of a FATIGUE case control with the usage of the MULTI key word on the FTGPARM entry for time-based fatigue analysis
OEFTGV	<b>SOL 108, 111 only.</b> This is the standard output data block for that is produced simply by the presence of a FATIGUE case control containing fatigue life/damage and related data for frequency-based fatigue analysis.
OES1FS	<b>SOL 101 only.</b> This is a physical stress tensor data block like OES that is created due to the presence of STROUT=1 usage in a FATIGUE case control.
OESFTGR	<b>SOL 103, 112 only.</b> This is a modal stress tensor data block like OES that is created due to the presence of STROUT=1 usage in a FATIGUE case control.
OEFTGRS	<b>SOL 101, 103, 112 only.</b> This is the scalar stress history response data block created due to the presence of STROUT=4 usage in a FATIGUE case control.
OES1FS	<b>SOL 112 only.</b> This is a tensor stress history data block like OES that is created due to the presence of STROUT=8 usage in a FATIGUE case control.



**FBODYLD**

## Free Body Load Output Request

Selects a set of submodels for which free body loads are to be produced and stored.

**Format:**

$$\text{FBODYLD(LID)} = \left\{ \begin{array}{c} \text{ALL} \\ \text{name1, name2, name3, ...} \end{array} \right\}$$

**Examples:**

```
FBODYLD=ALL
```

```
FBODYLD (100) =WINGLD
```

```
FBODYLD (200) =WINGLD, TAILLD
```

Descriptor	Meaning
LID	Optional user-defined load ID. If LID is not supplied, the subcase ID is used to define this value.
ALL	Loads will be produced for all FBODYLD Bulk Data entries.
name $i$	Name of an FBODYLD Bulk Data entry that defines the submodel to be used for the load.

**Remarks:**

1. It is recommended, but not required, that the LID be unique across subcases.
2. A separate load is created for each name $i$ .
3. The name list supplies one or more names separated by comma or blank.
4. Each load is stored individually as a one column matrix that is qualified by LID, name $i$ , submodel name, loadcase label, and submodel label (where submodel name is the name on the FBODYSB Bulk Data entry, loadcase label is the label on the FBODYLD Bulk Data entry, and submodel is the label on the FBODYSB Bulk Data entry).

## FEMCHECK

Specifies model checking options

Specifies model checking options at the start of the run. Specifically checks for RBE3 elements with unconnected independent (Gij) grids and RBE2 elements with unconnected dependent (GMi) grids. Also validates some other case controls in Frequency and Transient analysis before start of the run. RBE2 will only result in WARNING messages, and all other options will result in FATAL messages.

### Format:

$$FEMCHECK = \left\{ \begin{array}{c} NONE \\ ALL \\ RBE3, RBE2, DLOAD, FREQ, SDAMP, TSTEP \end{array} \right\}$$

### Examples:

```
FEMCHECK=ALL
FEMCHECK=RBE3, RBE2, DLOAD, FREQ, SDAMP, TSTEP
```

Describer	Meaning
NONE	No checking. This is the default.
ALL	Turns on all checking listed in the following.
RBE3	To make sure every independent (Gij) grid on a RBE3 bulk data entry is attached to an element, PLOTEL, or DMIG.
RBE2	To give a WARNING message for each dependent (GMi) grid on a RBE2 bulk data entry that is not attached to an element, PLOTEL, or DMIG.
DLOAD	For frequency analysis, check if DLOAD case control is specified. For transient analysis, check if DLOAD or IC case control is specified. Make sure DLOAD case control refers to a valid bulk data entry, i.e. DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2, ACSRCE, ACLOAD.
FREQ	Check if FREQUENCY case control is specified for frequency analysis. FREQ case control must also refer to a valid bulk data entry, i.e. FREQ, FREQ1, FREQ2, FREQ3, FREQ4, FREQ5.
SDAMP	Check if SDAMPING Case Control references a valid Bulk Data entry. It must refer to a valid bulk data entry, i.e. TABLED1, TABLED2, TABLED3, TABLED4, TABLED5, TABDMP1.
TSTEP	For SOL 108/109/111/112, TSTEP case control must refer to a TSTEP bulk data entry. For SOL 129/159, TSTEP case control should refer to a TSTEPNL bulk data entry.

### Remark:

1. This command should be applied above all SUBCASEs.

**FLSFSEL**

## Control for Fluid-Structure Frequency Selection

Control for fluid-structure frequency selection.

**Format:**

$$\begin{aligned} \text{FLSFSEL} \quad & \left[ \text{LFREQFL} = \begin{Bmatrix} 0.0 \\ fl_1 \end{Bmatrix} \right], \left[ \text{HFREQFL} = \begin{Bmatrix} 1. + 30 \\ fl_2 \end{Bmatrix} \right], \\ & \left[ \text{LFREQ} = \begin{Bmatrix} 0.0 \\ fs_1 \end{Bmatrix} \right], \left[ \text{HFREQ} = \begin{Bmatrix} 1. + 30 \\ fs_2 \end{Bmatrix} \right], \\ & \left[ \text{LMODESFL} = \begin{Bmatrix} 0 \\ mf \end{Bmatrix} \right], \left[ \text{LMODES} = \begin{Bmatrix} 0 \\ ms \end{Bmatrix} \right], \\ & \left[ \text{FLUIDSE} = \begin{Bmatrix} 0 \\ seidf \end{Bmatrix} \right] \end{aligned}$$

**Example:**

FLSFSEL            HFREQ = 4.            HFREQFL = 9

Descriptor	Meaning
LFREQFL	Requests in Hertz, lower bound frequency for modal fluid calculations.
$fl_1$	Lower frequency range for fluid, real number.
HFREQFL	Requests in Hertz, upper bound frequency for modal fluid calculations.
$fl_2$	Upper frequency range for fluid, real number.
LFREQ	Requests in Hertz, lower bound frequency for modal structure calculations.
$fs_1$	Lower frequency range for structure, real number.
HFREQ	Requests in Hertz, upper bound frequency for modal structure calculations.
$fs_2$	Upper frequency range for structure, real number
LMODESFL	Lowest modes for fluid portion of model, 0 implies LFREQFL-HFREQFL will determine number of modes.
mf	Number of lowest modes to use for fluid portion of model.
LMODES	Lowest modes for structure portion of model, 0 implies LFREQ-HFREQ will determine number of modes.
ms	Number of lowest modes to use for structure portion of model.

Describer	Meaning
FLUIDSE	Defines a specified superelement to be used for fluids only.
seidf	Defines a fluid only superelement.

**Remarks:**

1. This entry represents a collection of PARAM,name,value entries. See [Parameters](#) for detailed description of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given in [Parameters](#) of this guide.
2. If LMODES (or LMODESFL)=0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

**FLSPOUT**

## Control for Fluid-Structure Mode Participation Output

Control for fluid-structure mode participation output.

**Format:**

$$\begin{aligned}
 \text{FLSPOUT} \quad & \left[ \text{FLUIDMP} = \begin{Bmatrix} \text{ALL} \\ n_{\text{modes}} \\ \text{NONE} \end{Bmatrix} \right], \left[ \text{GRIDFMP} = \begin{Bmatrix} \text{ALL} \\ \text{setf participations} \end{Bmatrix} \right] \\
 & \left[ \text{OUTFMP} = \begin{Bmatrix} \text{ALL} \\ p_{\text{highest}} \\ \text{NOPRINT} \end{Bmatrix} \right], \left[ \text{FEPS} = \begin{Bmatrix} 1. - 11 \\ \text{epsf} \end{Bmatrix} \right], \\
 & \left[ \text{ARF} = \begin{Bmatrix} 0.95 \\ \text{arf}_v \end{Bmatrix} \right], \\
 & \left[ \text{STRUCTMP} = \begin{Bmatrix} \text{ALL} \\ m_{\text{modes}} \\ \text{NONE} \end{Bmatrix} \right], \left[ \text{OUTSMP} = \begin{Bmatrix} \text{ALL} \\ q_{\text{highest}} \\ \text{NOPRINT} \end{Bmatrix} \right] \\
 & \left[ \text{PANELMP} = \begin{Bmatrix} \text{ALL} \\ \text{setp participations} \\ \text{NONE} \end{Bmatrix} \right], \left[ \text{GRIDMP} = \begin{Bmatrix} \text{ALL} \\ \text{setg participations} \\ \text{NONE} \end{Bmatrix} \right] \\
 & \left[ \text{SEPS} = \begin{Bmatrix} 1. - 11 \\ \text{eps} \end{Bmatrix} \right], \left[ \text{ARS} = \begin{Bmatrix} 0.95 \\ \text{ars}_v \end{Bmatrix} \right] \\
 & \left[ \text{PSORT} = \begin{Bmatrix} \left( \begin{Bmatrix} \text{ABSOLUTE} \\ \text{REAL} \\ \text{IMAGINARY} \end{Bmatrix}, \begin{Bmatrix} \text{DESCENDING} \\ \text{ASCENDING} \end{Bmatrix} \right) \\ \text{O2E} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \end{Bmatrix} \right]
 \end{aligned}$$

**Examples:**

SET 23 = ROOF, DRIVERSD  
 SET 211 = 1023, 4069, 56790

FLSPOUT      FLUIDMP = 30      STRUCTMP = 40,      OUTSMP = 30,  
                  PANELMP = 23      GRIDMP = 211

Describer	Meaning
FLUIDMP	Requests fluid participation calculation of fluid response on selected fluid points.
ALL	Requests that all the fluid modes extracted be used.
n	Requests that up to the first n fluid modes be used.
NONE	Requests no participation calculation.
GRIDFMP	Requests inclusion or exclusion of specific fluid grids to be used in all the requested types of participation calculations. These are also the fluid grids that can be referred to on plot and .op2 tables.
ALL	Requests inclusion in all the requested types of the participation calculations of all fluid points.
setf	Case Control set ID listing a selected set of fluid grids to be used in all the requested types of participation calculations.
OUTFMP	Requests the FLUID FLUIDMP participation factors to be output for print.
ALL	Requests that all FLUID FLUIDMP participation factors to be output for print.
p	Requests the p highest FLUIDMP participation factors to be output.
NOPRINT	Produces tables for plotting but do not print any results.
FEPS	Filters threshold for fluid participation.
epsf	Threshold value.
ARF	Acceptance ratio for fluid participation.
arf_v	Fluid participation values $< arf\_v * \max\_value$ in a column of the output matrix will be set to zero.
STRUCTMP	Requests structural, load, and panel participation calculations on the selected fluid points. FLUIDMP must be specified for this command to become active.
ALL	Requests that all the structural modes extracted be used.
m	Requests that up to the first m structural modes be used.
NONE	Requests no participation calculation.
OUTSMP	Requests that structural STRUCTMP participation factors to be output for print.
ALL	Request that all STRUCTMP participation factors be output.
q	Requests that the q highest STRUCTMP participation factors be output.
NOPRINT	Produces tables for plotting but does not print any results.
PANELMP	Requests inclusion or exclusion of panel participation calculations on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active.

Describer	Meaning
ALL	Requests all panels defined be included in the participation calculations on the selected fluid points.
setp	Case Control set ID listing selected panels for panel participation calculations on the selected fluid points. the set consists of the character names of the panels (new V2001)
NONE	Requests exclusion from the participation calculations.
GRIDMP	Requests inclusion or exclusion of a structural panel grid participation calculation on the selected fluid points. FLUIDMP and STRUCTMP must both be specified for this command to become active.
ALL	Requests, for panels selected, that each and every individual panel grid be included as a separate calculation in the participation calculations on selected fluid points.
setg	Case Control set ID listing structural panel grids for grid mode participation on the selected fluid points.
NONE	Requests exclusion from the participation calculations.
SEPS	Filter threshold for structure participation.
epss	Threshold value.
ARS	Acceptance ratio for structure related fluid participation.
ars_v	Structure fluid participation values $< arf\_v^* \max\_value$ in a column of the output matrix will be set to zero.
PSORT	Requests type of sort.
O2E	Controls generation of tables of mode participation versus natural frequency for excitation frequencies. These tables are accessible in XYPLOT.

### Remarks:

1. This entry represents a collection of PARAM,name,value entries and must appear above the subcase level. See [Parameters](#) for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given in [Parameters](#) on this guide.
2. If  $n$ ,  $m$ ,  $p$ , or  $q$  are greater than the number computed, Nastran will invoke the ALL option for the current value.
3. PSORT values must occur in pairs such as (ABSOLUTE,DESCENDING).
4. The underlined item in the { } braces give the value of the keyword if the keyword and its describers are omitted from this entry. For example, if FLUIDMP is omitted from the FLSPOUT entry, then no fluid mode participation will be computed (unless a PARAM,FLUIDMP,value explicitly appears in a subcase or [Bulk Data Entries](#)).

## FLSTCNT

## Miscellaneous Fluid-Structure Control Parameters

Control for fluid-structure symmetry and force requests.

### Format:

$$\text{FLSTCNT} \left[ \text{ACSYM} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \right], \left[ \text{ACOUT} = \begin{Bmatrix} \text{PEAK} \\ \text{RMS} \\ \text{PEAKINT} \\ \text{RMSINT} \end{Bmatrix} \right] \\
 \left[ \text{PREFDB} = \begin{Bmatrix} \text{1.0} \\ \text{prp} \end{Bmatrix} \right], \left[ \text{ASCOUNP} = \begin{Bmatrix} \text{YES} \\ \text{NO} \end{Bmatrix} \right] \\
 \left[ \text{SKINOUT} = \begin{Bmatrix} \text{NONE} \\ \text{PUNCH} \\ \text{PRINT} \\ \text{ALL} \end{Bmatrix} \right]$$

### Example(s):

FLSTCNT      ACSYM = YES      ACOUT = RMS

Descriptor	Meaning
ACSYM	Requests symmetric or nonsymmetric solution for fluid-structure analysis.
YES	Requests symmetrized coupled fluid-structure analysis.
NO	Requests no symmetric coupled fluid-structure analysis.
ACOUT	Requests peak or rms for output to be used with the FORCE request.
PEAK, PEAKINT	Requests peak value output to be used with the FORCE request.
RMS, RMSINT	Requests rms value output to be used with the FORCE request.
PREFDB	Specifies the peak reference pressure.
prp	Value for the peak reference pressure.
ASCOUNP	Request a coupled or noncoupled fluid-structure analysis.
YES	Request a coupled fluid-structure analysis.
NO	Request a noncoupled fluid-structure analysis.
SKINOUT	Request that sets of grid point and element lists be output for both the fluid and structure at the fluid-structure interface.



Describer	Meaning
NONE	Requests no output of sets.
PUNCH	Requests set output to punch file (.pch) only.
PRINT	Requests set output to .f06 file only.
ALL	Requests set output to both .pch and .f06 files.

**Remarks:**

1. This entry represents a collection of PARAM,name,value entries. See [Parameters](#) for detailed descriptions of the parameters collected on this entry. The value of any of these parameters may be given as either the character value given in this description, or the numeric value given under the parameter description in this guide.
2. Options with suffix INT generates INTENSITY, instead of ACCLERATION. Unlike ACCELERATION which is complex, INTENSITY is a scalar quantity.

## FLUX Heat Transfer Gradient and Flux Output Request

Requests the form and type of gradient and flux output in heat transfer analysis.

### Format:

$$\text{FLUX} \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Examples:

```
FLUX=ALL
FLUX (PUNCH, PRINT) =17
FLUX=25
```

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL Flux for all elements will be output.

NONE Flux for no elements will be output.

n Set identification of a previously appearing SET command. Only fluxes of elements with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. FLUX=ALL in SOL 159 may produce excessive output.
2. FLUX=NONE overrides an overall request.

**FMETHOD**

## Flutter Analysis Method Parameter Selection

Selects the parameters to be used by the aerodynamic flutter analysis.

**Format:**

FMETHOD=n

**Example:**

FMETHOD=72

Describer	Meaning
n	Set identification number of a FLUTTER Bulk Data entry (Integer > 0).

**Remarks:**

1. An FMETHOD command is required for flutter analysis.
2. A CMETHOD command is also required for the K-method of flutter analysis.
3. If this entry is being used in SOL 200 in conjunction with flutter design conditions, the METHOD selected on the FLUTTER Bulk Data entry must be "PK" or "PKNL".

## FORCE

### Element Force Output or Particle Velocity Request

Requests the form and type of element force output, or particle velocity output, in coupled fluid-structural analysis. Note: ELFORCE is an equivalent command.

#### Format:

$$\text{FORCE} \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{l} \text{CENTER} \\ \text{CORNER or BILIN} \\ \text{SGAGE} \\ \text{CUBIC} \end{array} \right], \left[ \begin{array}{l} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right], \left[ \begin{array}{l} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \text{RPUNCH} \right) = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

#### Examples:

```
FORCE=ALL
FORCE (REAL, PUNCH, PRINT)=17
FORCE=25
FORCE (SORT2, PRINT, PSDF, CRMS, RPUNCH)=20
FORCE (PRINT, RALL, NORPRINT)=ALL
```

Descriptor	Meaning												
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.												
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
CENTER	Output CQUAD4, CQUADR, and CTRIAR element forces at the center only. The default for CQUAD4 is CENTER. The default for CQUADR and CTRIAR is CORNER.
CORNER or BILIN	Output CQUAD4, QUADR, and CTRIAR element forces at the center and at the grid points using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element forces at the center and at the grid points using strain gage approach.
CUBIC	Output CQUAD4 element forces at the center and at the grid points using cubic bending correction.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Forces for all elements will be output.
n	Set identification of a previously appearing SET command. Only forces of elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	Forces for no elements will be output.

**Remarks:**

1. ALL should not be used in a transient problem.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.

3. ELFORCE is an alternate form and is entirely equivalent to FORCE.
4. FORCE=NONE overrides an overall request.
5. If PARAM,SPARSEDR,NO is specified, then to request force output on damping elements in modal frequency response analysis (e.g., SOL 111), the mode displacement method (PARAM,DDRMM,-1) must be selected.

Force output on damping elements is not available in transient response analysis.

6. In nonlinear transient analysis, the FORCE request is ignored for nonlinear elements in SOL129. In SOL400 with "ANALYSIS=NLTRAN" Element FORCE output for the CWELD/CFAST elements is available. All other elements capable of force output such as the CBEAM, CQUAD4, etc., will not produce nonlinear transient force output.

If "PARAM, OLDWELD, YES" is specified, then the CWELD/CFAST elements will not produce with "ANALYSIS=NLTRAN" Element FORCE output.

7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE Case Control commands. Consequently, options specified in subcases other than the first subcase will be ignored. The BILIN or CORNER option is not available for advanced nonlinear elements and nonlinear material or composite elements. The default option will be applied to those elements.
  - a. If the STRESS command is specified in the first subcase, then the option on the STRESS command is used in all subcases with STRESS, STRAIN, and FORCE commands.
  - b. If the STRAIN command and no STRESS command is specified in the first subcase, then the option on the STRAIN command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
  - c. If the FORCE command and no STRESS or STRAIN commands is specified in the first subcase, then the option on the FORCE command is used in all subcases containing STRESS, STRAIN, and FORCE commands.
  - d. If STRESS, STRAIN, and FORCE commands are not specified in the first subcase, then the CENTER option is used in all subcases containing STRESS, STRAIN, and FORCE commands.
8. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
9. In general, for nonlinear elements, force output is not available. For CGAP, CELASi, or CBUSH elements in nonlinear analysis, use the STRESS in NLSTRESS command to obtain force output.
10. Forces and moment output for line elements are total forces and for shell elements they are forces per unit length.
11. The FORCE request is ignored for solid elements in all solution sequences as it has no meaning for solids.
12. Random response for element type CBEAM3 is not supported.

## FREQUENCY

## Frequency Set Selection

Selects the set of forcing frequencies to be solved in frequency response problems.

### Format:

FREQUENCY=*n*

### Example:

FREQUENCY=1 7

Describer	Meaning
<i>n</i>	Set identification number of <i>FREQ</i> , <i>FREQ1</i> , <i>FREQ2</i> , <i>FREQ3</i> , <i>FREQ4</i> , and <i>FREQ5</i> Bulk Data entries. (Integer > 0)

### Remarks:

1. A frequency set selection is required for a frequency response problem.
2. A frequency set selection is required for transient response by Fourier methods (SOL 146).
3. All *FREQ<sub>i</sub>* entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined *FREQ<sub>i</sub>* entries, respectively.

4. If there are multiple frequency response subcases, improved performance can be obtained by making the all the subcases that share the same FREQUENCY ID contiguous. For SOL 200, it is an error if there are noncontiguous subcases that have the same boundary condition and FREQUENCY ID.
5. When a FATIGUE case control is also present to perform random vibration fatigue analysis using SOLs 108 and 111, only one FREQUENCY case control set is allowed for all transfer function SUBCASEs and should be above the SUBCASE level or only present in 1st SUBCASE. The FREQUENCY set cannot change from SUBCASE to SUBCASE or a fatal error is issued.

**FRF**

Frequency Response Function (FRF) Generation and/or FRF Based Assembly (FBA) Specification

Specifies the information needed for FRF generation and/or the FBA process. SOLs 108 and 111 only.

**Format:**

$$\text{FRF} \left[ \left[ \left[ \begin{array}{c} \text{GEN} \\ \text{ASM} \\ \text{GENASM} \end{array} \right] \right] \left[ \text{COMPID} = \text{cmpid} \right] \left[ \text{COMPNAME} = \text{cmpname} \right] \right]$$

$$\left[ \text{CONNPTS} = \text{setid} \right] \text{XITOUT} = \left[ \begin{array}{c} \text{UNIT} \\ \text{UNITALL} \\ \text{USER} \\ \text{USERTOTL} \end{array} \right]$$

$$\left[ \text{ASMOUT} = \left[ \begin{array}{c} \text{COMP} \\ \text{ALL} \\ \text{ASSEMBLY} \\ \text{CONNINFO} \\ n \\ \text{cname} \end{array} \right] \right] \left[ \text{LOADLBL} = \left[ \begin{array}{c} \text{STD} \\ \text{ALT} \\ \text{ALT X} \end{array} \right] \right]$$

$$\left[ \left[ \begin{array}{c} \text{DB} \\ \text{OP2} = \text{unit} \end{array} \right] \right]$$



$$\left[ \begin{array}{l} \text{ICFGEN=} \\ \text{ICFUSE=} \\ \text{ICFAUTO=} \end{array} \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{compname} \end{array} \right\} \right] = \left\{ \begin{array}{l} \text{ICFDB} \\ \text{ICFOP2}=\{\text{icfunit}\} \end{array} \right\}$$

**Examples:**

```

FRF
FRF (COMPID = 10 COMPNAME = WING CONNPTS = 1000 LOADLBL = ALT)
FRF (COMPID = 20 COMPNAME = STRUT CONNPTS = 2000 OP2 = 25)
FRF (COMPID = 30 COMPNAME = BODY CONNPTS = 300 XITOUT = UNITALL)
FRF (ASM)
FRF (ASM LOADLBL = ALTX)
FRF (ASM ASMOUT = ALL)
FRF (GENASM COMPID = 50 COMPNAME = SUSPENSX CONNPTS = 200)
FRF (ASM ICFGEN = ALL)
FRF (ASM ICFUSE = 100)
FRF (ASM ICFGEN = 200 ICFOP2 = 33)
FRF (ASM ICFUSE = -1 ICFOP2 = 33)
FRF (ASM ICFAUTO = 100)
FRF (ASM ICFAUTO = -5 ICFOP2 = 31)

```

Descriptor	Meaning
GEN (Default)	Generate the FRFs for the specified component. See Remarks 3. and 4., and the Examples in Remark 13.
ASM	Compute the FRFs of an assembly of components from the FRFs of the individual components. See Remarks 5., 9., 10. and 11., and Examples 2 and 3 in Remark 13.
GENASM	Generate the FRFs for the specified component and follow it by computing the FRFs of an assembly of components from the FRFs of the individual components. See Remarks 4., 9., 10. and 11., and Examples 4 and 5 in Remark 13.
COMPID = <i>compid</i>	<i>compid</i> (Integer > 0) is the identification number of the component whose FRFs are to be generated. See Remarks 3. through 7., and Examples 2 through 5 in Remark 13.
COMPNAME = <i>compname</i>	<i>compname</i> (up to eight characters) is the name of the component whose FRFs are to be generated. See Remarks 3. through 7., and Examples 2 through 5 in Remark 13.

Describer	Meaning
CONNPTS = <i>setid</i>	<p><i>setid</i> (integer &gt; 0) refers to the set that defines the points at which the FRF component specified by the COMPID/COMPNAME keywords is to be connected during a subsequent FRF based assembly (FBA) process. Only those points that are defined in this set (and no others) will be considered for connection during the FBA process. See Remarks 7., 12.(c) and 12.(d), and Examples 2 through 5 in Remark 13.</p>
XITOUT = UNIT	<p>Output the FRF results only for those unit excitations that are specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control request. See Remark 8.</p> <p>The output for each of the above excitations is identified by a separate subcase. The IDs of these subcases are numbered consecutively starting from 1.</p>
XITOUT = UNITALL	<p>Output FRF results not only for unit excitations specified explicitly via FRFXIT / FRFXIT1 Bulk Data entries or implicitly via the DLOAD Case Control command, but also for unit excitations that are internally applied automatically by the program at the connection points of the FRF component(s). See Remarks 8. and 12.(c), and Example 2 in Remark 13.</p> <p>The output for each of the above excitations is identified by a separate subcase. The IDs of these subcases are numbered consecutively starting from 1.</p>

Describer	Meaning
XITOUT = USER	<p>Output the FRF results for the following excitations implied by the DLOAD Case Control request:</p> <ol style="list-style-type: none"> <li>a. A separate excitation for each individual DOF that has a non-zero load value specified for it</li> <li>b. An excitation representing the total load</li> </ol> <p>Thus, if a DLOAD Case Control request involves non-zero load values on N DOFs, then this request gives results for (N+1) excitations, with the first N such excitations representing individual and separate loads on the N DOFs and the (N+1)th excitation representing the total load. See Remark 8.</p> <p>The output for each of the above (N+1) excitations is identified by a separate coded subcase ID of the form <i>xxxxyyyy</i>. Here <i>xxxx</i> is the user subcase ID corresponding to the DLOAD under consideration. For the first N excitations, <i>yyyy</i> has values ranging from 1 through N (with leading zeros where appropriate). For the (N+1)th excitation representing the total load, the coded subcase ID is of the form <i>xxxx9999</i>.</p> <p>Because of the above coded numbering scheme, when XITOUT = USER is specified (or assumed; see Remark 8), the program will not allow any user subcase ID to exceed 9999. If it does, the program terminates the job with an appropriate fatal message.</p>
XITOUT = USERTOTL	<p>Output the FRF results for the single excitation representing the total load implied the DLOAD Case Control request. This corresponds to the (N+1)th excitation mentioned earlier. See Remark 8.</p> <p>The output for this single excitation representing the total load is identified by the coded subcase ID of the form <i>xxxx9999</i> where <i>xxxx</i> is the user subcase ID corresponding to the DLOAD under consideration.</p> <p>Because of the above coded numbering scheme, when XITOUT = USERTOTL is specified, the program will not allow any user subcase ID to exceed 9999. If it does, the program terminates the job with an appropriate fatal message.</p>
ASMOUT = CONNINFO	<p>In the FBA process, terminate the job after generating the FRF component connection information output without performing any further FRF assembly calculations.</p>
ASMOUT = COMP (Default)	<p>In the FBA process, output the FRF results for all of the individual FRF components comprising the assembly. See Remarks 10. and 11.</p>

Describer	Meaning
ASMOUT = ALL	In the FBA process, output the FRF results not only for all of the individual FRF components comprising the assembly, but also for the assembled FRF configuration considered as a separate entity. See Remarks 10. and 11., and Example 3 in Remark 13. See also Remark 2.
ASMOUT = ASSEMBLY	In the FBA process, output the FRF results only for the assembled FRF configuration considered as a separate entity. This is equivalent to specifying ASMOUT = 0 (described later). See Remarks 2. and 10.
ASMOUT = <i>n</i>	<p><i>n</i> is an integer with the following meanings:</p> <p><i>n</i> = 0</p> <p>In the FBA process, output the FRF results only for the assembled FRF configuration considered as a separate entity. This is equivalent to specifying ASMOUT = ASSEMBLY (described earlier). See Remarks 2. and 10.</p> <p><i>n</i> &gt; 0</p> <p>In the FBA process, output the FRF results only for those FRF components of the assembly whose IDs are specified by SET ID <i>n</i>.</p> <p><i>n</i> &lt; 0</p> <p>In the FBA process, output the FRF results only for that single FRF component of the assembly whose ID is given by <math> n </math>.</p>
ASMOUT = <i>cname</i>	In the FBA process, output the FRF results only for that single FRF component of the assembly whose name is given by <i>cname</i> .
LOADLBL = STD (Default)	The load labels in the output in the FRF and FBA jobs explicitly identify the grid (or scalar) point and its component where the load is applied.
LOADLBL = ALT	<p>The load labels in the output in the FRF and FBA jobs identify the grid (or scalar) point and its component where the load is applied by using the following notation:</p> <p>GGGGGGGG:+C</p> <p>where GGGGGGGG is the grid (or scalar) point ID and C indicates the component where the load is applied. C may have the following values:</p> <p>X Indicates grid point component 1 or a scalar point                      Y Indicates grid point component 2                      Z Indicates grid point component 3                      RX Indicates grid point component 4                      RY Indicates grid point component 5                      RZ Indicates grid point component 6</p>
LOADLBL = ALTX	Same as LOADLBL = ALT except that the load labels also identify whether the load applied is a unit load or a user load.

Describer	Meaning
DB (Default)	Store the FRF matrices and other information on the database. See Examples 2 and 4 in Remark 13.
OP2 = <i>unit</i>	Store the FRF matrices and other information on an OUTPUT2 file whose Fortran unit number is given by <i>unit</i> (integer > 0). See Examples 3 and 5 in Remark 13.
ICFGEN = ALL	Generate ICF information in the FBA process for all of the FRF components of the assembly
ICFGEN = n	<p>n is a non-zero integer with the following meanings: n &gt; 0</p> <p>Generate ICF information in the FBA process only for those FRF components of the assembly whose IDs are specified by SET ID</p> <p>n,</p> <p>n&lt;0</p> <p>Generate ICF information in the FBA process only for that single FRF component of the assembly whose ID is given by  n .</p>
ICFGEN = compname	FRF component whose name is given by compname.
ICFUSE = n	<p>n is a non-zero integer with the following meanings:</p> <p>n &gt; 0</p> <p>Use ICF information by employing in the FBA process a Configuration that consists of only those FRF components whose IDs are specified by SET ID n.</p> <p>n &lt; 0</p> <p>Use ICF information by employing in the FBA process a Configuration that consists of only that single FRF component whose ID is given by  n .</p>
ICFUSE = compname	Configuration that consists of only that single FRF component whose name is given by compname.

Describer	Meaning
ICFAUTO = n	n is a non-zero integer with the following meanings:  n > 0  First generate and then use ICF information in the FBA process only for those FRF components whose IDs are specified by SET  ID n.  n < 0  First generate and then use ICF information in the FBA process only for that single FRF component whose ID is given by  n .
ICFAUTO = compname	only for that single FRF component whose name is given by compname.
ICFDB (Default)	ICF information is to be stored or is resident on the database.
ICFOP2 = <i>icfunit</i>	ICF information is to be stored or is resident on an OUTPUT2 file whose Fortran unit number is given by icfunit (integer > 0).

**Remarks:**

1. This command is supported only in SOLs 108 and 111.
2. A component ID of 0 is assigned to the assembled FRF configuration resulting from the FBA process.
3. The COMPNAME keyword must be specified if the COMPID keyword is specified and vice versa.
4. If the COMPID/COMPNAME keywords are specified along with the GEN/GENASM keyword, then it implies that the FRFs computed for the specified component are employed in a subsequent FBA process. In this case, the FRF generation results will be saved on the specified medium, and the .asm (assembly punch) file will be generated and saved with a single FRFCOMP Bulk Data entry in it for subsequent use in an FBA process.
5. If the COMPID/COMPNAME keywords are not specified with the GEN/GENASM keyword, then it implies that the FRFs computed are for a single shot configuration with no subsequent FBA process involved. In this case, the GENASM keyword is equivalent to the GEN keyword.
6. The COMPID/COMPNAME keywords are ignored if the ASM keyword is specified.
7. The CONNPTS keyword must be specified if the COMPID/COMPNAME keywords are specified. It is ignored otherwise.
8. If the XITOUT keyword is not specified, the default of XITOUT = UNIT is assumed if there is no DLOAD Case Control request and the default of XITOUT = USER is assumed if there is a DLOAD Case Control request. If the user specifies XITOUT = USER or XITOUT = USERTOTL, but there is no DLOAD Case Control request, the program issues a warning message and assumes XITOUT = UNIT.

9. If the ASM/GENASM keyword is specified, the resulting FBA process will generate a connection information table in the .f06 file, indicating the relationship between the internal point IDs of the assembled FRF configuration (referred to as component 0 as indicated in Remark 2.) and the external point IDs of the associated FRF components.
10. The normal output from an FBA process run, implied by the default of ASMOUT = COMP, gives the results for the individual FRF components that comprise the assembled FRF configuration. If output is also desired for the assembled configuration as a separate entity (component 0 as indicated in Remark 2.), then ASMOUT = ALL must be specified in the FRF command to obtain the expanded output. However, in this case, the output for component 0 will be limited to displacements, velocities, and accelerations, and these will be output in terms of the internal point IDs mentioned in Remark 9. See Example 3 in Remark 13.
11. The ASMOUT keyword is ignored if the GEN keyword is specified.
12. The generation of FRFs for a component and their use in a subsequent FBA process using the FRF Case Control command involves running a standard SOL 108 or SOL 111 job, with the following additional data:
  - a. The DOFs where loads are to be applied must be specified either indirectly via the DLOAD Case Control command and/or directly via the FRFXIT/FRFXIT1 Bulk Data entries. The DLOAD Case Control command points to appropriate Bulk Data loading entries. All DOFs with nonzero load values will have unit loads applied to them. The FRFXIT entry permits specification of unit load for a single DOF with a label. The FRFXIT1 entry permits specification of unit loads at multiple DOFs.
  - b. There is no requirement that unit loading data be defined for every component for which FRFs are generated, since some components in a configuration may not have any loads applied to them.
  - c. Regardless of whether an FRF component has unit loads explicitly specified for it, as in Remark 12(a) or not, as in Remark 12(b), the program will internally apply unit loads automatically at all DOFs for all connection points comprising the set referenced by the CONNPTS keyword. This ensures that correct results are obtained from subsequent FBA processes.
  - d. The specific points at which FRFs are computed in an FRF generation run consist of the following:
    - All points specified via DISP, VELO, and ACCE requests
    - All points associated with elements for which STRESS/FORCE requests are specified
    - All points at which unit loads are applied (as per the scheme indicated in Remark 12(a))
    - All points comprising the set referenced by the CONNPTS keyword
    - All grid points referenced in PLOTTEL Bulk Data entries
  - e. It is assumed in an FBA process that the FRFs of all of the FRF components have been generated at the same forcing frequencies, and that these are also the forcing frequencies at which the FBA process is to be performed. As a result, the FBA process derives these forcing frequencies from the saved data of the first of the FRF components being assembled, and uses them in the FBA process.

In order to ensure the validity of the FBA process, the program checks to make sure that all of the FRF components have been generated using the same number of forcing frequencies and further that all of these forcing frequencies are the same for all of the FRF components. If both of these conditions are not met, the program terminates the job with an appropriate fatal message.

13. The following examples illustrate details of job setups for FRF generation and the subsequent FBA process for various scenarios.

Example 1 involves FRF generation for a single shot configuration without any FBA process.

Examples 2 through 5 assume that there are three components – 10, 20 and 30 – for which FRFs are to be generated (frgen10.dat, frgen20.dat and frgen30.dat) and that the FRFs of these components are to be subsequently assembled in an FBA process to obtain the FRFs of the assembled configuration.

Example 6 illustrates an FBA process involving the assembly of two components – 10 and 20 – whose FRFs are generated by Nastran, with a third FRF component – 40 – whose FRFs have been generated from test, with its FRF and other information resident on an Universal File (UF).

Examples 2, 3 and 6 illustrate the use of the ASM option for the FBA process while Examples 4 and 5 illustrate the use of the GENASM option for the FBA process.

Examples 7 through 13 deal with an airplane model comprising five FRF components, namely, fuselage (1), horizontal tail (2), vertical tail (3), inboard wings (4) and outboard wings (5). Example 7 illustrates the generation of the FRFs for these five components while the other examples are FBA jobs that illustrate the generation and usage of ICFs (inter component forces) for several scenarios using the ICFGEN, ICFUSE and ICFAUTO keywords of the FRF Case Control command, employing both database and OUTPUT2 usage.

Loading data must be defined for Example 1 either via FRFXIT/FRFXIT1 Bulk Data entries or via the DLOAD Case Control request. For all other examples, loading data may be defined as desired either in the FRF generation runs or in the FBA process or both.

Example 1. Generate FRFs for a Single Shot Configuration with No Subsequent FBA Process
<ul style="list-style-type: none"> <li>■ FRF Generation Job                             <ul style="list-style-type: none"> <li>• Case Control Requirement</li> </ul> </li> </ul>
FRF
Loading data must be defined either via the DLOAD Case Control command and/or via FRFXIT/FRFXIT1 Bulk Data entries.



### Example 2. Generate FRFs for Components 10, 20, and 30 Using the DB Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option

#### ■ FRF Generation Jobs

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200)

FRF component 30: FRF (COMPID = 30 COMPNAME = COMP30 CONNPTS = 300)

For these jobs, scr = no should be specified on the Nastran job command lines to ensure that the databases are saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

#### ■ FBA Job

- File Management Section (FMS) Requirement

ASSIGN dbname10 = 'frfgen10.MASTER'

ASSIGN dbname20 = 'frfgen20.MASTER'

ASSIGN dbname30 = 'frfgen30.MASTER'

DBLOCATE DATABLE = (FRFDB) LOGICAL = dbname10

DBLOCATE DATABLE = (FRFDB) LOGICAL = dbname20

DBLOCATE DATABLE = (FRFDB) LOGICAL = dbname30

(Note: All of the data blocks stored on the databases from the FRF generation runs have the same common name of FRFDB.)

- Case Control Requirement

FRF (ASM XITOUT = UNITALL)

The XITOUT = UNITALL request gives output for all unit excitations (both user specified unit loads and internally applied unit loads).

- Bulk Data Requirement

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

INCLUDE 'frfgen30.asm'

**Example 3. Generate FRFs for Components 10, 20, and 30 Using the OP2 Option and Subsequently Assemble Their FRFs to Obtain FRFs of the Assembled Configuration Using the ASM Option**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2 = 'frfgen10\_op2' UNIT = 25 DELETE

ASSIGN OUTPUT2 = 'frfgen20\_op2' UNIT = 26 DELETE

ASSIGN OUTPUT2 = 'frfgen30\_op2' UNIT = 27 DELELE

For these jobs, scr = yes may be specified on the Nastran job command lines since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

FRF component 30: FRF (COMPID = 30 COMPNAME = COMP30 CONNPTS = 300 OP2 = 27)

■ FBA Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2 = 'frfgen10\_op2' UNIT = 25

ASSIGN INPUTT2 = 'frfgen20\_op2' UNIT = 26

ASSIGN INPUTT2 = 'frfgen30\_op2' UNIT = 27

- Case Control Requirement

FRF (ASM ASMOUT = ALL)

The ASMOUT = ALL request gives output from the FBA process not only for FRF components 10, 20 and 30, but also for the assembled configuration as a separate entity (component 0) as indicated in Remark 10.

- Bulk Data Requirement

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

INCLUDE 'frfgen30.asm'

**Example 4. Generate FRFs for Components 10 and 20 Using the DB Option and Subsequently Assemble their FRFs with Those of Component 30 Using the DB and GENASM Options to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200)

For these jobs, scr = no should be specified on the Nastran job command line to ensure that the databases are saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ Combined FRF Generation and FBA Job

- File Management Section (FMS) Requirement

ASSIGN dbname10 = 'frfgen10.MASTER'

ASSIGN dbname20 = 'frfgen20.MASTER'

DBLOCATE DATBLK=(FRFDB) LOGICAL=dbname10

DBLOCATE DATBLK=(FRFDB) LOGICAL=dbname20

(Note: All data blocks stored on the databases from the FRF generation runs have the same common name of FRFDB.)

For this job, scr = no should be specified on the Nastran job command line to ensure that the database for FRF component 30 is saved for subsequent use by the FBA process.

- Case Control Requirement

FRF (GENASM COMPID = 30 COMPNAME = COMP30 CONNPTS = 300)

- Bulk Data Requirement

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

**Example 5. Generate FRFs for Components 10 and 20 Using the OP2 Option and Subsequently Assemble their FRFs with Those of Component 30 Using the OP2 and GENASM Options to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2 = 'frfgen10\_op2' UNIT = 25 DELETE

ASSIGN OUTPUT2 = 'frfgen20\_op2' UNIT = 26 DELETE

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

For these jobs, scr = yes may be specified on the Nastran job command line since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ Combined FRF Generation and FBA Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2 = 'frfgen10\_op2' UNIT = 25

ASSIGN INPUTT2 = 'frfgen20\_op2' UNIT = 26

ASSIGN OUTPUT2 = 'frfgen30\_op2' UNIT = 27 DELETE

- Case Control Requirement

FRF (GENASM COMPID = 30 COMPNAME = COMP30 CONNPTS = 300 OP2 = 27)

- Bulk Data Requirement

The following INCLUDEs are required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

**Example 6. Generate FRFs for Components 10 and 20 Using the OP2 Option and Subsequently Assemble their FRFs with Those of Test FRF Component 40 Using the ASM Option to Obtain the FRFs of the Assembled Configuration.**

■ FRF Generation Jobs

- File Management Section (FMS) Requirement

ASSIGN OUTPUT2 = 'frfgen10\_op2' UNIT = 25 DELETE

ASSIGN OUTPUT2 = 'frfgen20\_op2' UNIT = 26 DELETE

- Case Control Requirement

FRF component 10: FRF (COMPID = 10 COMPNAME = COMP10 CONNPTS = 100 OP2 = 25)

FRF component 20: FRF (COMPID = 20 COMPNAME = COMP20 CONNPTS = 200 OP2 = 26)

For these jobs, scr = yes may be specified on the Nastran job command line since there is no need for the databases to be saved at the end of the jobs.

These jobs automatically generate .asm files for subsequent use by the FBA job.

■ FBA Job

- File Management Section (FMS) Requirement

ASSIGN INPUTT2 = 'frfgen10\_op2' UNIT = 25

ASSIGN INPUTT2 = 'frfgen20\_op2' UNIT = 26

ASSIGN UNVFILE = 'frfgen40\_unv' UNIT = 28 \$ Universal File

- Case Control Requirement

FRF (ASM)

- Bulk Data Requirement

FRFCOMP,40,TSTCMP40,UF,28

The following INCLUDEs are also required.

INCLUDE 'frfgen10.asm'

INCLUDE 'frfgen20.asm'

**Example 7. Generate FRFs for Components 1 through 5 Using the OP2 Option**

- File Management Section (FMS) Requirement

```
FRF Component 1:ASSIGN OUTPUT2 = 'fuselage_op2' UNIT=25 DELETE
FRF Component 2:ASSIGN OUTPUT2 = 'hor_tail_op2' UNIT=26 DELETE
FRF Component 3:ASSIGN OUTPUT2 = 'ver_tail_op2' UNIT=27 DELETE
FRF Component 4:ASSIGN OUTPUT2 = 'ib_wings_op2' UNIT=28 DELETE
FRF Component 5:ASSIGN OUTPUT2 = 'ob_wings_op2' UNIT=29 DELETE
```

- Case Control Requirement

```
FRF Component 1:FRF (COMPID = 1 COMPNAME = FUSELAGE CONNPTS =
1000 OP2=25)
FRF Component 2:FRF (COMPID = 2 COMPNAME = HOR_TAIL CONNPTS =
1000 OP2=26)
FRF Component 3:FRF (COMPID = 3 COMPNAME = VER_TAIL CONNPTS =
1000 OP2=27)
FRF Component 4:FRF (COMPID = 4 COMPNAME = IB_WINGS CONNPTS =
1000 OP2=28)
FRF Component 5:FRF (COMPID = 5 COMPNAME = OB_WINGS CONNPTS =
1000 OP2=29)
```

These jobs automatically generate .asm files for subsequent use by the FBA process.

For these jobs, scr = yes may be specified on the Nastran job command lines since there is no need for the databases to be saved at the end of the jobs.

**Example 8. Generate ICFs on the Database for All Five FRF Components of Example 7 (Step 1 of Two-Step Process)**

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25
ASSIGN INPUTT2 = 'hor_tail_op2' UNIT=26
ASSIGN INPUTT2 = 'ver_tail_op2' UNIT=27
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29
```

- Case Control Requirement

**Example 8. Generate ICFs on the Database for All Five FRF Components of Example 7 (Step 1 of Two-Step Process)**

```
FRF (ASM ICFGEN = ALL)
```

- Bulk Data Requirement

```
INCLUDE 'fuselage.asm'
INCLUDE 'hor_tail.asm'
INCLUDE 'ver_tail.asm'
INCLUDE 'ib_wings.asm'
INCLUDE 'ob_wings.asm'
```

For this job, scr = no should be specified on the Nastran job command line since the database containing the ICF information needs to be saved for use in a subsequent FBA job (Example 9).

**Example 9. Use ICFs of Example 8 for An Assembly Configuration Consisting of FRF Components 1, 4 and 5 (Step 2 of Two-Step Process)**

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29
ASSIGN ICFDATA = 'example8.MASTER'
DBLOCATE (DATBLK = ICFDB) LOGICAL = ICFDATA
```

- Case Control Requirement

```
SET 100 = 1,4,5
FRF (ASM ICFUSE = 100)
```

- Bulk Data Requirement

```
INCLUDE 'fuselage.asm'
INCLUDE 'ib_wings.asm'
INCLUDE 'ob_wings.asm'
```

For this job, scr = yes may be specified on the Nastran job command line since there is no need for the database to be saved at the end of the job.

### Example 10. Generate ICFs on an OUTPUT2 File for FRF Components 1, 2 and 3 of Example 7 (Step 1 of Two-Step Process)

- File Management Section (FMS) Requirement

```
ASSIGN INPUTT2 = 'fuselage_op2' UNIT=25  
ASSIGN INPUTT2 = 'hor_tail_op2' UNIT=26  
ASSIGN INPUTT2 = 'ver_tail_op2' UNIT=27  
ASSIGN INPUTT2 = 'ib_wings_op2' UNIT=28  
ASSIGN INPUTT2 = 'ob_wings_op2' UNIT=29  
ASSIGN OUTPUT2 = 'icf123_op2' UNIT=33 DELETE
```

- Case Control Requirement

```
SET 100 = 1,2,3  
FRF (ASM ICFGEN = 100 ICFOP2 = 33)
```

- Bulk Data Requirement

```
INCLUDE 'fuselage.asm'  
INCLUDE 'hor_tail.asm'  
INCLUDE 'ver_tail.asm'  
INCLUDE 'ib_wings.asm'  
INCLUDE 'ob_wings.asm'
```

For this job, scr = yes may be specified on the Nastran job command line since there is no need for the database to be saved at the end of the job.



**GPFORCE**

## Grid Point Force Output Request

Requests grid point force balance at selected grid points.

**Format:**

$$\text{GPFORCE} \left( \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{l} GE \\ G \\ B \\ DAMP \\ M \\ ALLDLDS \end{array} \right] \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \end{array} \right\}$$

**Examples:**

GPFORCE=ALL

GPFORCE=17

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 files can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output. See Remark 14.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees. See Remarks 13. and 14.
GE	Element appropriate element structural GE <sub>i</sub> value contribution of force $K4 \times U_c$ output in addition to the standard elastic force output.
G	Element appropriate element structural global G value contribution of force $K \times U_c$ output in addition to the standard elastic force output.
B	Element appropriate element viscous damping contribution of force $K \times i \times \omega \times U_c$ output in addition to the standard elastic force output.

Describer	Meaning
DAMP	Element appropriate GE, B, and G will be explicitly output in addition to explicit current elastic force output.
M	Element appropriate element mass contribution of force $M \times \omega^2 \times U_c$ output in addition to the standard elastic force output.
ALLDLDS	Element appropriate GE, B, G, and M will be output in addition to the standard elastic force output.
ALL	Grid point force balance for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in the grid point force balance output (Integer >0).

### Remarks:

- The printing of the grid point forces will be suppressed if PARAM,NOGPF,-1 appears in the Bulk Data.
- The Bulk Data entry PARAM,NOELOF,+1 will cause the output of the grid point forces to be aligned with the edges of the two-dimensional elements. The default value of -1 will suppress this output. See Remark 4.
- The Bulk Data entry PARAM,NOELOP,+1 will cause the output of the sum of the forces parallel to the edges of adjacent elements. The default value of -1 will suppress this output. See Remarks 4. and 10.
- The output of grid point forces aligned with the edges of elements is available for the following elements:
  - CBAR
  - CROD
  - CBEAM
  - CSHEAR
  - CONROD
  - CTRIA3
  - CQUAD4
  - CTUBE

The positive direction for grid point forces aligned with the edges of elements is from the reference point to the load point as indicated on the printed output. See Remark 10.

- The grid point force balance is computed from linear and nonlinear elements, and includes the sum of applied loads, contact force, thermal loads, MPC forces, rigid elements, general elements, DMIG entries and SPC forces. Effects not accounted for include those from mass elements in dynamic analysis (inertia loads), slideline force contributions, and boundary loads from upstream superelements. These effects may lead to an apparent lack of equilibrium at the grid point level. The following table summarizes those effects that are considered and those effects that are ignored in the calculation of grid point forces in the global coordinate system:

Contributions Included	Contributions Ignored
Applied Loads	DMI Forces
SPC Forces	Boundary Loads from Upstream Superelements
Element Elastic Forces	Forces on elements with geometric or material nonlinear properties during normal modes analysis in SOL 106; also called linear perturbation analysis.
GENEL Forces	Connector elements CFAST, CSEAM, and CWELD.
DMIG Referenced by K2GG Case Control command	
Thermal Loads	
MPC and Rigid Element Forces	
Contact Forces	

6. Only the element elastic forces are included when the grid point forces are aligned with the edges of elements. See Remark 10.
7. In inertia relief analysis, the SPCFORCE and applied load output includes both the effect of inertial loads and applied loads.
8. When pressure loads are applied, the GPFDR module uses the discrete load vector and does not include any distributed effects.
9. GPFORCE is not available in 129. For SOLs 108 and 111, if the only frequency dependency is associated with the CBUSH, CELAS1, CELAS3, CDAMP1, CDAMP3, or (PARAM, SHLDAMP DIFF), or if any PBUSH has a GE<sub>i</sub> value other than GE1, then PARAM,BUSHNM,YES (Default) must be used. GPFORCE, for any solution sequence, does not include forces from boundary superelements. The unbalance represents the forces from adjacent superelements.
10. Grid point force output is available for nonlinear static analysis for SOLs 106 and 400. SOL106 with thermal loading does not back out the thermal load so result will not sum to zero, this is a limitation of SOL106. Contributions from slideline elements are ignored. PARAM,NOELOF and PARAM,NOELOP are not supported in nonlinear and Remarks 2., 3., 4., and 6. do not apply.
11. The usages of REPCASE and OMODES Case Control directives is not supported for ESE, EKE, EDE and GPFO output selections and should not be used.
12. The (GE, G, B, DAMP, M, and ALLDLDS) are only meaningful in SOL108, SOL111 or ANALYSIS=DFREQ or MFREQ analysis. Frequency response CYCLIC SYMMETRY ANALYSIS is not supported.

<b>Elements and features Not supported (These elements will be ignored in the GPFORCE Calculations)</b>	
ELEMENT NAME	ANALYSIS
CDUM3-7	User Defined Element
<b>Fluid</b>	
CHEXA (CHEXPR)	PSOLID with FCTN=FLUID, PORO, or PSLDSDL
CPENTA (CPENPR)	PSOLID with FCTN=FLUID, PORO, or PSLDSDL
CTETRA (CTETPR)	PSOLID with FCTN=FLUID, PORO, or PSLDSDL
CPYRAM (CPYRPR)	PSOLID with FCTN=FLUID, PORO, or PSLDSDL
CAABSF	Acoustic Absorber Element
<b>Cyclic Symmetry</b>	
CTRIAX6	
<b>SOL400 elements associated with PLPLANE</b>	
QUAD4FD, QUADFD, QUADX4FD, QUADXFD, TRIA3FD, TRIAFD, TRIA3FD, TRIAXFD	Plane strain elements
<b>Structures</b>	
CBUSH2D	Rotor Dynamic element
CHACAB	Acoustic Absorber element
CHACBR	Acoustic Barrier Element
Unsupported elements will appear in the GPFORCE output with incomplete load calculations. Thus, grids associated with them, in general, will not sum to zero.	

13. In general, for frequency response, the “\*totals\*” rows will not be zero, except when ALLDLDS is chosen. Also if “PHASE” is also chosen, only the first row of “\*TOTALS\*” will be zero.

SOL111 and SOL200/SOL400 with ANALYSIS=MFREQ, by their nature, introduce noise into the solution. As a result, some “\*totals\*” may have poor zeros. This is especially true when heavy structural damping is present. The reason is that for structural damping, residual vectors are not computed.

For frequency dependent elements using master frequencies, for forcing frequencies not at the master frequencies, the interpolation for the solution vector is done on the KGG, K4GG, etc., matrices; while, for GPFORCE, the interpolations are done on the individual element stiffness matrices and the individual values of GE appearing on the MATiF entries. As a result, for an interpolated result, far from its bounding master frequencies, with ALLDLDS, the totals may be several orders higher than results when the interpolated results have tighter bounding master frequencies.

For frequency dependent elements, the F-OF-MPC, that include both Nastran MPC entry results and RIGID element results, may be in error. This will cause the “\*totals\*” to show very large values.

To avoid this, it is recommended that these types of constraints do not touch grids associated with frequency dependent elements. The reason for this, is that the MPC/RBEi forces of constraint are currently based on the nominal values of element properties.

14. For PRINT and PUNCH options, when PHASE is chosen, the F06 and PCH files will be in magnitude-phase format. However, in any analysis that involves complex numbers, any and all of the HDF5 output is automatically converted to real-imaginary format.
15. For frequency response, the GPFFORCE calculations are:

#### GPFFORCE Calculations

$F$  = KELM  $\times$  (UGR+UGI) - standard elastic force contribution at the grid

$F_{GE}$  =  $i \times$  GEVAL  $\times$  GEFACT  $\times$  KELM4  $\times$  (UGR+UGI) - structural damping force contribution at the grid

$F_B$  = BELM  $\times i \times \omega \times$  (UGR+UGI) - viscous damping force contribution at the grid

$F_G$  =  $i \times G \times$  KELM  $\times$  (UGR+UGI) - global G/GF - global damping force contribution at the grid

$F_M$  = WTMASS  $\times$  MELM  $\times \omega^2 \times$  (UGR + UGI) - inertia force contribution at the grid.

KELM - an element stiffness matrix

KELM4 - an element structural damping matrix

BELM - an element viscous damping matrix

MELM - an element mass matrix

UGR - real displacement at the grid

UGI - imaginary displacement at the grid

$\omega = 2 \pi f$  where  $f$  is the current forcing frequency.

$i = \sqrt{-1}$

### GPFORCE Calculations

Depending on the element type, GEVAL is determined from a MATi entry or a PROPERTY entry. For elements referring to extended damping on MAT2 or MAT9 entries, GEVAL=1.0 and the KELM4 contains the cumulative results.

When frequency dependent material or properties are evoked, when a frequency is between master frequencies the element forces will be interpolated based on MFREQUENCY case control, LINEAR or LOG10.

The various loads appear in the SOURCE column of the GPFORCE output as:

ELEMENT NAME	Corresponds to F the standard elastic force contribution at the grid.
F-OF-G	Corresponds to $F_G$ the global damping force contribution at the grid.
F-OF-GE	Corresponds to $F_{GE}$ the structural damping force contribution at the grid.
F-OF-B	Corresponds to $F_B$ the viscous damping force contribution at the grid.
F-OF-M	Corresponds to $F_M$ the inertia force contribution at the grid.
Special "Elements"	
GENEL	Corresponds to Non-frequency dependent standard elastic force contribution at the grid.
K2GG	Corresponds to Non-frequency dependent standard elastic force contribution at the grid.
K42GG	Corresponds to Non-frequency dependent F-OF-GE at the grid.
B2GG	Corresponds to Non-frequency dependent F-OF-B at the grid.
M2GG	Corresponds to Non-frequency dependent F-OF-M at the grid.

The above user supplied matrices are displayed using the user supplied naming convention.

#### SDAMP, RSDAMP, and Rayleigh damping

F-OF-SDP	Corresponds to Non-forcing frequency dependent Modal damping at the grid. PARAM, KDAMP, 1 (Default)
F-OF-K4D	Corresponds to Non-forcing frequency dependent Modal damping at the grid. PARAM, KDAMP, -1
F-OF-RAY	Corresponds to Non-forcing frequency dependent Rayleigh damping at the grid.

GEFACT (see Bulk Data entry DAMPING, default=1.0) effect appears as a scaling to F-OF-GE

F-OF-SDP, F-OF-K4D, and F-OF-RAY output is controlled by the appearance of the GPFORCE key words: GE, DAMP or ALLDLDS.

The remaining source entries: APP-LOAD, F-OF-SPC, F-OF-MPC remain the same as in statics.

**GPKE**

## Grid Point Kinetic Energy Output Request

Requests the output of the kinetic energy at selected grid points in normal modes analysis only.

**Format:**

$$\text{GPKE} \left[ \left( \begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right), \text{PUNCH}, \text{THRESH} = e \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

**Examples:**

GPKE=ALL

GPKE (PRINT, PUNCH)=19

Descriptor	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generates, but does not print, grid point kinetic energy output.
PUNCH	The punch file will be the output medium.
e	Minimum energy threshold. Only energies above this value will be printed and/or punched.
ALL	Grid point kinetic energy for all grid points will be output.
n	Set identification number of a previously appearing SET command. Only grid points with identification numbers that appear on this SET command will be included in output (Integer > 0).
NONE	Grid point kinetic energy for no points will be output.

**Remarks:**

1. Grid point kinetic energy is only available for normal modes analysis.
2. Both PRINT and PUNCH may be requested.
3. GPKE=NONE overrides an overall output request.
4. For models using the lumped mass formulation, the grid point kinetic energy can be used to examine the distribution of kinetic energy among the grid points. It is computed as:

$$E_{k_g} = \Phi_g^{mass} \otimes [M_{gg} \Phi_g^{mass}]$$

where  $\Phi_g^{mass}$  represents the mass-normalized eigenvectors so that the total grid point kinetic energy is scaled to be unity. Note that the operator  $\otimes$  indicates term-wise matrix multiplication.

5. The grid point kinetic energy output has limited meaning for a coupled mass formulation. Since this mass formulation produces a coupling of mass across grid points, the sharing of kinetic energy among grid points can occur. In general, this obscures the meaning of the computation as a means of identifying important model parameters to control modal behavior.



**GPRSORT**

## Composites Ply Results Sorted Output

Request sorted output of composites ply results (stress, strain, and failure indices) by global ply ID for a given element set.

**Format:**

$$\text{GPRSORT} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

**Examples:**

```
GPRSORT=ALL
GPRSORT=22
```

Describer	Meaning
ALL	All composite elements referencing a PCOMPG property entry type. See Remarks <a href="#">1.</a> and <a href="#">2.</a>
n	Set identification number of a previously appearing SET command.

**Remarks:**

1. Composite element output will be sorted by global ply ID and element ID. Note that this sorted output is only available for composite elements referencing a PCOMPG property entry. Global ply IDs can only be specified on the PCOMPG entry.
2. Composite elements referencing the PCOMP property entry will be excluded from the sorted output.

## GPSDCON

## Grid Point Stress Discontinuity Output Request

Requests mesh stress discontinuities based on grid point stresses.

### Format:

$$\text{GPSDCON} \begin{bmatrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{bmatrix} = \begin{Bmatrix} \text{ALL} \\ \text{n} \end{Bmatrix}$$

### Examples:

GPSDCON=ALL  
GPSDCON=19

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Stress discontinuity requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the stress discontinuity output request (Integer > 0).
NONE	No grid point stress discontinuity output.

### Remarks:

1. This output is available in linear static analysis SOLs 101 and 144 only. Output will be presented for each surface or volume as a tabular listing of stress discontinuities for each subcase.
2. Only elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.
3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands. Also, the GPSTRESS and STRFIELD commands must be present for printed output.

**GPSTRAIN**

## Grid Point Strain Output Request for Printing Only

Requests grid point strains for printing only.

**Format:**

$$\text{GPSTRAIN} \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

**Examples:**

```
GPSTRAIN=ALL
GPSTRAIN=19
```

Descriptor	Meaning												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Grid point strain requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point strain output request (Integer > 0).
NONE	No grid point strain output.

**Remarks:**

1. For statics, normal modes, and transient analysis, output will be presented for each surface or volume as a tabular listing of grid point strains for each load, eigenvalue, and time step. (See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.)
2. Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.

3. Element strain output (STRAIN) must be requested for elements referenced on SURFACE and VOLUME commands.
4. In nonlinear transient analysis, grid point strains are computed only if parameter LGDISP is -1, which is the default, and only for elements with linear material properties.
5. For the postprocessing of grid point strains using the .xdb file or the computation of mesh strain discontinuities, the STRFIELD command must also be specified.

**GPSTRESS**

## Grid Point Stress Output Request for Printing Only

Requests grid point stresses for printing only.

**Format:**

$$\text{GPSTRESS} \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

GPSTRESS=ALL  
GPSTRESS=19

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Grid point stress requests for all SURFACE and VOLUME commands defined in the OUTPUT(POST) Section will be output.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command will be included in the grid point stress output request (Integer > 0).
NONE	No grid point stress output.

**Remarks:**

1. For statics, normal modes, and transient analysis, output will be presented for each surface or volume as a tabular listing of grid point stresses for each load, eigenvalue, and timestep. (See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.)
2. Only grid points connected to elements used to define the surface or volume are output. See the description of the SURFACE or VOLUME commands.

3. Element stress output (STRESS) must be requested for elements referenced on SURFACE and VOLUME commands.
4. For the postprocessing of grid point stresses using the .xdb file or the computation of mesh stress discontinuities, the STRFIELD command must also be specified.
5. Grid point stress is not output for midside nodes.

**GROUNDCHECK**

## Rigid Body Motion Grounding Check

Perform grounding check analysis on the stiffness matrix to expose unintentional constraints by moving the model rigidly.

**Format:**

$$\text{GROUNDCHECK} \left[ \left( \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right), \text{PUNCH, SET} = \left( \begin{array}{l} \text{G, N, N + AUTOSPC, F, A} \\ \text{ALL} \end{array} \right) \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

$$\text{GRID} = \text{gid, THRESH} = e,$$

$$\text{DATAREC} = \left[ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right], (\text{RTHRESH} = r)$$

**Examples:**

GROUNDCHECK=YES

GROUNDCHECK (GRID=12, SET=(G, N, A), THRESH=1.E-5, DATAREC=YES) =YES

Descriptor	Meaning
PRINT	Write output to the print file. (Default)
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
SET	Selects degree-of-freedom set(s) (Default: SET=G).
gid	Reference grid point for the calculation of the rigid body motion.
e	Maximum strain energy which passes the check. The default value is computed by dividing the largest term in the stiffness matrix by 1.E10.
DATAREC	Requests data recovery of grounding forces (Default: DATAREC=NO).
r	Grounding forces which are larger than r percent of the largest grounding force will be printed if DATAREC=YES (Default = .10; 0. < r < 1.0).

**Remarks:**

- GROUNDCHECK must be specified above the subcase level.
- SET=N+AUTOSPC (check the N-set stiffness, including the effect of PARAM,AUTOSPC) uses the stiffness matrix for the n-set, with the rows corresponding to degrees-of-freedom constrained by the PARAM,AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.
- If DATAREC=YES, GROUNDCHECK FORCES will be printed in the displacement coordinate system of the associated GRID points.

4. For CBEAM/CBEAM3, it is recommended to use SPOINT ID for warping DOFs. Use of GRID ID for warping DOFs may cause one or more directions to fail the rigid body check.
5. If Lagrange multipliers are present via RIGID=LAGRAN or LGELIM then for degree-of-freedom sets N, N+AUTOSPC, F, and A the check will be performed on degree-of-freedom sets NL, NL+AUTOSPC, FL, and AL. The output will also be labeled accordingly.



**GUST**

## Aerodynamic Gust Load Requests

Selects the gust field in an aeroelastic response problem.

**Format:**

GUST=n

**Example:**

GUST=73

Describer	Meaning
n	Set identification of a GUST Bulk Data entry (Integer > 0).

**Remark:**

1. The choice of transient or frequency response GUST depends upon the type of TLOAD or RLOAD referenced on the selected GUST entry.

## GVECTOR

### Eigenvector Output Request in SOLs 200 and 400

Requests the form and type of eigenvector output in SOLs 200 and 400. For other solutions use [DISPLACEMENT \(Case\)](#)

#### Format:

$$GVECTOR \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

#### Examples:

GVECTOR=ALL  
GVECTOR (PUNCH) =NONE

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Eigenvectors for all points will be output.
NONE	Eigenvectors for no points will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0).

#### Remarks:

1. GVECTOR=NONE overrides an overall output request.
2. Output will be presented as a tabular listing of grid points for each eigenvector.
3. GVECTOR is intended to specified in subcases with ANALYSIS=MFREQ, MTRAN, MCEIG, FLUTTER, and SAERO.

**HADAPT**

## Mesh Adaptivity Activation and Control

Specifies Mesh adaptivity control parameters in SOLs 101 and 400.

**Format:**

HADAPT=N

**Example:**

HADAPT=1

Describer	Meaning
N	Identification number for a HADAPTL Bulk Data entry (Integer > 0).

**Remarks:**

1. The HADAPT command can be used only in SOL 101 or SOL 400 with ANALYSIS=STATICS.
2. In SOL 101, with NO SUPERELEMENTS, the HADAPT command may appear either above all SUBCASEs or within specific SUBCASEs. In the last scenario, only stresses on the solution corresponding to the specific SUBCASE where the HADAPT command has been placed will be used to compute error indicators should the user requests an error indicator based refinement criterion (see Bulk Data entries, [HADACRI](#) and [HADAPTL](#)). If superelements are present, then the HADAPT command should appear only in Residual Superelement (SE0) SUBCASE structure.
3. In SOL 400, the HADAPT command can only be placed on a Linear static structural analysis SUBCASE (ANALYSIS=STATICS) either above all STEPS or within each single STEP. All STEPs must be Linear Static structural STEPs (ANALYSIS=STATICS). In other words an adaptive meshing linear analysis cannot be chained with any other analysis type.
4. HADAPT remeshing should only be performed on lower order elements.

## HARMONICS

### Harmonic Analysis or Printout Control

Controls the number of harmonics output in axisymmetric shell or axisymmetric fluid problems; controls the number of harmonics to be used for analysis in cyclic symmetry problems.

#### Format for Axisymmetric Problems:

$$\text{HARMONICS} = \left\{ \begin{array}{c} \text{ALL} \\ \text{NONE} \\ \text{h} \end{array} \right\}$$

#### Format for Cyclic Symmetric Problems:

$$\text{HARMONICS} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

#### Examples:

HARMONICS=ALL  
 HARMONICS=32

Descriptor	Meaning
ALL	All harmonics will be output in the case of axisymmetric shell or axisymmetric fluid problems. All harmonics will be used for analysis in cyclic symmetry problems.
NONE	No harmonics will be output. This option is not available for use in cyclic symmetry problems.
h	Available harmonics up to and including harmonic h will be output in the case of axisymmetric shell or axisymmetric fluid problems (Integer $\geq 0$ ).
n	Harmonics specified in SET n will be used for analysis in cyclic symmetry problems (Integer $> 0$ ).

#### Remarks:

1. If no HARMONICS command is present in the Case Control Section for axisymmetric shell or fluid problems, printed output is prepared only for the zero harmonic.
2. This command must be present in cyclic symmetry problems with HARMONICS=ALL or n; otherwise, the program will abort without performing any analysis.
3. In cyclic symmetry analysis, n must be defined as a set of integers on a SET command.

**HDOT****Heat Transfer Rate of Change of Enthalpy Output Request**

Requests form of rate of change of enthalpy vector output in transient heat transfer analysis (SOL 159).

**Format:**

$$\text{HDOT} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

HDOT=5

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each time.
SORT2	Output will be presented as a tabular listing of time for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Rate of change of enthalpy for all points will be output.
NONE	Rate of change of enthalpy for no points will be output.
n	Set identification of previously appearing SET command. Only rates of change of enthalpy for points with identification numbers that appear on this SET command will be output (Integer > 0).

**Remark:**

1. HDOT=NONE is used to override a previous HDOT=n or HDOT=ALL command.

## HOUTPUT

## Harmonic Output Request in Cyclic Symmetry Problems

Requests harmonic output in cyclic symmetry problems.

### Format:

$$\text{HOUTPUT}[(C, S, C^*, S^*)] = \left\{ \begin{array}{c} \text{ALL} \\ K \end{array} \right\}$$

### Examples:

```
HOUTPUT=ALL
HOUTPUT (C, S) =5
```

Describer	Meaning
C, S, C*, S*	Harmonic coefficients. See Remark 4.
ALL	All harmonics will be output.
k	Set identification number of harmonics for output (Integer > 0).

### Remarks:

1. Set k must be defined on a SET command, and output will be computed for all available harmonics in SET k.
2. HOUTPUT=ALL requests output for all harmonics specified on the HARMONICS command.
3. Either the HOUTPUT or NOUTPUT command is required to obtain data recovery in cyclic symmetry analysis.
4. C and S correspond to the cosine and sine coefficients when the STYPE field is ROT or AXI on the CYSYM Bulk Data entry.

C, S, C\*, and S\* correspond to the cosine symmetric, sine symmetric, cosine antisymmetric, and sine antisymmetric coefficients, respectively, when the STYPE field is DIH on the CYSYM Bulk Data entry.

## HTFLOW

## Elemental Heat Flow Output Request

Requests heat flow output at selected structural elements.

### Format:

$$\text{HTFLOW} \left[ \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{NOPRINT} \end{array} \right) \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

### Example:

```
HTFLOW = ALL
HTFLOW = 15
```

Describer	Meaning
PRINT	The printer will be the output medium.
NOPRINT	Generate, but do not print out, the output.
PUNCH	The punch file will be the output medium.
ALL	Heat flow for all structural elements will be output.
n	Set identification of previously appearing SET command. Only structural elements with identification numbers that appear on this SET command will be included in the heat flow output (Integer > 0).

### Remarks:

1. Elemental heat flow output is available for steady state thermal analysis (SOL 101 and SOL 153) and transient thermal analysis (SOL 159).
2. Heat flow is computed from the applied heat loads and the effect of convection and radiation heat transfer on boundary elements (CHBDYE, CHBDYG, and CHBDYP).
3. See Remarks 6.-8. of the descriptions of CHBDYE Bulk Data for the side conventions of solid elements, shell elements, and line elements.

## IC Transient Analysis Initial Condition Set Selection

Selects the initial conditions for transient analysis (SOLs 109, 112, 129, 159, 400 and 600).

### Format:

$$IC \begin{bmatrix} \text{PHYSICAL} \\ \text{MODAL} \\ \text{STATSUB[,DIFFK]} \end{bmatrix} = n$$

### Examples:

```
IC = 10
IC (PHYSICAL) = 100
IC (MODAL) = 200
IC (STATSUB) = 1000
IC (STATSUB, DIFFK) = 2000
```

Descriptor	Meaning
PHYSICAL	The TIC Bulk Data entries selected by set n define initial conditions for coordinates involving grid, scalar, and extra points (Default).
MODAL	The TIC Bulk Data entries selected by set n define initial conditions for modal coordinates and extra points. See Remark 3.
STATSUB	Use the solution of the static analysis subcase n as the initial condition. See Remark 4.
DIFFK	Include the effects of differential stiffness in the solution. See Remarks 4. and 5.
n	For the PHYSICAL (Default) and MODAL options, n is the set identification number of TIC Bulk Data entries for structural analysis (SOLs 109, 112, 129, and 600) or TEMP and TEMPD entries for heat transfer analysis (SOLs 159, 400 and 600). For the STATSUB option, n is the ID of a static analysis subcase (Integer > 0).

### Remarks:

1. For structural analysis, TIC entries will not be used (therefore, no initial conditions) unless selected in the Case Control Section.
2. Only the PHYSICAL option (Default) may be specified in heat transfer analysis (SOLs 159, 400 and 600). The initial temperature of a point should be equal to its boundary temperature at time = 0. For SOL 400 transient thermal analysis, the boundary temperature at time = 0 takes precedence over the initial temperature if both temperatures are not identical.
3. IC(MODAL) may be specified only in modal transient analysis (SOL 112).
4. IC(STATSUB) and IC(STATSUB,DIFFK) cannot be specified together in the same run. This will cause FATAL and stop the execution.
5. The DIFFK keyword is meaningful only when used in conjunction with the STATSUB keyword.
6. The following examples illustrate the usage of the various options of the IC Case Control command.



**Example (a)**

```
$ SPECIFY INITIAL CONDITIONS FOR PHYSICAL COORDINATES  
$ IN SOL 109 OR SOL 112  
IC(PHYSICAL) = 100
```

or

```
IC = 100
```

**Example (b)**

```
$ SPECIFY INITIAL CONDITIONS FOR MODAL COORDINATES  
$ IN SOL 112  
IC(MODAL) = 200
```

**Example (c)**

```
$ SPECIFY STATIC SOLUTION AS INITIAL CONDITION  
$ IN SOL 109 OR SOL 112  
$ (DIFFERENTIAL STIFFNESS EFFECT NOT INCLUDED)  
SUBCASE 10 $ STATIC ANALYSIS  
LOAD = 100  
SUBCASE 20 $ TRANSIENT ANALYSIS  
IC(STATSUB) = 10 $ POINTS TO STATIC ANALYSIS SUBCASE ID
```

**Example (d)**

```
$ SPECIFY STATIC SOLUTION AS INITIAL CONDITION  
$ IN SOL 109 OR SOL 112  
$ (DIFFERENTIAL STIFFNESS EFFECT INCLUDED)  
SUBCASE 100 $ STATIC ANALYSIS  
LOAD = 1000  
SUBCASE 200 $ TRANSIENT ANALYSIS  
IC(STATSUB,DIFFK) = 100 $ POINTS TO STATIC ANALYSIS SUBCASE ID
```

## ICF Inter-Component Forces (ICF) Output Request

Requests the form and type of inter-component force (ICF) vector output from an FBA job.

### Format:

$$\text{ICF} \left[ \left( \left[ \begin{array}{l} \text{PHYSICAL} \\ \text{MODAL} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] \right) \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

### Examples:

```
ICF = 5
ICF (SORT2, PRINT, PUNCH, IMAG) = ALL
ICF (SORT2) = 20
ICF = ALL
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each frequency. See Remarks 1. through 4.
SORT2	Output will be presented as a tabular listing of frequency for each grid point. See Remarks 1., 3. and 5.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
- PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
- ALL ICFs for all points will be output. See Remarks 1. through 5.

Describer	Meaning
n	Set identification of a previously appearing SET command. Only ICFs for points with identification numbers that appear on this SET command will be output (Integer > 0). See Remarks 1. through 6.
NONE	ICFs for no points will be output.

**Remarks:**

1. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In SORT1 format, ICFs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
3. ICFs are defined only for the connection points of FRF components in an FBA job and are regarded as null for non-connection points of FRF components. Therefore, ICF output is meaningful only for the connection points of FRF components in an FBA job.
4. In SORT1 format, ICF output is ignored for non-connection points and will be output for the requested connection points of FRF components only if their ICFs are non-null.
5. In SORT2 format, ICF output is honored for all requested connection and non-connection points of FRF components even if their ICFs are null.
6. ICF output request is ignored for undefined points.

## IMPERFECT

### Select Geometric Imperfection

Select geometric imperfection cases in SOL 400.

#### Format:

```
IMPERFECT = n
```

#### Example:

```
IMPERFECT = 5
```

Descriptor	Meaning
------------	---------

n	Identification number of an IMPGEOM or IMPCASE bulk data entry.
---	---

#### Remark:

1. This command must be above all subcases.

**INCLUDE**

## Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

```
INCLUDE 'filename'
```

**Example:**

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYCASE.DATA:

```
SOL 101
CEND
TITLE = STATIC ANALYSIS
LOAD = 100
INCLUDE 'MYCASE.DATA'
BEGIN BULK
ENDDATA
```

Describer	Meaning
filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks (').

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example, the file:

```
/dir123/dir456/dir789/filename.dat
```

may be included with the following input:

```
INCLUDE '/dir123
           /dir456
           /dir789/filename.dat'
```

3. See the for more examples.

## INTENSITY

## Acoustic Intensity Output Request

Requests output of acoustic intensity on wetted surface. SOLs 108 and 111 only.

### Format:

$$\text{INTENSITY} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{c} \text{PRINT,PUNCH} \\ \text{PLOT} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

Describer	Meaning
SORT1	Output will be presented as tabular listing of grid points for each excitation frequency (Default).
SORT2	Output will be presented as tabular listing of excitation frequencies for each grid point.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Intensities will be computed for all grid points of the wetted surface.
n	Set identification of a previously defined set of grid points. Intensities will be computed for the grid points in this set only.
NONE	Acoustic intensities will not be processed.

### Remarks:

1. INTENSITY = NONE overrides an overall request.
2. This Case Control command can be used in SOL 108 and SOL 111 only.

**IRLOAD**

## Nonlinear Inertia Relief Selection (SOL 400 only)

Selects nonlinear inertia relief set for SOL 400

**Format:** IRLOAD={ QLINEAR  
NONE }

**Example:**

IRLOAD=QLINEAR

Describer	Meaning
QLINEAR	Inertia Load Calculation with small displacement (Quasi-Linear) is activated in SOL 400.
NONE	No Inertia Relief (Default)

**Remark:**

1. This command is active only in SOL 400 with ANALYSIS=NLSTATIC and its use requires a set of STATIC supports that constrain all six rigid body motions.
2. IRLOAD=QLINEAR, which has to be applied above to all SUBCASES, is a global case control command and activates the inertia load calculations in SOL 400 for all applied static loads. In nonlinear static analyses (ANALYSIS=NLSTAT), it also activates the inertia relief analysis with small displacement. When IRLOAD=QLINEAR with large displacement (PARAM,LGDISP,1), a fatal error message will be issued. Also superelements in conjunction with IRLOAD=QLINEAR will cause a fatal error.  
IRLOAD=NONE (default) deactivates the inertia load calculations.
3. IRLOAD=QLINEAR is ignored by perturbation analyses in SOL 400.

**K2GG**

## Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrix or matrices.

**Format:**

K2GG=name

**Example:**

```
K2GG = KDMIG
K2GG = KDMIG1, KDMIG2, KDMIG3
K2GG = 1.25*KDMIG1, 1.0*KDMIG2, 0.82*KDMIG3
SET 100 = K2, K3, K4
K2GG = 100
```

Describer	Meaning
name	Name of a [ $K_{gg}^2$ ] matrix that is input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 6. (Character).

**Remarks:**

- DMIG matrices will not be used unless selected.
- Terms are added to the stiffness matrix before any constraints are applied.
- The matrix must be symmetric and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
- A scale factor may be applied to this input using the PARAM, CK2 entry. See [Parameters](#).
- The matrices are additive if multiple matrices are referenced on the K2GG command.
- The formats of the name list:
  - Names without factor.  
Names separated by comma or blank.
  - Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
- If there are multiple subcases in the analysis, the K2GG command must appear above the first subcase or in the first subcase. K2GG requests in the second and subsequent subcases will be ignored. For superelements, it should occur in the first subcase for the appropriate superelement.



**K2PP**

## Direct Input Stiffness Matrix Selection

Selects direct input stiffness matrix or matrices, which are not included in normal modes.

**Format:**

K2PP=name

**Example:**

```
K2PP = KDMIG
K2PP = KDMIG1, KDMIG2, KDMIG3
K2PP = 5.06*KDMIG1, 1.0*KDMIG2, 0.85*KDMIG3
K2PP = (1.25, 0.5) *KDMIG1, (1.0,0.0) *KDMIG2, (0.82, -2.2) *KDMIG3
```

Describer	Meaning
name	Name of a $[K_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors. See Remark 6.

**Remarks:**

1. DMIG and DMIAX entries will not be used unless selected by the K2PP command.
2. The matrix must be square or symmetric, and field 4 on the DMIG,name Bulk Data entry must contain a 1 or 6.
3. It is recommended that PARAM,AUTOSPC,NO be specified. See the [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
4. K2PP matrices are used only in dynamic response problems. They are not used in normal modes.
5. The matrices are additive if multiple matrices are referenced on the K2PP Case Control command.
6. The formats of the name list:
  - a. Names without factor  
Names separated by comma or blank.
  - b. Names with factors.

Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers separated by a comma, within parentheses, as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be paired with a factor including 1.0 for real and (1.0, 0.0) for complex.

## K42GG

## Direct Input Stiffness Element Damping Selection

Selects direct input structural element damping matrix or matrices.

### Format:

K42GG=name

### Example:

K42GG = KDMIG

K42GG = KDMIG1, KDMIG2, KDMIG3

K42GG = 2.03\*KDMIG1, 0.84\*KDMIG2

Descriptor	Meaning
name	Name of a $[K_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 4.

### Remarks:

1. DMIG matrices will not be used unless selected.
2. Terms are added to the structural element damping matrix before any constraints are applied.
3. The matrix must be symmetric, and field 4 on the DMIG,name Bulk Data entry must contain the integer 6.
4. The formats of the name list:
  - a. Names without factor.  
Names separated by comma or blank.
  - b. Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
5. If there are multiple subcases in the analysis, the K42GG command must appear above the first subcase or in the first subcase. K42GG requests in the second and subsequent subcases will be ignored. For superelements, it should occur in the first subcase for the appropriate superelement.

**LABEL**                      Output Label

Defines a character string that will appear on the third heading line of each page of printer output.

**Format:**

LABEL=label

**Example:**

LABEL=DEMONSTRATION PROBLEM

Describer	Meaning
label	Any character string.

**Remarks:**

1. LABEL appearing at the subcase level will label output for that subcase only.
2. LABEL appearing before all subcases will label any outputs that are not subcase-dependent.
3. If no LABEL command is supplied, the label line will be blank.
4. LABEL information is also placed on plotter output, as applicable. Only the first 65 characters will appear.

## LDLABEL

Label assigned to a subcase (static load case) in the external superelement creation run

Assigns a character string to an external superelement creation run's subcase in order to identify the static load in the assembly run.

### Format:

```
LDLABEL [=load_label]
```

### Example:

```
LDLABEL  
LDLABEL=GRAVITY LOAD ON THE OUTBOARD NACELLE
```

Describer	Meaning
load_label	Any character string.

### Remarks:

1. The LDLABEL command is used only when the EXTSEOUT command is specified.
2. If the LDLABEL is specified in any subcase but not all subcases then the following defaults will be applied to those subcases without LDLABEL in the following order:
  - a. LABEL inside the subcase
  - b. If no LABEL, then SUBTITLE inside the subcase
  - c. If no SUBTITLE, then TITLE inside the subcaseIf it is desired that the default logic is to be applied to all subcases then specify LDLABEL with no argument above all subcases.
3. If the same load\_label is specified in more than one SUBCASE then a User Warning Message will be issued.
4. Load\_label may be referenced on the [SELOAD](#) Bulk Data entry.

**LINE****Maximum Lines Per Printed Page**

Defines the maximum number of output lines per printed page.

**Format:**

LINE=n

**Example:**

LINE=35

Describer	Meaning
n	Maximum number of output lines per page (Integer > 0; Default = 50).

**Remarks:**

1. For 11 inch paper, 50 lines per page is recommended; for 8-1/2 inch paper, 35 lines per page is recommended.
2. The NASTRAN statement keyword NLINES may also be used to set this value. See the [NLINES \(9\)](#) Nastran Statement.

## LOAD

### External Static Load Set Selection

Selects an external static load set.

#### Format:

LOAD=n

#### Example:

LOAD=15

Descriptor	Meaning
n	Set identification of at least one external load Bulk Data entry. The set identification must appear on at least one ACCEL, ACCEL1, FORCE, FORCE1, FORCE2, FORCEAX, GRAV, LOAD, MOMAX, MOMENT, MOMENT1, MOMENT2, MPCD, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADB3, PLOADX, QVOL, QVECT, QHBDY, QBDY1, QBDY2, QBDY3, PRESAX, RFORCE, SPCD, or SLOAD entry (Integer > 0).

#### Remarks:

1. A GRAV entry cannot have the same set identification number as any of the other loading entry types. Apply a gravity load along with other static loads, a LOAD Bulk Data entry must be used.
2. LOAD is only applicable in linear and nonlinear statics, inertia relief, differential stiffness, buckling, and heat transfer analyses.
3. The total load applied will be the sum of external (LOAD), thermal (TEMP(LOAD)), element deformation (DEFORM), and constrained displacement (SPC) loads.
4. Static, thermal, and element deformation loads should have unique set identification numbers.

## LOADNAME

Name to be Associated with a SUBCASE in PAA

Provides a name to be associated with a loading condition.

### Format:

LOADNAME = load\_name

### Example:

LOADNAME = GRAVITY LOAD ON OUTBOARD NACELLEL

Describer	Meaning
Load_name	Character string (up to 64 characters long) providing a user-defined name for a loading in PAA.

### Remark:

1. This command is used only in PAA and provides a user-defined name for a loading condition.
2. LOADNAMEs may be used to combine loads in a COMBINE or SOLVE PAA run using a LOADCNAM Bulk Data entry.
3. The LOADNAME command is ignored in non-PAA applications

## LOADSET

### Static Load Set Selection

Selects a sequence of static load sets to be applied to the structural model. The load sets may be referenced by dynamic load commands.

**Format:**

LOADSET=*n*

**Example:**

LOADSET=100

Descriptor	Meaning
<i>n</i>	Set identification number of at least one LSEQ Bulk Data entry. (Integer > 0)

**Remarks:**

1. When used in superelement analysis, this command must be used for all superelements. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
2. When the LOADSET command is used in superelement statics, the residual structure should have as many loading conditions as the number of unique EXCITEID sets defined on all LSEQ entries. The subcases after the first should contain only SUBTITLE and LABEL information, and residual structure output requests. SUBTITLE and LABEL information for all superelements will be obtained from the residual structure.
3. When multiple subcases are specified in the dynamic solution sequences (SOLs 108, 109, 111, 112, 118, 146), the LOADSET must appear in the first subcase or above all subcases. In SOL 200 with ANALYSIS=DFREQ, MFREQ, or MTRAN, a different LOADSET may be specified in the first subcase pertaining to each ANALYSIS command.
4. In SOL 101, the design sensitivity output will identify all expanded subcases by a sequence of unique integers beginning with *n*.
5. In the nonlinear static solution sequences (SOLs 106 and 153), the LOADSET must appear above all subcases and only one LOADSET may be specified.
6. Only one LOADSET command is allowed per superelement and it must be specified in the superelement's first subcase.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of a LOADSET Case Control command, all static loads whose load set IDs match the EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE Bulk Data entries are automatically processed.



8. . If you use the LOADSET command in a static analysis, the loading matrix will be generated corresponding to, and in ascending order of, the EXCITEID's on the referenced LSEQ entries. Any LOAD commands within subcases will be ignored.
9. Use of the LOADSET/LSEQ should be avoided as it is an obsolete entry and is never needed and is only documented for legacy code. Some new features such as SUBSTEP do not support it and will issue a fatal message.

## M2GG

## Direct Input Mass Matrix Selection

Selects direct input mass matrix or matrices.

### Format:

M2GG=name

### Example:

```

M2GG = MDMIG
M2GG = MDMIG1, MDMIG2, MDMIG3
M2GG = 1.25*MDMIG1, 1.0*MDMIG2, 0.82*MDMIG3
SET 200 = M1, M2
M2GG = 200

```

Describer	Meaning
name	Name of a $[M_{gg}^2]$ matrix that is input on the DMIG Bulk Data entry, or name list with or without factors see Remark 6. (Character).

### Remarks:

- DMIG matrices will not be used unless selected.
- Terms are added to the mass matrix before any constraints are applied.
- The matrix must be symmetric, and field 4 on the DMIG, name entry must contain a 6.
- M2GG input is not affected by PARAM,WTMASS. M2GG input must either be in consistent mass units or PARAM,CM2 may be used.
- The matrices are additive if multiple matrices are referenced on the M2GG command.
- The formats of the name list:
  - Names without factor.  
Names separated by comma or blank.
  - Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are real numbers. Each name must be paired with a factor including 1.0.
- If there are multiple subcases in the analysis. the M2GG command must appear above the first subcase or in the first subcase. M2GG requests in the second and subsequent subcases will be ignored. For superelements. it should occur in the first subcase for the appropriate superelement.

**M2PP****Direct Input Mass Matrix Selection**

Selects direct input mass matrix or matrices, which are not included in normal modes.

**Format:**

M2PP=name

**Example:**

```
M2PP = MDMIG
M2PP = MDMIG1, MDMIG2, MDMIG3
M2PP = 5.06*MDMIG1, 1.0*MDMIG2, 0.85*MDMIG3
M2PP = (1.25, 0.5) *MDMIG1, (1.0, 0.0) *MDMIG2, (0.82, -2.2) *MDMIG3
```

Describer	Meaning
name	Name of a $[M_{pp}^2]$ matrix that is input on the DMIG or DMIAX Bulk Data entry, or name list with or without factors, see Remark 7. (Character).

**Remarks:**

- DMIG and DMIAX entries will not be used unless selected by the M2PP input.
- M2PP input is not affected by PARAM,WTMASS. M2PP input must be in consistent mass units.
- The matrix must be square or symmetric, and field 4 on the DMIG, name entry must contain a 1 or 6.
- It is recommended that PARAM,AUTOSPC,NO be specified. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
- M2PP matrices are used only in dynamic response problems. They are not used in normal modes problems.
- The matrices are additive if multiple matrices are referenced on the M2PP command.
- The formats of the name list:
  - Names without factor  
Names separated by comma or blank.
  - Names with factors.

Each entry in the list consists of a factors followed by a star, followed by a name. The entries are separated by commas or blanks. The factors are either all real numbers, or all complex numbers in the form of two real numbers, separated by a comma, within parenthesis as shown in the preceding example. The first real number of the pair is the real part, and the second is the imaginary part. Either part may be zero or blank, but not both. Mixed real numbers and complex numbers are not allowed. Each name must be with a factor including 1.0 for real and (1.0, 0.0) for complex.

## MASSET

### Mass Combination Selection

Selection of a mass combination in multiple mass combination analysis (MMC). This case control command can be used in various solutions sequences SOLs 101, 103, 107-112, 144-146, 200 and 400.

#### Format:

MASSET = n

#### Example:

MASSET = 11

Describer	Meaning
n	Set ID of a MASSET bulk data entry (Integer > 0).

#### Remarks:

- Using a particular mass combination in analysis only requires referencing the mass combinations ID in MASSET case control at the subcase level (or at step level in Sol 400). If no MASSET is defined in the subcase then (like traditional non MMC analysis) the base mass is used as the mass for that subcase. Example for selecting mass combinations at the subcase level:

```
SUBCASE 101
...
MASSET = 11 $
...
SUBCASE 102
...
MASSET = 12 $
```

- The MASSET bulk data entry defines a linear combination of incremental/base mass cases to form the subcase-dependent mass. The bulk data for the incremental mass case is preceded by “BEGIN MASSID=yyy”, where yyy > 0 is the integer ID of the mass increment case. The only data that will be used from the incremental mass bulk data section is element connections and element mass properties including materials (Parameters and grid data included in the incremental mass section will be ignored). The incremental mass will be computed based on the grid data supplied in the base mass bulk data section and the element properties supplied in the incremental mass section. Typically the user will construct the incremental mass by adding concentrated masses (e.g. CONM2, CMASS1 etc.) or by changing the material or element properties of the set of elements defined in incremental mass bulk data section.

```
BEGIN massid=101 label='mass increment using conm2s'
BEGIN massid=102 label='mass increment using materials'
```

- If Part SE are used in the model, mass increment related to these SEs can be specified in a separate bulk data section, “BEGIN SUPER=zzz MASSID=yyy”, where zzz > 0 is the Superelement ID and yyy is the mass increment ID.

```
BEGIN SUPER=2 massid=101 label='mass increment using conm2s'
```

**MASTER**

## Redefine the MASTER Subcase

Allows the redefinition of a MASTER subcase.

**Format:**

```
SUBCASE n
```

```
MASTER
```

**Example:**

```
SUBCASE 10
```

```
MASTER
```

Describer	Meaning
n	

**Remarks:**

1. All commands in a MASTER subcase apply to the following subcases until a new MASTER subcase is defined.
2. Suppose that superelement 10 has SPC set 10, MPC set 10, and LOAD sets 101 and 102. Suppose also that superelement 20 has SPC set 20, MPC set 20, and LOAD sets 201 and 202. Then the following Case Control setup specifies the required subcase structure:

```
TITLE = MY MODEL
DISP = ALL
SEALL = ALL
SUBCASE 101
MASTER
  SPC = 10
  MPC = 10
  SUPER = 10, 1
  LOAD = 101
  LABEL = SUPER 10
  ESE = ALL
SUBCASE 102
  LOAD = 102
  SUPER = 10, 2
SUBCASE 201
MASTER
  SPC = 20
  MPC = 20
  SUPER = 20, 1
  LOAD = 201
  LABEL = SUPER 20
SUBCASE 202
  LOAD = 202
  SUPER = 20, 2
```

3. MASTER may also be used advantageously in multiple boundary condition Case Control setups. Suppose that constraint sets 10 and 20 are to be solved with three loading conditions each: 1, 2, and 3; and 4, 5, and 6, respectively. The following Case Control Section may be used:

```
TITLE = MULTIPLE BOUNDARY CONDITIONS
DISP = ALL
SYM 1
MASTER
  SPC = 10
  LOAD = 1
SYM 2
  LOAD = 2
SYM 3
  LOAD = 3
SYM 4
  MASTER
  SPC = 20
  LOAD = 4
SYM 5
  LOAD = 5
SYM 6
  LOAD = 6
SYMCOM 10
  SYMSEQ = 1., 1., 1., -1., -1., -1.
SYMCOM 20
  SYMSEQ = -1., -1., -1., 1., 1., 1.
```

4. The MASTER command must appear immediately after a SUBCASE or SYM command.

**MAXLINES**

## Maximum Number of Output Lines

Sets the maximum number of output lines.

**Format:**

MAXLINES=n

**Example:**

MAXLINES=150000

Describer	Meaning
n	Maximum number of output lines allowed (Integer > 0; Default = 999999999).

**Remarks:**

1. If MAXLINES is exceeded, the program will terminate.
2. MAXLINES may also be specified on the NASTRAN statement with SYSTEM(14). See the [MAXLINES \(14\)](#).
3. The code counts the number of pages and assumes that the number of lines output is the number of lines allowed per page, specified by the “LINES” command, times the number of pages.

## MAXMIN

## MAXMIN Survey Output Request (old form)

Specifies options for max/min surveys of certain output data associated with grid points.

### Format:

$$\text{MAXMIN} \left( \left[ \left\{ \begin{array}{c} \text{MAX} \\ \text{BOTH} \\ \text{MIN} \\ \text{VMAG} \end{array} \right\}, = \text{num} \right], \left[ \text{CID} = \left\{ \begin{array}{c} \text{GLOBAL} \\ \text{BASIC} \\ \text{cid} \end{array} \right\} \right], \text{oplist}, \text{COMP} = \text{list} \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Example:

MAXMIN (BOTH=10, CID=1000, DISP, COMP=T1/T3)=501

Descriptor	Meaning
MAX	Specifies that only maximum values are to be output. See Remark 1.
MIN	Specifies that only minimum values are to be output. See Remark 1.
BOTH	Specifies that both maximum and minimum values are to be output. See Remark 1.
VMAG	Specifies that vector magnitude resultants are to be output. See Remark 2.
num	Specifies the maximum number of values that will be output. See Remark 3. (Integer > 0, Default = 5).
CID	Specifies the coordinate system frame of reference in which the max/min values will be output. See Remarks 1. and 3.
GLOBAL	Requests output in the global coordinate system frame of reference.
BASIC	Requests output in the basic coordinate system frame of reference.
cid	Requests output in the local coordinate system defined by cid (Integer > 0).
oplist	Specifies a list of one or more standard Case Control commands for which max/min results are to be produced. The list may include any of DISP,SPCF, OLOAD,MPCF,VELO,ACCE, or ALL. See Remark 6 (Character, no Default).
list	Specifies a list of grid point degree of freedom component directions that will be included in the max/min survey output. The components are separated by slashes and are selected from T1, T2, T3, R1, R2, and R3. See Remarks 4. and 5. (Character, Default=/T1/T2/T3/R1/R2/R3).
ALL	MAXMIN survey results for all points will be output.
NONE	MAXMIN survey results for no points will be output.
n	Set identification of a previously appearing SET command. The max/min results survey will be output only for the points specified SET n (Integer > 0).



**Remarks:**

1. The MAXMIN command produces an algebraically ascending sorted list of the output quantities specified for all of the points in the selected set. MAX refers to the largest magnitude positive values, while MIN refers to the largest magnitude negative values. The output format is similar to that of displacement output. All components will be output for a grid point, and the order of the grid points will be in sort on the particular component that was surveyed. The output title contains the identification number of the set of points participating in the max/min output, the coordinate system frame of reference, the number of MAX and MIN values output, and the component that was surveyed. When the output being surveyed is in the global output coordinate system reference frame, and BASIC or a local output coordinate system is specified as cid, both the sorted system output and the original reference system output are displayed for the grid point if these systems are different.
2. Vector magnitude results are computed for both translations and rotations and are displayed under the T1 and R1 column headings. The presence of the COMP keyword is ignored.
3. The default value of 5 generates a minimum of 10 output lines for the BOTH option. There will be 5 maximum values and 5 minimum values produced. In addition, if coordinate system are involved, both surveyed and original data will be output. This could result in as many as 10 more lines of output for each surveyed component.
4. Multiple MAXMIN commands may be specified for a subcase. This permits different output quantities to have different MAXMIN specification within a subcase. For example,
 

```
SET 501=1,3,5,7 THRU 99, 1001, 2001
MAXMIN (DISP, COMP=T3) = 501
MAXMIN (SPCF, COMP=T1/R3) = ALL
```
5. Scalar point output is included only if component T1 is included in the list.
6. MAXMIN output will only be generated for items in the oplist when there is an associated Case Control command present. For example, a DISP Case Control command must be present in order for the MAXMIN(DISP) = ALL command to produce output. Use of ALL keywords for the oplist requests MAXMIN output for all output commands acceptable to MAXMIN that are present in Case control Section.

## MAXMIN

## Requests Output of Maximums and Minimums in Data Recovery (new form)

Requests the output of maximums and minimums in data recovery.

### Format:

$$\text{MAXMIN} \left[ \left[ \begin{array}{c} \text{GRID} \\ \text{ELEM} \\ \text{BOTH} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{NOPRINT} \end{array} \right], \left[ \begin{array}{c} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \right. \\ \left. \left[ \begin{array}{c} \text{CENTER} \\ \text{CUBIC} \\ \text{SGAGE} \\ \text{CORNER or BILIN} \end{array} \right], \left[ \begin{array}{c} \text{STRCUR} \\ \text{FIBER} \end{array} \right] \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Example:

```
MAXMIN (GRID) = 12
MAXMIN (ELEM) = ALL
MAXMIN = NONE
```

Descriptor	Meaning
GRID	Indicates the request is applied to grid points.
ELEM	Indicates the request is applied to elements.
BOTH	Indicates the request is applied to both elements and grid points (Default).
PRINT	Compute and write the output to the .f06 file (Default).
PUNCH	Compute and write output to the punch file.
NOPRINT	Compute but do not write out the results.
ALL	Max/min results will be reported for all elements.
n	Set identification number. The referenced SET command defines a set of elements or grid points to be monitored.
NONE	Max/min results will not be reported.
VONMISES	von Mises stress/strains are output.
MAXS or SHEAR	Maximum shear stress/strains are output.
STRCUR	Strain at the reference plane and curvatures are output for plate elements.
FIBER	Stress/Strain at locations Z1, Z2 are computed for plate elements.
CENTER	Output CQUAD4 stress/strains at the center only.

Describer	Meaning
CORNER or BILIN	Output CQUAD4 element stress/strains at the center and grid points using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element stress/strains at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element stress/strains at center and grid points using cubic bending correction.

**Remarks:**

1. MAXMIN is not allowed in REPCASE, but is allowed in SUBCOM and SYMCOM.
2. The OTIME command may be used to limit the time span of monitoring.
3. No corresponding output request such as DISP, STRESS, etc., is required (Except for STRAIN). Also, SET=n may request more elements or grid points for monitoring than is requested by Outputs. Currently for strain the output request STRAIN=n is required where n can reference any element. If strain at fibers is requested then, then STRAIN (FIBER) = n is required.

The output is comprised of two parts: (1) a summary of the maximum/minimum values and the times they occur, and (2) the associated output for all components of the element or grid. The first part is always output, and the second part is only output if the FULL describer is specified on the MAXMIN(DEF) Case Control command. See the next section for a description of the new Case Control commands. Here are sample Case Control commands for the output of maximum von Mises stresses using the BRIEF option:

```
MAXMIN(DEF) STRESS QUAD4 SMAX1 ABSOLUTE(5) BRIEF
SET 100=4
MAXMIN(ELEM)=4
```

4. See DISPLACEMENT, STRESS, or STRAIN Case Control commands for additional keyword implications.

## MAXMIN(DEF)

Defines Parameters for Monitoring Maximums and Minimums (new form)

Defines parameters and output options for the monitoring of maximums and minimums in data recovery. MAXMIN(DEF) must be specified above all subcases. The MAXMIN(ELEM) and/or MAXMIN(GRID) Case Control command is then required to print the max/min results.

### Format for Grid Point Output:

$$\text{MAXMIN(DEF)} \{ \text{DISP, VELO, ACCE, MPCF, SPCF, OLOAD} \}$$

$$\{ \text{T1 T2 T3 R1 R2 R3 MAGT MAGR} \},$$

$$\left[ \text{ABSOLUTE(p) MINALG(q) MAXALG(r)} \right] \left[ \text{RMS} \right] \left[ \begin{array}{l} \text{BRIEF} \\ \text{FULL} \end{array} \right],$$

$$\left[ \begin{array}{l} \text{ALL(p)} \\ \text{CID} = \left\{ \begin{array}{l} \text{GLOBAL} \\ \text{BASIC} \\ \text{cid} \end{array} \right\} \end{array} \right]$$

### Format for element output:

$$\text{MAXMIN(DEF)} \left\{ \begin{array}{l} \text{STRESS} \\ \text{STRAIN} \\ \text{FORCE} \end{array} \right\} \{ \text{eltype1 [, eltype2, ...]} \}$$

$$\left\{ \text{comp1} \left[ \begin{array}{l} \text{CENTER} \\ \text{ALL} \\ \text{GROUP} \\ \text{ENDS} \end{array} \right] \left[ \text{ , comp2...} \right] \right\},$$

$$\left[ \text{ABSOLUTE(p) MINALG(q) MAXALG(r)} \right] \left[ \text{RMS} \right] \left[ \begin{array}{l} \text{BRIEF} \\ \text{FULL} \end{array} \right]$$

$$\left[ \begin{array}{l} \text{ALL(p)} \end{array} \right]$$

### Examples:

```
MAXMIN(DEF) stress (quad4,smxl) MAXA=5
MAXMIN(DEF) disp T1 T2 T3 MAGT RMS FULL
```

Describer	Meaning
DISP, VELO, etc. (grid point output) STRESS, etc. (element output)	Type of result to be monitored.
T1, T2, etc.	Name of the grid point component to be monitored.
MAGT	Specifies the magnitude of the translational components are to be monitored.
MAGR	Specifies the magnitude of the rotational components are to be monitored
eltype1 eltype2 ...	Name of the element type(s) to be monitored. At least one element type must be present. All elements designations are generic and the "C", designation Connections, should be dropped. The exceptions for this are the CONROD and CONV element types.
comp1 comp2...	Name of the element component(s) to be monitored; e.g., etmax1 for max shear strain in the Z1 plane. The component names are defined in the Nastran Data Definition Language (NDDL). Also, the item codes from <a href="#">Item Codes</a> may be used.
ABSOLUTE(p)	Print out the top p absolute values (Default for p is 5).
MINALG(q)	Print out the bottom q minimum algebraic values (Default for q is 5).
MAXALG(r)	Print out the top r maximum algebraic values (Default for r is 5).
ALL(p)	Print out all options: ABSOLUTE, MINALG, and MAXALG. (Default for p is 5).
BRIEF	Print out only the maxmin results (Default).
FULL	Print out the maxmin results followed by the standard data recovery format for the elements and grids at the retained set of the maximum or minimum occurrences.
GLOBAL	Selects the global coordinate system (see CD on the GRID entry) for monitoring grid point results.
BASIC	Selects the basic coordinate system for monitoring grid point results (Default).
cid	Specifies a coordinate system ID for a system defined on a CORDij entry for monitoring grid point results.
RMS	Print out the root-mean-square value of each maximum or minimum value requested by ABSOLUTE, MIN, or MAX over all time steps.
CENTER	Component selector when element allows for component name to occur in multiple places (Default).

Describer	Meaning
ALL	Selects all locations in an element where multiple locations exist.
GROUP	Reduces all occurrences of a component name to a single value before the action is performed.
ENDS	Selects the ends of a BEAM element ignoring intermediate stations.

### Remarks:

1. MAXMIN(DEF) must be specified above all subcases, but this is not sufficient to request monitoring of maximums and minimums. The MAXMIN command must also be specified above or inside subcases.
2. MAXMIN(DEF) may be specified more than once.
3. Multiple element types may be grouped together, if the same component name is to be monitored across those types, by enclosing the element types in parentheses. This grouping does not combine the element types during processing. Each type and component action is performed in class.
4. Multiple component names may be compared collectively to the current maximum (or minimum), but only the maximum (or minimum) component in the group will be reported in the output. This is requested by enclosing the component names in parentheses.
5. Grid point component output is always converted to the basic coordinate system for monitoring when processing "sort1". The global system is the default when processing "sort2".
6. Results for layers in composite elements, or intermediate stations in CBAR and CBEAM elements, are not supported.
7. Only real data recovery is supported.
8. When no MINA, MAXA, or ABSO keyword is supplied, the default values of p, q, and s will be 5. When any keyword is supplied, the other unreferenced keyword values will be set to zero, and no output will be created.
9. The component action keywords of CENTER, ALL, GROUP, and ENDS can only be applied to component names defined in the NDDL and occur at multiple places in element data recovery. They cannot be used with Item Codes.
10. Specify CQUAD4C and CTRIAC for corner stresses of CQUAD4 and CTRIA3 elements.
11. MAXMIN data output to the .op2 and .xdb files are not supported.
12. SORT1 and SORT2 output Case Control options (such as DISP(SORT2)=ALL), cause MAXMIN to behave differently.
  - SORT1 is only supported in SOLs 101, 103, 105, and 109. For each static loadcase, mode buckling eigenvalue, or time step, all the grids or elements will be searched for maximums and minimums.
  - SORT2 is only supported in SOL 112. For each selected grid or element, all the time steps will be searched for maximums and minimums.

## MCFRACTION

### Modal Contribution Fractions Output Request

Requests the form and type of modal contribution fraction output.

#### Format:

$$\begin{aligned}
 & \text{MCFRACTION}(\text{[STRUCTURE]}, \left[ \begin{array}{c} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{[REAL or IMAG]} \\ \text{PHASE} \end{array} \right], \text{[(SORT = sorttype)]}, \\
 & \left[ \text{KEY} = \begin{array}{c} \text{[FRACTION]} \\ \text{sortitem} \end{array} \right], \left[ \text{ITEMS} = \begin{array}{c} \text{[FRACTION]} \\ \text{ALL} \\ \text{(itemlist)} \end{array} \right] \left[ \text{SOLUTION} = \begin{array}{c} \text{[ALL]} \\ \text{m} \\ \text{NONE} \end{array} \right] \left[ \text{FILTER} = \begin{array}{c} \text{[0.001]} \\ \text{fratio} \end{array} \right] \\
 & \left[ \text{NULL} = \begin{array}{c} \text{[12]} \\ \text{ipowr} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

#### Example:

```

SET 1001 = 10.0, 20.0, 100.0
SET 2001 = 716/T3, 809/T3, 412/T1
MCFRACTION (STRUCTURE, PRINT, PUNCH, ITEM=FRACTION,
             SORT=ABSD, KEY=PROJECTION, SOLUTION=1001)=2001
MCFRACTION (ITEMS=(FRACTION, PROJECTION), FILTER=0.01)=2001
    
```

Describer	Meaning
STRUCTURE	Request pertains to structure points only.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG      Requests rectangular format (real and imaginary) of complex output.

Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output.
SORT	Keyword selecting one of the sort type options. Default is to produced output in increasing natural mode number order.
sorttype	One of the following modal contribution fraction output sorting options: ABSA - output will be sorted by absolute value in ascending order. ABSD - output will be sorted by absolute value in descending order. ALGA - output will be sorted algebraically in ascending order. ALGD - output will be sorted algebraically in descending order.
KEY	Keyword selecting a sorting operation key quantity.
sortitem	Item from the item list table on which the SORT operation is performed. (Default is FRACTION.)
ITEMS	Keyword specifying data selection options.
itemlist	One (or more) of the following modal contribution fraction output items. If more than one item is selected, the list must be enclosed in parentheses:

Item Identifier	Description
RESPONSE	Each mode's response at each degree of freedom is selected.
PROJECTION	Projection of modal response on solution.
FRACTION	Fraction of total displacement per mode (PROJECTION divided by total).
SCALED	Scaled magnitude (PROJECTION divided by largest magnitude of modal response).
MODEDISP	Modal displacements (complex solution at each DOF by mode number).
MODERESP	Modal response for each mode (polar format with respect to total displacement).

SOLUTION	Keyword specifying the output solution time step, forcing frequency, or complex eigenvalue selections for which modal fractions will be generated (Default = ALL).
m	Results for solutions in SET m will be output.
FILTER	Keyword specifying the value of the printed output data filter.
fratio	Value of output filter ratio (Default = 0.001).
NULL	Keyword specifying the power of ten used to detect a null response quantity.
ipowr	The power of ten used to detect a null response quantity (Default = 12).
n	Results for grid point components in SET n will be output.
ALL	Results for all solutions and/or grid point components will be output.
NONE	No modal contribution fractions will be output.



**Remarks:**

1. The MCFRACTION Case Control command is useful in modal frequency response (SOL 111), modal transient response (SOL 112), and modal complex eigenvalue analysis (SOL 110) solution sequences only. If superelements are used, its use is restricted to residual structure data recovery operations only. MCFRACTION is not supported in SOL 200 or 400.
2. Printed output includes results for all of the data items described in the itemlist table.
3. Punched output includes results for only the data items selected by the ITEMS keyword.
4. Modal contribution fractions are sorted by increasing order of mode number unless the SORT keywords specifies a particular sorting order. If a sorting order is specified, the KEY keyword selects the particular data item in the printed results tabular output listing that is sorted. When MODEDISP is selected, the magnitude is sorted. When MODERESP is selected, the real portion of the response is sorted.
5. The SOLUTION keyword can be used to select a subset of the solutions available. If SET m is specified, the items in the SET list are forcing frequency values, time step values, or complex eigenvalue mode numbers, depending upon the solution sequence used.
6. The FILTER keyword specifies a filter ratio value that is used to limit the amount of printed output produced. It applies to the data item selected by the KEY keyword if it is specified. If no KEY keyword is present, the default value of KEY=FRACTION will be used. The maximum value for the selected data item across all natural modes is determined. If the ratio of the data item value to the maximum data item value is less than fratio for any natural mode, no output for that natural mode is produced.
7. If the magnitude of the total response at a selected grid pint component is less than  $1.0 \times 10^{-\text{ipowr}}$ , no modal contribution fraction output is generated for that degree of freedom. If ipowr is not in the range of 1 to 31, the default value of 12 is used for ipowr, producing a null response threshold of  $1.0 \times 10^{-12}$ .
8. For modal transient response solution sequences, response quantities are real numbers. There are no imaginary terms. Therefore, polar representations of the data has no meaning. Furthermore, projections of responses onto the total response are simply the individual modal contribution to the total response at a degree of freedom. Thus, the only items available for output are the individual modal response magnitude (PROJECTION), the modal fraction (FRACTION), and the scaled response magnitude (SCALED). Selection of any of the other items from the itemlist table causes selection of the modal response magnitude (PROJECTION) item.
9. A request of MCFRACTION output for a SET that contains no degrees-of-freedom in the analysis set, will result in the message: "SYSTEM WARNING MESSAGE 2001 (MCFRAC)"

## MCHSTAT

## Change State in SOL 600

Indicates which MCHSTAT Bulk Data entry option will be used to control the state variables (temperatures) used in this subcase for a thermal stress simulation in SOL 600.

### Format:

MCHSTAT=N

### Example:

MCHSTAT=10

Describer	Meaning
N	ID of a matching Bulk Data MSCHSTAT entry specifying temperatures or other state variable values to be used from a previous analysis for this subcase.

### Remark:

1. This entry may only be used in SOL 600.

## MEFFMASS

## Modal Effective Mass Output Request

Requests the output of the modal effective mass, participation factors, and modal effective mass fractions in normal modes analysis.

### Format:

$$\text{MEFFMASS} \left[ \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right] \left[ \begin{array}{l} \text{PUNCH} \\ \text{NOPUNCH} \end{array} \right], \text{GRID} = \text{gid}, \left[ \begin{array}{l} \text{SUMMARY, PARTFAC,} \\ \text{MEFFM, MEFFW,} \\ \text{FRACSUM, ALL} \end{array} \right] = \left\{ \begin{array}{l} \text{YES} \\ \text{NO} \end{array} \right\}$$

### Examples:

```
MEFFMASS
MEFFMASS (GRID=12, SUMMARY, PARTFAC)
```

Describer	Meaning
PRINT	Write output to the print file (Default).
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
NOPUNCH	Do not write output to the punch file (Default).
gid	Reference a grid point for the calculation of the rigid body mass matrix. The default is the origin of the basic coordinate system.
SUMMARY	Requests calculation of the total effective mass fraction, modal effective mass matrix, and the rigid body mass matrix (Default).
PARTFAC	Requests calculation of modal participation factors.
MEFFM	Requests calculation of the modal effective mass in units of mass.
MEFFW	Requests calculation of the modal effective mass in units of weight.
FRACSUM	Requests calculation of the modal effective mass fraction.

### Remarks:

- The SUMMARY describer produces three outputs:

- Modal effective mass matrix  $[\varepsilon^T][m][\varepsilon]$

where:

$$\varepsilon = \text{Modal participation factors: } [m^1][\phi^T][M_{aa}][D_{ar}].$$

$$m = \text{Generalized mass matrix.}$$

$$\phi = \text{Eigenvectors.}$$

$M_{aa}$  = Mass matrix reduced to the a-set (g-set for superelements).

$D_{ar}$  = Rigid body transformation matrix with respect to the a-set.

- b. A-set rigid body mass matrix:  $[D_{ar}^T][M_{aa}][D_{ar}]$ . For superelement this is computed at the g-set
  - c. Total effective mass Fraction: i.e., diagonal elements of the modal effective mass matrix divided by the rigid body mass matrix.
2. The PARTFAC descriptor outputs the modal participation factors table  $\epsilon$ .
  3. The MEFFM descriptor outputs the modal effective mass table  $\epsilon^2$ , the term-wise square of the modal participation factors table.
  4. The MEFFW descriptor outputs the modal effective weight table; i.e., the modal effective mass divided by PARAM, WTMAS.
  5. The FRACSUM descriptor outputs the modal effective mass fraction table; i.e., the generalized mass matrix (diagonal term) multiplied by the modal effective mass divided by the rigid body mass matrix (diagonal term).
  6. For superelements, the MEFFMASS Case Control command uses the residual structure eigenvalues and eigenvectors, by default. If, however, PARAM, FIXEDB, -1 is specified, then the MEFFMASS Case Control command uses the component mode eigenvalues and eigenvectors.

**METHOD**

## Real Eigenvalue Extraction Method Selection

Selects the real eigenvalue extraction parameters.

**Format:**

$$\text{METHOD} \begin{bmatrix} \text{BOTH} \\ \text{STRUCTURE} \\ \text{FLUID} \\ \text{COUPLED} \end{bmatrix} = n$$

**Examples:**

METHOD=33

METHOD (FLUID) =34

METHOD (COUPLED) =100

Describer	Meaning
BOTH	The referenced EIGR or EIGRL Bulk Data entry will be applied to both the structure and the fluid portion of the model (Default).
STRUCTURE or FLUID	The referenced EIGR or EIGRL Bulk Data entry is applied to the structural or fluid portion of the model.
COUPLED	The referenced EIGR or EIGRL Bulk Data entry is applied to the structural and fluid coupled system of the model.
n	Set identification number of an EIGR or EIGRL Bulk Data entry for normal modes or modal formulation, or an EIGB or EIGRL entry for buckling (Integer > 0).

**Remarks:**

1. An eigenvalue extraction method must be selected when extracting real eigenvalues.
2. If the set identification number selected is present on both EIGRL and EIGR and/or EIGB entries, the EIGRL entry will be used.
3. METHOD(FLUID) and METHOD(STRUCTURE) permits a different request of EIGR or EIGRL for the fluid portion of the model in coupled fluid-structural analysis.
  - If METHOD(STRUCTURE) or METHOD(FLUID) is also specified, then they will override the METHOD(BOTH) selection.
  - The METHOD(FLUID) and METHOD(STRUCTURE) may be specified simultaneously in the same subcase for the residual structure only. Do not specify METHOD(FLUID) in a superelement subcase even if the superelement contains fluid elements.

- The auto-omit feature (see [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide*) is not recommended. Therefore, only those methods of eigenvalue extraction that can process a singular mass matrix should be used; e.g., the EIGRL entry, or MGIV and MHOU on the EIGR entry.
4. METHOD(COUPLED) is only used when STRUCTURE and FLUID COUPLED problem. The corresponding EIGRL or EIGR requires either the frequency range of interest or ND (number of roots desired) entry. The real coupled modes computation is done with combination of Lanczos method and Subspace iteration method.

METHOD(COUPLED) is recommended for heavy fluid (like liquids) model. When it is applied to the light fluid model, it might require more coupled modes to get the correct result.

Real coupled method is not supported in SOL 200 and SOL 400. If Case Control METHOD(COUPLED) is used in SOL 200 or SOL 400, the following error message will be returned:

```
*** USER FATAL MESSAGE 22996 (SUBDMAP PREOPT)
```

```
    CASE CONTROL COMMAND METHOD(COUPLED) IS NOT SUPPORTED IN SOL  
200 and SOL 400.
```

**MFLUID**

## Fluid Boundary Element Selection

Selects the MFLUID Bulk Data entries to be used to specify the fluid-structure interface.

**Format:**

MFLUID = n

**Example:**

MFLUID = 919

Describer	Meaning
n	Set identification number of one or more MFLUID Bulk Data entries (Integer > 0).

**Remark:**

1. For additional discussion, see [Coupled Fluid-Structure Analysis](#) in the *Dynamic Analysis User's Guide*.
2. Use parameter PARAM,VMOPT controls how the virtual mass is processed.
3. Virtual mass is not supported with the following features:
  - a. Cyclic symmetry analysis (SOLs 114, 115, 116 and 118)
  - b. ADAMS MNF generation (ADAMSMNF Case Control command)
  - c. AVL/EXCITE™ generation (AVLEXB Case Control command).

## MFREQUENCY

### Master Frequency Set Selection Frequency Response

Selects the set of forcing frequencies to be used in frequency analysis as master frequencies for K, B, and K4 matrix generation at each master frequency, in frequency dependent analysis.

#### Format:

$$MFREQUENCY\left(\begin{matrix} \text{Linear} \\ \text{Log10} \end{matrix}\right), [TOL= 0.1] = \begin{pmatrix} \text{AUTO} \\ \text{NOAUTO} \\ n \end{pmatrix}$$

#### Example:

MFREQ (TOL=.15) ← Nastran will generate a master frequency list with TOL=.15

Describer	Meaning
Linear	Linear interpolation will be used between master frequencies to compute K, B, and K4 matrices found on selected Case Control FREQ entry. (Character, Default)
Log10	Log interpolation will be used between master frequencies to compute K, B, and K4 matrices found on selected FREQ entry. (Character)
TOL	For the default option AUTO, for frequency dependent TABLEDi entries associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries a master frequency will be selected whenever there is a TOL jump in any table value. A TOL=0.0 is the same as NOAUTO. (Real ≥0.0)
AUTO	For frequency dependent analysis associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries, Nastran will automatically, depending on associated frequency dependent TABLEDi entries and the required Case Control FREQ=m entry, generate a series of master frequencies. (Character, Default).  If the user has supplied a Case Control MFREQ entry pointing to a FREQ, FREQ1, or FREQ2 bulk data entry, Nastran will use the user supplied entry and its associated frequency list.
NOAUTO	If the user has not supplied a MFREQ Case Control entry and the analysis meets the requirements of AUTO above, then NOAUTO will prevent the generation of master frequencies. In this case, the required Case Control FREQ will be used for all frequencies. And all frequencies are considered master frequencies. (Character)
n	Set identification number of FREQ, FREQ1, and FREQ2 Bulk Data entries. (Integer > 0)



**Remark:**

1. Both direct frequency and modal frequency allow, through MAT1F, MAT2F, MAT8F, MAT9F, and PBUSHT, PELAST, PDAMPT entries frequency dependent analysis. The presence of any active MAT1F, MAT2F, MAT8F, or MAT9F entry or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries a master frequency will initiate the master frequency feature in frequency analysis.
2. MFREQ should be above or in the first subcase.
3. The K, B, and K4 matrices, at master frequencies, are formed in the standard way, using material and property values at the associated current master frequency.
4. For frequencies selected between master frequencies, interpolation of the K, B, and, K4 matrices are performed. These interpolated frequencies are selected using the required Case Control FREQ=m entry.
5. Interpolation between master frequencies  $K^{lm}$  and  $K^{2m}$ ;  $K_{ij} = (d_2 K^{lm}_{ij} + d_1 K^{2m}_{ij}) / (d_1 + d_2)$  where.

For Linear:

$$d1 = (f-f1); \quad d2 = (f2 - f)$$

For Log10:

$$d1 = (\log_{10}(f) - \log_{10}(f1)); \quad d2 = (\log_{10}(f2) - \log_{10}(f))$$

6. For master frequencies there is full displacement, velocity, acceleration type data recovery as well as full element force and stress type data recovery.
7. For interpolated frequencies, there is only (a-set) or modal (h-set) displacement, velocity, acceleration type data recovery. Because, the solution of interpolated frequencies is based on interpolated K, B, and K4, the element stress and force data will be less accurate as the interpolated frequency gets farther from a master frequency. This is, also true, for fluid data recovery such as Panel Participation Factors. If accurate element results are required at specific frequencies, then MFREQ=n should be used to define Master frequencies. If accurate element results are required at all frequencies, then the user should either set MFREQ to the same set as defined on the Case Control Command FREQ=n or set MFREQ=NOAUTO.
8. For modal frequency analysis, associated with master frequencies, residual vector methods for viscous and structural damping have been greatly enhanced and orthogonalization techniques are employed which require the removal of free – free modes. For this reason the structure must be statically constrained with the use of SUPORT entries. Case Control selectable SUPORT1 entries may also be used.
9. For preloading (STATSUB), with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries, a fatal message will be issued.

\*\*\* USER FATAL MESSAGE 22711 (MTMD56)

FREQUENCY case control command missing for frequency response analysis.

STATSUB static loading is not currently supported for frequency dependent material/property.

10. PARAM, NMNLFREQ, REAL (Default=0.0) allows users to select material or property values for frequency dependent materials or properties at a desired frequency other than the “nominal” values specified on the MATi or PBUSH, PCOMP, PCOMPG, PDAMP, or PELAS entries. These values are used in determining the eigenvalue solution used in the modal frequency analysis.
11. If a user supplies a FREQ, FREQ1, or FREQ2 entry with a 0.0 value of forcing frequency the following message will be issued:

\*\*\* USER WARNING MESSAGE 22714 (SUBDMAP MFREQRS)

Forcing frequency value of 0.0 detected, nominal (initial) material properties used.

The rationale for this selection is that 0.0 Hz is a static solution and SOL 101 would use material nominal.

12. Some examples of interaction between MFREQ=AUTO and an always required FREQ=m and interpolated values are as follows:

If frequency dependent TABLEDi entries associated with the presence of any MAT1F, MAT2F, MAT8F, MAT9F entries or the combined presence of any MAT1F, MAT2F, MAT8F, MAT9F entries and any PBUSHT, PELAST, PDAMPT entries are detected, and the user has not specified any MFREQ Case Control entries, then Nastran will automatically scan the TABLEDi entries. Nastran does this by using the required FREQ=m entry and for any TABLEDi where there is a TOL jump between two consecutive FREQ=m frequencies determining a “Master frequency”.

```
SOL111
CEND
FREQ = 10 ← Required
...
BEGIN BULK
FREQ1, 10, 10.0, 5.0, 78
...
MAT1, 2, 35.46, , .49, 1.1-9
MAT1F, 2, 101, 103, 102 ← All 3 Tables for E, G, NU
required.
...
TABLED1, 101,
, .01, 7.3941, 50., 18.017, 100., 24.386, 200., 35.46,
, 300., 44.45, 400., 51.264, 500., 56.321, ENDT
TABLED1, 102
, .01, 0.49, 1000., 0.49, ENDT
TABLED1, 103
, .01, 2.481, 50., 6.076, 100., 8.1832, 200., 11.899
, 300., 14.916, 400., 17.203, 500., 18.90, ENDT
TABLED1, 200 ← Based on say an active PBUSHT entry.
, .01, 0.4, 50., 0.65, 100., 0.78, 200., 0.92
, 300., 0.99, 400., 1.03, 500., 1.07, endt
```

**Internally Nastran Generates in Case Control:**

**MFREQ=11**

**Internally Nastran Generates in "Bulk Data":**

```
FREQ, 11, 10., 15., 20., 30., 40., 50., 65.,
, 85., 105., 130., 160., 190., 230., 275., 330.,
, 400.
```

Where for the above, Nastran has based on the user supplied  $FREQ=10$  Case Control entry, scanned all the active user supplied  $TABLEDi$  entries, and whenever there is a jump in table value of 10% (default TOL), it creates an entry into the master frequency list  $MFREQ=11$ .

$MFREQ$  will issue the message:

```
***AUTOMATIC MASTER FREQUENCY LIST HAS BEEN SELECTED FOR  
SUBCASE ID      1:  
FREQ, 11, 10., 15., 20., 30., 40., 50., 65.,  
      ,85., 105., 130., 160., 190., 230., 275., 330.,  
      ,400.
```

## MODALKE

### Modal Kinetic Energy Request

Requests a modal kinetic energy calculation and specifies the output form.

#### Format:

$$\text{MODALKE} \left[ \left( \left[ \text{SORT1} \right] \left[ \text{PRINT} \right], \text{PUNCH} \right) \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] \left[ \text{ESORT} = \left\{ \begin{array}{c} \text{MODE} \\ \text{ASCEND} \\ \text{DESCEND} \end{array} \right\} \right] \right]$$

$$\left[ \text{THRESH} = e \right] \left[ \left\{ \begin{array}{c} \text{TIME} \\ \text{FREQ} \end{array} \right\} = \left\{ \begin{array}{c} \text{ALL} \\ r \end{array} \right\} \left[ \left\{ \begin{array}{c} \text{AVERAGE} \\ \text{AMPLITUDE} \\ \text{PEAK} \end{array} \right\} \right] \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

#### Examples:

```
MODALKE= ALL
SET 200= 1, 3, 4, 5, 7
MODALKE (ESORT=DESCEND, THRESH=.001) = 200
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of modes for each frequency or time step.
SORT2	Output will be presented as a tabular listing of frequencies or time steps for each mode.
PRINT	Write the results to the .f06 file (Default).
NOPRINT	Do not write the results to the .f06 file.
PUNCH	Write the results to the punch (.f07) file.
ESORT	Present the modal energies sorted by mode number, ascending energy value or descending energy value (Default is MODE)
THRESH	Write out only those energies greater than e (Default = 0.001).
TIME or FREQ	Compute energies at all time steps, or frequencies, or the set of frequencies defined by SET r (Default = ALL).
AVERAGE	Requests average energy in frequency response analysis only (Default).
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
ALL, n, NONE	Compute modal energies for (1) all modes, (2) the modes defined on SET n, or (3) no modes.

**Remarks:**

1. Modal kinetic energy calculations will be limited to SOLs 112 (modal transient response) and 111 (modal frequency response).
2. The MODES descriptor selects from the set of the modes selected by the combination of the Case Control command MODESELECT, and user parameters PARAM,LMODES; PARAM,LFREQ; PARAM,HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) descriptor selects from the set of the time steps (or forcing frequencies) selected by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.
4. For frequency response analysis, the energy may be output as real or complex values. When REAL is specified, the output will be one of AVERAGE, AMPLITUDE or PEAK energy. (See Remark 4. of the EKE Case Control Command for a definition of these terms.) When IMAG or PHASE is specified, the output will be complex energy values.

**Output Format:**

The output formats for complex energy in frequency response analysis follow.

**For SORT1 option:**

FREQUENCY = 2.100000E+01			
M O D A L K I N E T I C E N E R G Y (REAL/IMAGINARY)			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	1.171343E-04 0.0	1.409326E-02 0.0	1.065749E-02 0.0
2	2.049217E-05 0.0	2.465558E-03 0.0	1.864483E-03 0.0
3	5.175167E-04 0.0	6.226612E-02 0.0	4.708635E-02 0.0
4	1.732168E-05 0.0	2.084095E-03 0.0	1.576016E-03 0.0

This format repeats for each frequency.

**For SORT2 option:**

MODE NUMBER = 1			
M O D A L K I N E T I C E N E R G Y (MAGNITUDE/PHASE)			
FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	8.147644E-04 0.0	1.000000E+00 0.0	9.924866E-01 0.0
2.000000E+00	4.066132E-03 0.0	1.000000E+00 0.0	9.936857E-01 0.0
3.000000E+00	1.411670E-02 0.0	1.000000E+00 0.0	9.955440E-01 0.0

This format repeats for each mode number.

The output formats for AVERAGE energy in frequency response analysis follow. The particular type (AVERAGE, AMPLITUDE or PEAK) is identified in the output title.

### For SORT1 option:

```
FREQUENCY = 2.100000E+01
```

MODAL KINETIC ENERGY ( AVERAGE )			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	5.856715E-05	1.409326E-02	1.065749E-02
2	1.024608E-05	2.465558E-03	1.864483E-03
3	2.587583E-04	6.226612E-02	4.708635E-02
4	8.660840E-06	2.084095E-03	1.576016E-03

This format repeats for each frequency.

### For SORT2 option:

```
MODE NUMBER = 1
```

MODAL KINETIC ENERGY ( AVERAGE )			
FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	4.073822E-04	1.000000E+00	9.924866E-01
2.000000E+00	2.033066E-03	1.000000E+00	9.936857E-01
3.000000E+00	7.058351E-03	1.000000E+00	9.955440E-01
4.000000E+00	2.872413E-02	1.000000E+00	9.977741E-01

This format repeats for each mode number.

The output formats for transient response analysis follow.

### For SORT1 option:

```
TIME STEP = 1.000000E+00
```

MODAL KINETIC ENERGY			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	1.483182E-02	3.610759E-01	2.316040E-01
2	4.107675E-02	1.000000E+00	6.414274E-01
3	2.165481E-04	5.271792E-03	3.381472E-03
4	1.387959E-05	3.378941E-04	2.167345E-04

This format repeats for each time step.

### For SORT2 option:

```
MODE NUMBER = 2
```

MODAL KINETIC ENERGY			
TIME STEP	ACTUAL	NORMALIZED	FRACTIONAL
0.000000E+00	0.0	0.0	0.0
9.999999E-02	2.512383E-03	1.000000E+00	9.420497E-01
2.000000E-01	8.475699E-03	1.000000E+00	9.56881E-01
3.000000E-01	1.415435E-02	1.000000E+00	9.568216E-01

This format repeats for each mode number.

### Processing:

Modal kinetic energy is calculated using the following relations:

For transient analysis,

$$[\text{Actual KE}] = \frac{1}{2} \{\dot{u}_h\}^T [M_{hh}] \{\dot{u}_h\}$$

For frequency response analysis,

$$[\text{Actual KE}] = \frac{1}{2} \{u_h\}^T [\text{diag}(\omega_i^2)] [M_{hh}] \{u_h\}$$

where  $\omega_i$  is the excitation frequency and output is complex. See Remark 4 of the [EKE \(Case\)](#) Case Control command for energy calculation relations when output is real.

[Normalized Kinetic Energy] = norm[Actual Kinetic Energy], normalized per column.

[Fractional Kinetic Energy] = [Normalized Kinetic Energy]/[diagonal [ $\{1.0\}^T$  [Normalized Kinetic Energy]]], term-by-term division.

## MODALSE

### Modal Strain Energy Request

Requests modal strain energy calculation and specifies the output form.

#### Format:

$$\text{MODALSE} \left[ \left( \left[ \text{SORT1} \right] \left[ \text{PRINT} \right] \left[ \text{NO PRINT} \right], \text{PUNCH} \right) \left[ \left[ \text{REAL or IMAG} \right] \left[ \text{PHASE} \right] \right] \left[ \text{ESORT} = \left\{ \begin{array}{l} \text{MODE} \\ \text{ASCEND} \\ \text{DESCENT} \end{array} \right\} \right] \right] \\ \left[ \text{THRESH} = e \right] \left[ \left\{ \begin{array}{l} \text{TIME} \\ \text{FREQ} \end{array} \right\} = \left\{ \begin{array}{l} \text{ALL} \\ r \end{array} \right\} \right] \left[ \left[ \left[ \text{AVERAGE} \right] \right] \left[ \left[ \text{AMPLITUDE} \right] \right] \left[ \left[ \text{PEAK} \right] \right] \right] \right] = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

#### Examples:

```
MODALSE= ALL
SET 100= 1, 3, 4, 5, 7
MODALSE (ESORT=ASCEND, THRESH=.0001) = 100
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of modes for each frequency or time step.
SORT2	Output will be presented as a tabular listing of frequencies or time steps for each mode.
PRINT	Write the results to the .f06 file (Default).
NO PRINT	Do not write the results to the .f06 file.
PUNCH	Write the results to the punch (.f07) file.
ESORT	Present the modal energies sorted by mode number, ascending energy value or descending energy value (Default is MODE).
THRESH	Write out only those energies greater than e (Default = 0.001).
TIME or FREQ	Compute energies at all time steps or frequencies or the set of frequencies defined by SET r (Default = ALL).
AVERAGE	Requests average energy in frequency response analysis only (Default).
AMPLITUDE	Requests amplitude of energy in frequency response analysis only.
PEAK	Requests peak energy for frequency response analysis only. PEAK is the sum of AVERAGE and AMPLITUDE.
ALL, n, NONE	Compute modal energies for all modes, the modes defined on SET n, or (3) no modes.



**Remarks:**

1. Modal strain energy calculations will be limited to SOLs 112 (modal transient response) and 111 (modal frequency response).
2. The MODES descriptor selects from the set of the modes prescribed by the combination of Case Control command MODESELECT, and user parameters PARAM,LMODES; PARAM,LFREQ; and PARAM,HFREQ. If a mode is selected outside this set, a User Warning Message is issued.
3. The TIME (or FREQ) descriptor selects from the set of the time steps (or forcing frequencies) prescribed by the OTIME (or OFREQ) Case Control command. If a time or frequency is selected outside this set, a User Warning Message is issued.
4. For frequency response analysis, the energy may be output as real or complex values. When REAL is specified, the output will be one of AVERAGE, AMPLITUDE or PEAK energy. (See Remark 4 of the ESE Case Control Command for a definition of these terms.) When IMAG or PHASE is specified, the output will be complex energy values.

**Output Format:**

The output formats for complex energy in frequency response analysis follow.

**For SORT1 option:**

FREQUENCY = 2.100000E+01			
M O D A L S T R A I N E N E R G Y (REAL/IMAGINARY)			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	1.171343E-04 0.0	1.409326E-02 0.0	1.065749E-02 0.0
2	2.049217E-05 0.0	2.465558E-03 0.0	1.864483E-03 0.0
3	5.175167E-04 0.0	6.226612E-02 0.0	4.708635E-02 0.0
4	1.732168E-05 0.0	2.084095E-03 0.0	1.576016E-03 0.0

This format repeats for each frequency.

**For SORT2 option:**

MODE NUMBER = 1			
M O D A L S T R A I N E N E R G Y (MAGNITUDE/PHASE)			
FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	8.147644E-04 0.0	1.000000E+00 0.0	9.924866E-01 0.0
2.000000E+00	4.066132E-03 0.0	1.000000E+00 0.0	9.936857E-01 0.0
3.000000E+00	1.411670E-02 0.0	1.000000E+00 0.0	9.955440E-01 0.0

This format repeats for each mode number.

The output formats for AVERAGE energy in frequency response analysis follow. The particular type (AVERAGE, AMPLITUDE or PEAK) is identified in the output title.

### For SORT1 option:

```
FREQUENCY = 2.100000E+01
```

MODAL STRAIN ENERGY ( AVERAGE )			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	5.856715E-05	1.409326E-02	1.065749E-02
2	1.024608E-05	2.465558E-03	1.864483E-03
3	2.587583E-04	6.226612E-02	4.708635E-02
4	8.660840E-06	2.084095E-03	1.576016E-03

This format repeats for each frequency.

### For SORT2 option:

```
MODE NUMBER = 1
```

MODAL STRAIN ENERGY ( AVERAGE )			
FREQUENCY	ACTUAL	NORMALIZED	FRACTIONAL
1.000000E+00	4.073822E-04	1.000000E+00	9.924866E-01
2.000000E+00	2.033066E-03	1.000000E+00	9.936857E-01
3.000000E+00	7.058351E-03	1.000000E+00	9.955440E-01
4.000000E+00	2.872413E-02	1.000000E+00	9.977741E-01

This format repeats for each mode number.

The output formats for transient response analysis follow.

### For SORT1 option:

```
TIME STEP = 1.000000E+00
```

MODAL STRAIN ENERGY			
MODE NUMBER	ACTUAL	NORMALIZED	FRACTIONAL
1	1.483182E-02	3.610759E-01	2.316040E-01
2	4.107675E-02	1.000000E+00	6.414274E-01
3	2.165481E-04	5.271792E-03	3.381472E-03
4	1.387959E-05	3.378941E-04	2.167345E-04

This format repeats for each time step.

### For SORT2 option:

```
MODE NUMBER = 2
```

MODAL STRAIN ENERGY			
TIME STEP	ACTUAL	NORMALIZED	FRACTIONAL
0.000000E+00	0.0	0.0	0.0
9.999999E-02	2.512383E-03	1.000000E+00	9.420497E-01
2.000000E-01	8.475699E-03	1.000000E+00	9.56881E-01
3.000000E-01	1.415435E-02	1.000000E+00	9.568216E-01

This format repeats for each mode number.

### Processing:

Modal strain energy is calculated using the following relations:



$$[\text{Actual SE}] = \frac{1}{2} \{u_h\}^T [M_{hh}] \{u_h\}$$

For frequency response analysis real output, see Remark 4 of the [ESE \(Case\)](#) Case Control command for energy calculation relations.

$$\begin{aligned} [\text{Normalized Strain Energy}] &= \text{norm}[\text{Actual Strain Energy}], \text{ normalized per column.} \\ [\text{Fractional Strain Energy}] &= [\text{Normalized Strain Energy}] / [\text{diagonal} [\{1.0\}^T [\text{Normalized Strain Energy}]]], \text{ term-by-term division.} \end{aligned}$$

**MODES****Subcase Repeater**

Repeats a subcase.

**Format:**

MODES=n

**Example:**

MODES=3

Descriptor	Meaning
n	Number of times the subcase is to be repeated (Integer > 0).

**Remarks:**

1. This Case Control command can be illustrated by an example. Suppose stress output is desired for the first five modes only, and displacements for the next two modes, and forces for the remaining modes. The following example would accomplish this:

```
SUBCASE 1 $ FOR MODES 1 THRU 5
MODES = 5
STRESS = ALL
SUBCASE 6 $ FOR MODES 6 AND 7
DISPLACEMENTS = ALL
MODES = 2
SUBCASE 8 $ FOR MODE 8 AND REMAINING MODES
FORCE = ALL
```

2. This command causes the results for each mode to be considered as a separate, successively numbered subcase, beginning with the subcase number containing the MODES command. In the preceding example, this means that subcases 1, 2, 3, etc. are assigned to modes 1, 2, 3, etc., respectively.
3. If this command is not used, eigenvalue results are considered to be a part of a single subcase. Therefore, any output requests for the single subcase will apply for all eigenvalues.
4. All eigenvectors with mode numbers greater than the number of subcases defined in the Case Control Section are printed with the descriptors of the last subcase. For example, to suppress all printout for modes beyond the first three, the following Case Control commands could be used:

```
SUBCASE 1
MODES = 3
DISPLACEMENTS = ALL
SUBCASE 4
DISPLACEMENTS = NONE
BEGIN BULK
```

5. This command may be of no use in non-eigenvalue analysis and may cause data recovery processing to be repeated.

## MODESELECT

### Computed Mode Selection

Selects a subset of the computed modes for inclusion or exclusion in modal dynamic analysis.

#### Format:

Mode selection based on arbitrary mode numbers:

$$\text{MODESELECT} \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right) = n$$

#### Alternate Format 1:

Mode selection based on number of lowest modes:

$$\text{MODESELECT} \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right) \text{LMODES} = lm$$

#### Alternate Format 2:

Mode selection based on range of mode numbers:

$$\text{MODESELECT} \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right) [\text{LMODENM} = lom][\text{HMODENM} = him]$$

#### Alternate Format 3:

Mode selection based on frequency range:

$$\text{MODESELECT} \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right) [\text{LFREQ} = lof][\text{HFREQ} = hif]( [\text{UNCONSET}] = m )$$

#### Alternate Format 4:

Mode selection based on modal effective mass fraction (MEFFMFRA) criteria:

$$\text{MODESELECT} \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right) \left[ \begin{array}{l} [\text{T1FR} [= t1fr]][\text{T2FR} [= t2fr]][\text{T3FR} [= t3fr]] \\ [\text{R1FR} [= r1fr]][\text{R2FR} [= r2fr]][\text{R3FR} [= r3fr]] \\ [\text{ALLFR} [= allfr]][\text{UNCONSET} = m] \left[ \begin{array}{c} \text{SUM} \\ \text{ANYMIN} \\ \text{ALLMIN} \end{array} \right] \end{array} \right]$$

See Remark 14. for examples illustrating the use of the preceding formats.

Describer	Meaning
STRUCTURE	References computed modes of the structure (Default).
FLUID	References computed modes of the fluid.
$n > 0$	Set identification of a previously appearing SET command. ONLY those modes whose mode numbers are in SET $n$ will be included in the analysis. If SET $n$ is not defined, then ONLY mode $n$ will be included in the analysis (Integer).
$n < 0$	$ n $ refers to the set identification of a previously appearing SET command. The modes whose mode numbers are in SET $ n $ will be EXCLUDED from the analysis. If SET $ n $ is not defined, then mode $ n $ will be EXCLUDED from the analysis (Integer).
lm	Number of lowest modes that are to included. (Integer > 0).
lom	Lower limit of the mode number range for selecting the modes. See Remark 5. (Integer > 0).
him	Upper limit of the mode number range for selecting the modes. See Remark 5. (Integer > lom > 0).
lof	Lower limit of the frequency range for selecting the modes. See Remark 6. (Real $\geq$ 0.0).
hif	Upper limit of the frequency range for selecting the modes. See Remark 6. (Real > lof $\geq$ 0.0).
UNCONSET	Specifies a single mode or a set of modes for unconditional_inclusion or exclusion, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes.
$m > 0$	Set identification of a previously appearing SET command. Modes whose mode numbers are in SET $m$ will be included in the analysis, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes. If SET $m$ is not defined, then mode $m$ will be included in the analysis, regardless of the selection criterion, and regardless of the inclusion or exclusion of other modes (Integer).
$m < 0$	$ m $ refers to the set identification of a previously appearing SET command. Modes whose mode numbers are in SET $ m $ will be excluded from the analysis, regardless of the selection criterion and regardless of the inclusion or exclusion of other modes. If SET $ m $ is not defined, then mode $ m $ will be excluded from the analysis, regardless of the selection criterion and regardless of the inclusion or exclusion of other modes (Integer).
TiFR / RiFR	Flags explicitly listing components whose modal effective mass fraction (MEFFMFRA) values are to be considered for mode selection.
<i>tifr / rifr</i>	Threshold values for the listed TiFR / RiFR components. See Remark 8. (0.0 < Real $\leq$ 1.0).
ALLFR	Flag indicating that the MEFFMFRA values of components not explicitly listed by the TiFR / RiFR flags are also to be considered for mode selection.

Describer	Meaning
allfr	Threshold value for components not explicitly listed by the TiFR / RiFR flags. See Remark 8. ( $0.0 < \text{Real} \leq 1.0$ ).
SUM	For each specified component, the modes are selected as follows:  The modes are first sorted in descending order of the corresponding MEFFMFRA values. Then, starting from the first mode in this sorted list, the modes are selected until the sum of corresponding MEFFMFRA values equals or just exceeds the threshold value for that component (Default).
ANYMIN	Any mode whose MEFFMFRA value for any specified component equals or exceeds the threshold value for that component will be selected.
ALLMIN	Any mode whose MEFFMFRA values for all of the specified components equal or exceed the corresponding threshold values for those components will be selected.

**Remarks:**

1. This command is meaningful only in modal dynamic analysis (SOLs 110, 111, 112, 145, 146, and 200). It is ignored in all other analyses.
2. Only one MODESELECT command is allowed for the structure and only one MODESELECT command is allowed for the fluid and these should be specified above the subcase level.
3. The various formats of this command may not be combined.
4. The computed modes used for mode selection include the augmented modes (if any) resulting from residual vector calculations.
5. If LMODENM is specified without HMODENM, a default value of 10000000 (ten million) is assumed for HMODENM. If HMODENM is specified without LMODENM, a default value of 1 is assumed for LMODENM.
6. If LFREQ is specified without HFREQ, a default value of  $1.0\text{E}+30$  is assumed for HFREQ. If HFREQ is specified without LFREQ, a default value of 0.0 is assumed for LFREQ.
7. If the format involving the MEFFMFRA criteria is employed, it is not necessary to specify a MEFFMASS Case Control command or, even if such a command is specified, to explicitly request the calculation of the modal effective mass fractions. In the absence of such a command or request, the program will automatically perform the necessary calculations internally to ensure that the required modal effective mass fractions are computed.
8. If the T1FR / R1FR / ALLFR keywords are specified without the corresponding *tifr* / *r1fr* / *allfr* threshold values, then a default value of 0.95 (that is, 95%) is assumed for these threshold values. If the selection criterion is SUM, and a default value of 0.05 (that is, 5%) is assumed if the selection criterion is ANYMIN or ALLMIN.
9. The modal effective mass for a given mode is a measure of how much mass is associated with that mode, and indicates the sensitivity of that mode to base excitation. Modal effective mass is meaningful only for fixed base modes. If a structure is not restrained, all the modal effective mass will be associated with its rigid body modes.

10. When the MODESELECT Case Control command is used in conjunction with the parameters LMODES/LMODESFL, LFREQ/LFREQFL, and HFREQ/HFREQFL, the hierarchy of their usage is as follows:
  - a. If there is a MODESELECT Case Control command, it takes precedence over the parameters LMODES/LMODESFL, LFREQ/LFREQFL, and HFREQ/HFREQFL. (It does not matter whether these parameters are defined directly via PARAM entries, or indirectly using the FLSFSEL Case Control command.)
  - b. If there is no MODESELECT Case Control command, then parameter LMODES/LMODESFL takes precedence over parameters LFREQ/LFREQFL and HFREQ/HFREQFL. In this case, the number of lowest modes specified by LMODES/LMODESFL will be included in the modal dynamic analysis.
  - c. If there is no MODESELECT Case Control command and no LMODES/LMODESFL parameter, then parameters LFREQ/LFREQFL and HFREQ/HFREQFL are honored. In this case, all of the computed modes whose frequencies are in the range specified by LFREQ/LFREQFL and HFREQ/HFREQFL will be included in the modal dynamic analysis.
  - d. If there is no MODESELECT Case Control command and no LMODES/LMODESFL, LFREQ/LFREQFL, or HFREQ/HFREQFL parameter, then all of the computed modes will be included in the modal dynamic analysis.
11. If a subset of the computed modes is selected for subsequent use in the modal dynamic analysis, the user is informed of this by a User Information Message. Also, a new eigenvalue table indicating the actual modes selected for the analysis is output. If the user has employed a MODESELECT command involving the MEFFMFRA criteria, the modal effective mass fractions for the selected modes are also output.
12. If the mode selection criterion results in no modes being selected for subsequent use in the modal dynamic analysis, the program terminates the job with a fatal message indicating that no modal formulation is possible.
13. If the use of the MODESELECT command results in the selection of all of the computed modes for subsequent use, the user is informed of this by a User Information Message.
14. The following examples illustrate the use of the various formats of the MODESELECT command described above.

### Examples Illustrating Mode Selection Based on Arbitrary Mode Numbers:

```
$ INCLUDE ONLY STRUCTURE MODES 7, 9 AND 12 IN THE ANALYSIS
SET 100 = 7, 9, 12
MODESELECT = 100
$ EXCLUDE FLUID MODES 5 AND 6 FROM THE ANALYSIS
SET 200 = 5, 6
MODESELECT (FLUID) = -200
$ EXCLUDE STRUCTURE MODE 5 FROM THE ANALYSIS
MODESELECT = -5 $ (SET 5 NOT DEFINED)
```



**Examples Illustrating Mode Selection Based on Number of Lowest Modes:**

```

$ INCLUDE THE LOWEST 10 STRUCTURE MODES IN THE ANALYSIS
MODESELECT (LMODES = 10)
$ INCLUDE THE LOWEST 5 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID LMODES = 5)

```

**Examples Illustrating Mode Selection Based on Range of Mode Numbers:**

```

$ INCLUDE ONLY STRUCTURE MODES 10 THRU 20 IN THE ANALYSIS
MODESELECT (LMDENM = 10 HMDENM = 20)
$ INCLUDE ALL STRUCTURE MODES HIGHER THAN THE 6th MODE
$ IN THE ANALYSIS
MODESELECT (LMDENM = 7)
$ INCLUDE THE LOWEST 10 FLUID MODES IN THE ANALYSIS
MODESELECT (FLUID HMDENM = 10)

```

**Examples Illustrating Mode Selection Based on Frequency Range:**

```

$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ IN THE RANGE OF 0.1 HZ. TO 100.0 HZ. IN THE ANALYSIS
MODESELECT (LFREQ = 0.1 HFREQ = 100.0)
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR BELOW 50.0 HZ., BUT INCLUDE THE 10th AND 11th
$ MODES REGARDLESS OF THEIR CYCLIC FREQUENCIES
SET 1000 = 10, 11
MODESELECT (HFREQ = 50.0 UNCONSET = 1000)
$ INCLUDE ALL STRUCTURE MODES WITH CYCLIC FREQUENCIES
$ EQUAL TO OR ABOVE 5.0 HZ., BUT EXCLUDE THE 6 MODE
$ REGARDLESS OF ITS CYCLIC FREQUENCY
MODESELECT (LFREQ = 5.0 UNCONSET = -6) $ SET 6 NOT DEFINED

```

**Examples Illustrating Mode Selection Based on Modal Effective Mass Fraction (MEFFMFRA) Criteria:**

```
MODESELECT (T3FR)
```

The default selection criterion of SUM is assumed, and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T3.

As many modes with the highest MEFFMFRA(T3) values as possible, such that the sum of the values is equal to or just exceeds 0.95, will be selected.

```
MODESELECT (T1FR = 0.90 T2FR R3FR = 0.85)
```

The default selection criterion of SUM is assumed, and a default value of 0.95 (95%) is therefore assumed for the threshold value for component T2.

As many modes with the highest MEFFMFRA(T1) values as possible, such that the sum of the values is equal to or just exceeds 0.90, will be selected.

Similarly, as many modes with the highest MEFFMFRA(T2) values as possible, such that the sum of those values is equal to or just exceeds 0.95, will be selected.

As many modes with the highest MEFFMFRA(R3) values as possible, such that the sum of those values is equal to or just exceeds 0.85, will be selected.

```
MODESELECT (T1FR T3FR = 0.10 UNCONSET = -6 ANYMIN)
$ SET 6 NOT DEFINED
```

Since the selection criterion is specified as ANYMIN, a default value of 0.05 (5%) is assumed for the threshold value for component T1.

All modes, excluding mode 6, whose:

```
MEFFMFRA(T1) values are equal to or greater than 0.05 OR
MEFFMFRA(T3) values are equal to or greater than 0.10
```

will be selected.

```
SET 1000 = 20, 30
MODESELECT (T2FR = 0.1 R3FR = 0.15 ALLFR UNCONSET = 1000 ALLMIN)
```

The ALLFR flag indicates that the T1, T3, R1, and R2 components which are not explicitly specified above must also be considered in mode selection. Since the selection criterion is specified as ALLMIN, a default value of 0.05 (5%) is assumed for the threshold value for these components.

All modes whose:

```
MEFFMFRA(T1) values equal or exceed 0.05 AND
MEFFMFRA(T2) values equal or exceed 0.10 AND
MEFFMFRA(T3) values equal or exceed 0.05 AND
MEFFMFRA(R1) values equal or exceed 0.05 AND
MEFFMFRA(R2) values equal or exceed 0.05 AND
MEFFMFRA(R3) values equal or exceed 0.15
```

will be selected.

Modes 20 and 30 will be selected regardless of their MEFFMFRA values.

**MODTRAK**

## Mode Tracking Request

Selects mode tracking options in design optimization (SOL 200).

**Format:**

MODTRAK = n

**Example:**

MODTRAK=100

Describer	Meaning
n	Set identification of a MODTRAK Bulk Data entry (Integer > 0).

**Remark:**

1. Selection of a MODTRAK Bulk Data entry with the MODTRAK Case Control command activates mode tracking for the current subcase. This request is limited to normal modes subcases (ANALYSIS=MODES) in design optimization (SOL 200).

## MONCARL

### Control for Monte-Carlo simulation

Control for Monte-Carlo simulation using the non-parametric variability method (NPVM). This Case Control command can be used in SOL 111 only.

#### Format:

MONCARL ( [NSAMPLE=m] , [SEED=k] , [OFFD= r ] ) = n

#### Example:

MONCARL (NSAMPLE=100) =5

Descriptor	Meaning
NSAMPLE = m	Specifies the number of random samples, m ( Integer >=0, Default = 0).
SEED = k	Specifies the random number generator seed, k (Integer > 0, Default = 777).
OFFD = r	Specifies the scale factor for off-diagonal entries of the random matrix, r (Real, Default = 1.0).
n	Set identification number of a MONCARL Bulk Data entry (Integer > 0).

#### Remarks:

1. Only one MONCARL command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. Keeping in mind the size and the naively parallel nature of Monte-Carlo problem typical simulation would be run in a parallel mode. The parallel DMP simulation is restricted to Multi-Master mode in order to get a good scalable performance. Multi-Master mode can be specified using "RUNOPT=MULTIPAR", in "DOMAINSOLVER" Executive Control statement, e.g.:  
DOMAINSOLVER ACMS (PARTOPT=DOF, RUNOPT=MULTIPAR)
3. Running the Monte-Carlo simulation in Nastran would result in a concatenated OP2, HDF5 and PCH file containing the response results of each random sample of Monte-Carlo simulation. Each random sample would appear as a unique subcase in the OP2, HDF5 and PCH file.
4. PLT Viewer (provided with Nastran) can be used to read the result of Monte-Carlo simulation to generate meaningful response statistics. Currently the PLT Viewer can only read HDF5 and "PCH files in SORT 1 format" (not OP2).
5. HDF5 is the recommended output method for reading into PLT Viewer. Use NASTRAN system cell (702) HDF5=1 to request HDF5 output.

## MONITOR

### Print Selection for Monitor Data

Specifies options in the printing of monitor data.

#### Format:

$$\text{MONITOR}[(\text{REAL or IMAG}, \text{PHASE}, \text{NODSP1}, \text{NOPNT1}, \text{NOPNT2}, \text{NOPNT3})] = \begin{cases} \text{ALL} \\ \text{NONE} \end{cases}$$

#### Example:

```
MONITOR (PHASE, NOPNT1) =ALL
MONITOR (IMAG, NODSP1) =ALL
```

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output (Default).
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
NODSP1	Do not include MONDSP1 results in the MONITOR point prints (default is to provide these prints).
NOPNT1	Do not include MONPNT1 results in the MONITOR point prints (default is to provide these prints).
NOPNT2	Do not include MONPNT2 results in the MONITOR point prints (default is to provide these prints).
NOPNT3	Do not include MONPNT3 results in the MONITOR point prints (default is to provide these prints).
ALL	Print all monitor point results, except for those deselected.
NONE	Do not print monitor point results

#### Remarks:

1. The MONITOR command is required in order to obtain MONITOR results in the printed output in SOLs 101, 103, 108, 109, 111, 112, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. SOL 144 does not require the MONITOR command and is ignored.
3. The MONITOR command should be above the subcase level or in the first subcase (above the step level for SOL 400). MONITOR commands in subcases subsequent to the first one or in any steps for SOL 400 are ignored.
4. MONPNT1 is not applicable in SOL 103.

5. The MONITOR command is not required for SOL 144 except in the special case of MONPNT3 with any exclusion flags other than SMAD. Monitor points with exclusion flags require special processing of the GRID point force data and are not recommended because of the potential excessive cost.
6. Results for dynamic solutions (SOLs 108, 109, 111, 112, 146 and 400 with NLTRAN) will be in SORT2 format. All other solutions will be in SORT1 format.

### Output Format:

In frequency response analysis, there is no output format of complex energy.

### For SOLS 101, 103, and 144 - SORT1:

```

S T R U C T U R A L   I N T E G R A T E D   F R E E   B O D Y   M O N I T O R   P O I N T   L O A D S (MONPNT3)
MONITOR POINT NAME = FORCSM1                                SUBCASE NO.      1
LABEL = SUMMATION OF FORCES AT GRID 9
CP = 0                                X = 8.000000E+01      Y = 0.000000E+00      Z = 0.000000E+00

  AXIS   REST.  APPLIED
  ----   -
  CX     1.600000E+01
  CY     1.600000E+01
  CZ     1.600000E+01
  CMX    0.000000E+00
  CMY   -3.200000E+02
  CMZ    3.200000E+02
    
```

This format repeats for each monitor point and repeats for each subcase or mode.

### SOLS 108, 109, 111, 112 and 146 - SORT2:

```

S T R U C T U R A L   I N T E G R A T E D   F R E E   B O D Y   M O N I T O R   P O I N T   L O A D S (MONPNT3)
MONITOR POINT NAME = FORCSM1                                COMPONENT = CX                                SUBCASE NO.      1
LABEL = SUMMATION OF FORCES AT GRID 9
CP = 0                                X = 8.000000E+01      Y = 0.000000E+00      Z = 0.000000E+00

  TIME STEP   RESULTANT
  ----
  0.000000E+00  0.000000E+00
  1.000000E-02  7.692307E-01
  2.000000E-02  1.301775E+00
  3.000000E-02  1.137915E+00
  4.000000E-02  6.557894E-01
  5.000000E-02  9.679956E-01
  6.000000E-02  1.354058E+00
  7.000000E-02  9.230635E-01
  8.000000E-02  6.696146E-01
  9.000000E-02  1.178594E+00
  9.999999E-02  1.275434E+00
    
```

This format repeats for each monitor point.

**MPC****Multipoint Constraint Set Selection**

Selects a multipoint constraint set.

**Format:**

MPC = n

**Example:**

MPC=1 7

Describer	Meaning
n	Set identification number of a multipoint constraint set. This set identification number must appear on at least one MPC or MPCADD Bulk Data entry (Integer > 0).

**Remarks:**

1. In cyclic symmetry analysis, this command must appear above the first subcase command.
2. Multiple boundary conditions (MPC sets) are not allowed in superelement analysis. If more than one MPC set is specified per superelement (including the residual), then the second and subsequent sets will be ignored.
3. In addition to select MPC/MPCADD bulk data entries, MPC=n can also be used to select a group of rigid elements for the analysis via SET3,n bulk data entry with RBEin or RBEex in DES field of SET3 bulk data entry.
4. SET3,n is not needed if ALL rigid elements in the model are to be utilized.
5. Rigid element set section is supported in SOL 400 if 'RIGID=LINEAR' is present in case control deck.

## MPCFORCES

## Multipoint Forces of Constraint Output Request

Requests the form and type of multipoint force of constraint vector output.

### Format:

$$\text{MPCFORCES} \left( \left( \begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix} \right), \left[ \begin{matrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{matrix} \right], \left[ \begin{matrix} \text{REAL or IMAG} \\ \text{PHASE} \end{matrix} \right], \left[ \begin{matrix} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{matrix} \right], \right. \\ \left. \left[ \begin{matrix} \text{RPRINT} \\ \text{NORPRINT} \end{matrix} \right], \text{RPUNCH}, [\text{CID}] \right) = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix} \right\}$$

### Examples:

```
MPCFORCES=5
MPCFORCES (SORT2, PUNCH, PRINT, IMAG) =ALL
MPCFORCES (PHASE) =NONE
MPCFORCES (SORT2, PRINT, PSDF, CRMS, RPUNCH) =20
MPCFORCES (PRINT, RALL, NORPRINT) =ALL
```

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.
PRINT or (blank)	Printer File (.f06)
PUNCH	Punch File (.pch)
PLOT	Plot File (.op2/.h5)

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAGE Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.



Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests that the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis post-processing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control Section. See Remark 8.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request the printing of output coordinate system ID in printed output file (.f06) file.
ALL	Multipoint forces of constraint for all points will be output. See Remarks 2. and 5.
NONE	Multipoint forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only multipoint constraint forces for points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
3. MPCFORCES=NONE overrides an overall output request.
4. In SORT1 format, MPCFORCES recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, then it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, the line will not be printed. If a sextet cannot be formed, then zero values may be output.

5. MPCFORCE results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
6. MPCFORCE results due to linear elements only are available in SOL 129 with MDLPRM, MPCF129, 1.
7. In inertia relief analysis, the MPCFORCE output includes both the effects of applied and inertial loads.
8. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
9. Note that the CID keyword affects only grid point related output, such as DISplacement, VELO, ACCE, OLOAD, SPCForce and MPCF. In addition, the CID keyword needs to appear only once in a grid related output request anywhere in the Case Control Section to turn on the printing algorithm.

**MPRES**

## Fluid Pressure Output Request

Requests the pressure for selected wetted surface elements when virtual mass (MFLUID) is used.

**Format:**

$$\text{MPRES} \left( \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right), \left( \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

MPRES=5

MPRES (IMAG) =ALL

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Fluid pressures for all elements will be output.
NONE	Fluid pressures for no elements will be output.
n	Set identification number of a previously appearing SET command. Only fluid pressures for elements in this set will be output (Integer >0).

**Remark:**

1. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 is also required in the modal solution sequences (SOLs 111, 112, 146, and 200).

## NLBUCK

## Nonlinear Buckling Request

Perform a nonlinear buckling analysis in SOL 400.

### Format:

$$NLBUCK = \left[ \begin{array}{c} \text{END} \\ \text{ALL} \\ r \end{array} \right]$$

### Examples:

```
NLBUCK
NLBUCK=END
NLBUCK=ALL
NLBUCK=0.3
```

Descriptor	Meaning
END	At the end of the step an eigenvalue projection is made to predict the buckled load (Default).
ALL	After each converged load increment within the step an eigenvalue projection is made to predict the buckled load.
r	After every multiple of r load step an eigenvalue projection is made to predict the buckled load. Buckling will also be computed at the final load increment. A tolerance of 1.E-6 is used to determine the load step's closeness to r. (Real)

### Remark:

1. The eigenvalue projection is only attempted after the first two converged load increments in order for there to be at least two tangent stiffness matrices for the projection.
2. NLBUCK may be specified in any STEP or SUBCASE along with an NLSTEP command. The NLPARM command is not allowed with NLBUCK.
3. In order to obtain the most accurate value for critical buckling factor, the user must set NO=1 on the NLSTEP entry. Specifying PARAM,LGDISP>0 is also recommended.
4. In addition to NLBUCK, a METHOD Case Control command may be specified to reference an EIGB, EIGR or EIGRL Bulk Data entry, or if asymmetric follower stiffness exists then a CMETHOD Case Control command may be specified to reference an EIGC entry. The EIGx entries should request the computation of at least the first mode. If no METHOD/CMETHOD command is present, a minimum of two eigenvalues are extracted for the buckling projection using an eigenvalue extraction method appropriate for the form of the tangent stiffness (symmetric or asymmetric). For manual control of follower stiffness symmetry see the description of user PARAMETERS FOLLOWWK and FKSVMFAC.

5. If NLBUCK=0.3 and the TOTTIME on the NLSTEP Buck Data entry is 1.0 then the buckling will be computed at load steps 0.3, 0.6, 0.9, and 1.0. Also, if the load factor is not a multiple of  $r$  then the next load factor will be used; for example, if TOTIME=1.0 and there are 50 load increments and  $r=.45$  then buckling will be computed at load factors 0.46, 0.90, and 1.0.
6. There are three methods of eigenvalue extraction available for nonlinear buckling—Lanczos (EIGRL or EIGR entry with METHOD=LAN), enhanced inverse power method (EIGB entry with METHOD=SINV), and complex (EIGC entry) for unsymmetric stiffness due to follower stiffness.
  - a. If no METHOD and no CMETHOD command is specified, then the program will automatically attempt to compute two modes (ND=2) with an unspecified eigenvalue range (F1 and F2) using the real Lanczos method if the stiffness matrix is symmetric or the complex Lanczos method if the stiffness matrix is unsymmetric.
  - b. The Lanczos method is recommended in most cases especially in finding the lowest mode.
  - c. If no modes can be found with no eigenvalue range was specified, then it is highly recommended that a range (L1 and L2 on EIGB, F1 and F2 on EIGR, and V1 and V2 on EIGRL) be specified.
  - d. If higher modes are desired, then the enhanced inverse power method is recommended with a narrow eigenvalue range specified for L1 and L2 on the EIGB entry.
  - e. If a METHOD command is specified but the stiffness is unsymmetric then User Warning Message 9430 will be issued.
7. Guidelines and limitations:
  - a. NLPARAM Case Control command is not permitted in a nonlinear buckling step. NLSTEP must be specified in the nonlinear buckling step and KMETHOD=PFNT (default) is strongly recommended along with NO=1 for FIXED time stepping or INTOUT=YES for ADAPT time stepping. If KMETHOD=ITER then KSTEP=1 is strongly recommended.
  - b. It is strongly recommended that PARAM,LGDISP,1 is specified.
  - c. It is strongly recommended to specify an eigenvalue range on the EIGR, EIGRL, EIGB, and EIGC Bulk Data entries. But with NLBUCK=ALL it may be difficult to define a range for all load increments.

## NLHARM

### Nonlinear Harmonic Analysis Parameter Selection

Selects the parameters used for nonlinear harmonic response analysis.

#### Format:

NLHARM = n

#### Example:

NLHARM = 79

Describer	Meaning
n	Set identification of NLHARM Bulk Data entry. (Integer > 0)

#### Remarks:

1. This entry references the NLHARM Bulk Data entry which defines the number of subharmonics and harmonics to be used in the analysis as well as referencing the excitation frequency specification.
2. Gyroscopic terms are included if the RGYRO Case Control command and Bulk Data entry are specified. If gyroscopic terms are desired, the user must specify the constant roter speed (ASYNC option) for the reference rotor.

**NLIC****Nonlinear Initial Condition or Preload**

Selects a previously executed load increment as the initial conditions or preload for a nonlinear or perturbation STEP in SOL 400.

**Format:**

$$\text{NLIC} \left[ \text{SUBCASE } i \left[ , \text{STEP } j \left[ , \left[ \begin{array}{c} \text{LOADFACT} \\ \text{TIME} \end{array} \right] \left[ \begin{array}{c} (T = \text{tol}) \\ \text{NEAR} \end{array} \right] f \right] \right] \right]$$
**Example:**

```
NLIC SUBCASE 100 STEP 10 LOADFACT 0.8
NLIC SUBCASE 100 STEP 10 LOADFACT (t=0.01) 0.66
NLIC STEP 10 TIME 12.0
NLIC TIME NEAR 7.8
```

Describer	Meaning
SUBCASE	Keyword to select the SUBCASE ID
i	Specifies the identification number of a previously executed subcase. (Integer > 0; Default is the subcase where the current NLIC is located).
STEP	Keyword to select the STEP ID.
j	Specifies the identification number of a previously executed STEP (Integer > 0, Default is the last STEP).
LOADFACT	Keyword to select load factor or time. Both keywords are equivalent.
TIME	
f	Specifies the load factor of a previously executed load increment in linear or nonlinear static analysis (Real > 0.0, see Remark 3 for default).
T	Keyword to select the tolerance.
NEAR	Keyword for the nearest load factor or time to f.
tol	Specified the load factor or time tolerance to f. (Real ≥ 0.0; Default = 1.0E-06)

**Remarks:**

1. The NLIC command can only point to a load increment whose output flag is on - an available restart point in the static analysis. If NLIC is not pointing to an available restart point, a fatal error will be issued and the job will be terminated.
2. The NLIC can be used in nonlinear static, nonlinear transient and perturbation analysis. If SUBCASE referred in NLIC is not the current SUBCASE, this usage is not recommended. The system(779) = 1 may be used to avoid this usage, which stops the job in this case.

3. For nonlinear transient analysis (ANALYSIS=NLTRAN), NLIC can only appear in the first transient analysis STEP in a SUBCASE. (Note that the first transient analysis step may not be the first step of subcase.) Otherwise, it will be ignored and a warning message will be issued.
4. For linear perturbation analysis, NLIC must point to a previous NLSTATIC step in the same SUBCASE.
5. If NLIC is specified without any of the keywords, or NLIC is not present in a nonlinear STEP or perturbation STEP, the initial condition or PRELOAD is taken from the last available restart point in the immediate previous static step.
6. In the same nonlinear transient step, NLIC cannot appear together with the IC Case Control command. A fatal error message will be issued if NLIC and IC appear in the same step. Please note that IC is meaningful only in the first STEP of a SUBCASE, and the step is a nonlinear transient analysis.
7. When Keyword NEAR is used, the search for the nearest load factor or time is limited inside the specified SUBCASE i STEP j.
8. NLIC can be used in SOL 400 only.
9. If NLIC is used with ANALYSIS=NLSTATIC it may cause convergence problems. If NLIC is used across different subcases: the initial conditions, the deformation, the strains, the stresses, etc., are carried over between Subcases. However, the load is NOT carried over. This can cause an unbalanced load condition that may cause convergence problems.



**NLLOAD**

## Nonlinear Load Output Request

Requests the form and type of nonlinear load output for transient problems.

**Format:**

$$\text{NLLOAD}[(\text{PRINT}, \text{PUNCH})] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

NLLOAD=ALL

Describer	Meaning
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
ALL	Nonlinear loads for all solution points will be output.
NONE	Nonlinear loads will not be output.
n	Set identification of a previously appearing SET command. Only nonlinear loads for points with identification numbers that appear on this SET command will be output (Integer > 0).

**Remarks:**

1. Nonlinear loads are output only in the solution (d or h) set.
2. The output is available in SORT2 format only.
3. Both PRINT and PUNCH may be used.
4. NLLOAD=NONE allows overriding an overall output request.

## NLOPRM

## Nonlinear Regular and Debug Output Control Parameters - SOL 400

Controls MSC Nastran nonlinear solution output, debug printout, debug POST and punch-out of contact constraints of MPC and MPCY Bulk Data entries.

### Format:

NLOPRM = [OUTCTRL = {STD,SOLUTION,INTERM}]

$$\left[ \text{NLDBG} = \left\{ \begin{array}{l} \text{NONE} \\ \text{NLBASIC, NRDBG, ADVDBG, } \left\{ \begin{array}{l} \text{N3DBAS} \\ \text{N3DMED} \\ \text{N3DADV} \\ \text{N3DSUM} \end{array} \right\} \end{array} \right\} \right]$$

$$\left[ \text{DBGPOST} = \left\{ \begin{array}{l} \text{NONE} \\ \text{LTIME} \\ \text{LSTEP} \\ \text{LSUBC} \\ \text{ALL} \end{array} \right\}, \left[ \text{MPCPCH} = \left\{ \begin{array}{l} \text{NONE} \\ \text{BEGN, OTIME, STEP} \\ \text{YBEGN, YOTIME, YSTEP} \end{array} \right\} \right] \right]$$

$$\left[ \text{DELIMIT} = \left\{ \begin{array}{l} \text{No} \\ \text{Yes} \end{array} \right\} \right]$$

$$\left[ \text{GRIDINF} = \left\{ \begin{array}{l} \text{No} \\ \text{MAXGRID} \\ \text{GID} \end{array} \right\} \right]$$

### Example(s):

NLOPRM                    OUTCTRL=STD, SOLUTION                    DBGPOST=LTIME  
 NLOPRM                    OUTCTRL=(SOLUTION, INTERM),                    MPCPCH=(OTIME, STEP)

Descriptor	Meaning
OUTCTRL	Selects one or more following nonlinear solution output options
STD	Standard Nastran output (Default)
SOLUTION	Solution set output which does not include solutions for superelements

Describer	Meaning
	INTERM Intermediate nonlinear solution output in OP2 and HDF5 files, at each solution output interval, set by the user through nonlinear solution controls, such as NLPARM.
NLDBG	Keyword selecting one or more debug printout options
	NONE No debug print output (Default)
	NLBASIC Basic nonlinear information printout
	NRDBG Newton-Raphson iteration information printout
	ADVDBG Advance nonlinear debug printout
	N3DBASE For 3D contact, it requests to print out the error tolerance of each contact body in the F06 file.
	N3DMED For 3D contact, it requests to print out the summary table of all contact parameters, in addition to the printout by N3DBAS, in the .f06 file.
	N3DADV For 3D contact, it requests to print out the body contact information in long form, in addition to the printouts by both N3DBAS and N3DMED, in the F06 file.
	N3DSUM For 3D Contact, it simplifies body contact information in N3DADV.
DBGPOST	Selects one of the following debug POST options
	NONE No output (Default)
	LTIME To output all iterations in the last load or time increment
	LSTEP To output all iterations in the last STEP
	LSUBC To output all iterations in the last SUBCASE
	ALL To output all iterations
MPCPCH	Selects one or more following options to punch out contact constraint equations in terms of MPC or MPCY Bulk Data entries in Nastran PCH file. This is not available for segment-to-segment contact.
	NONE No MPC or MPCY punch output (Default)
	BEGN MPC punch in the beginning of the very first iteration
	OTIME MPC punch at every user requested output step
	STEP MPC punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE
	YBEGN MPCY punch in the beginning of the very first iteration
	YOTIME MPCY punch at every user requested output step
	YSTEP MPCY punch at the end of each load case, such as SUBSTEP, STEP and SUBCASE)
DELIMIT	Select output of delimiter for subcases, steps, increments, and iterations.

Describer	Meaning
	No No output (default)
	Yes Select output
GRIDINF	Select output of grid point information (1) displacements of the grid; (2) EID: element ID connected to this grid; (3) PID: property ID of relevant elements; (4) ID of contact body which the grid belongs to; and, (5) if the grid locates on the surface of contact body.
	No No output (default)
	MAXGRID Output information of grid which has maximum displacement component
	GID Output information of grid which user specifies

**Remarks:**

1. This Case Control command is used to better control nonlinear solution output during solution process, to provide MSC users a direct access to nonlinear solutions even the job is still running, to give the users some tools to debug the nonlinear solution process and gain some insight of nonlinear solution procedure, and to allow users to print out MPC and MPCY equations from contact constraints before and during solution process.
2. OUTCTRL=SOLUTION is designed to debug nonlinear analysis which may fail during the analysis. For other selections, some results such as displacement, SPC force, and so on, are written into F06 file only after the analysis is finished completely. This is helpful for user to understand the nonlinear analysis when job is terminated abnormally. With OUTCTRL=SOLUTION, only part of the results will be output. Availability of output request for debugging purposes are summarized at the following table.

Output Request	OP2	F06
BOUTPUT	NO	YES
DISPLACEMENT	YES	YES
GPFORCE	NO	NO
MPCFORCES	YES	YES
NLSTRESS	NO	YES
OLOAD	YES	YES
SPCFORCES	YES	YES
STRESS	NO	NO
STRAIN	NO	NO

Please note that OP2 file only includes the results at the end time of every load step, instead of the user-specified output intervals assigned in NLSTEP or NLPARM. It is recommended to use OUTCTRL=SOLUTION,INTERM to get nonlinear solutions in both F06 and intermediate OP2 files for debugging purposes.

3. For OUTCTRL = INTERM, the intermediate output is only available in OP2 file. GPFORCE is not available in intermediate OP2 files and can be obtained by OUCTRL=STD.
4. For NLDBG, all nonlinear information is printed out in only F06 file.
5. For DBGPOST, all nonlinear information is printed out in the MASTER/DBALL.
6. For MPCPCH, it is a punch output that MPC and MPCY requests cannot be mixed.
7. In MPCPCH punch output, it is restricted that SID=1 for a general contact and SID=2 for the glued contact.
8. When using MPCPCH for the permanent glued contact, user may specify any keyword of BEGN, STEP and OTIME to punch output MPC type of constraints, or any keyword of YBEGN, YSTEP and YOTIME to punch out MPCY type of constraints.
9. The output to the file is also influenced by the NLPACK Param. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```

It has total 2000 increments, and is asked output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With "intermediate output request, you will have only one OP2 file.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, you will have 100 OP2 files.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, you will have 50 OP2 files.

10. MPCPCH is not supported in thermal contact with DQNEAR because MPC constraints do not apply to the near thermal contact behavior.

## NLPARM

### Nonlinear Static Analysis Parameter Selection

Selects the parameters used for nonlinear static analysis.

#### Format:

NLPARM = n

#### Example:

NLPARM=10

Describer	Meaning
n	Set identification of NLPARM and NLPCI Bulk Data entries (Integer > 0).

#### Remarks:

1. NLPARM and NLPCI Bulk Data entries will not be used unless selected.
2. NLPARM may appear above or within a subcase.
3. For SOL 600, the only fields used are ID, NINC, DT (creep only), EPSU, EPSP, and EPSW. Use PARAM,MARCOTIM instead of INTOUT. For other fields, advanced convergence controls are available using the NLAUTO, NLSTRAT, and PARAM,MARCDEF Bulk Data entries.

## NLRESTART

## Nonlinear Restart Request

Request a RESTART execution at a specified point for SOL 400.

Please note that when running NLRESTART with Advanced Nonlinear Element, the Data Blocks for Advanced Nonlinear Element must be saved in the corresponding cold start run with the proper DBSAVE, NLPACK, and INTOUT. See Remark 12.

### Format:

$$\text{NLRESTART} \left[ \text{SUBCASE } i \left[ , \text{STEP } j \left[ , \begin{array}{l} \text{LOADFAC } f \\ \text{TIME } t \end{array} \right] \right] \right] [\text{OVERRIDE}]$$

### Example:

```
NLRESTART SUBCASE 1, STEP 2, LOADFAC 0.3
```

Descriptor	Meaning
i	Specifies the identification number of a previously executed SUBCASE (Integer; Default is the first SUBCASE).
j	Specifies the identification number of a previously executed STEP (Integer; Default is the first STEP).
f	Specifies the load factor of a previously executed load increment in nonlinear static analysis (Real; $0.0 \leq f \leq 1.0$ ; Default = 1.0).
t	Specifies the time of a previously executed time step in nonlinear transient analysis (Real; $t_0 \leq t \leq t_n$ , where $t_0$ is the initial time of STEP j, and $t_n$ is the last time of STEP j; Default = $t_n$ ).
OVERRIDE	To force the run to continue even though there are fundamental changes to the model's geometry, properties, and/or connectivity. In general, NLRESTART does not allow changes to the model's geometry, properties, or connectivity and UFM 9424 will be issued if a change is detected. Please be cautious in using OVERRIDE because the model changes may result in wrong answers and/or a fatal termination and, therefore, is not recommended in all cases.

### Remarks:

1. The NLRESTART command can be used in SOL 400 (NONLIN) only.
2. The NLRESTART command must appear before any SUBCASE command.
3. To perform a restart, the data base for the original run must be made available by using the ASSIGN File Management statement or other equivalent method.

4. The restart run can only be executed at a load increment (or time step) whose output flag is on an available restart point. (See the field INTOUT on the NLPARM Bulk Data entry, and NO on ). When a user-specified restart point is not available, the closest previous restart point that is available will be applied automatically.
5. If only NLRESTART is specified, a restart begins from the last available restart point in the previous run. Otherwise, at least one set of the SUBCASE i, STEP j, or LOADFC f (or TIME t) must be specified.
6. In static analysis, f is reset to 0.0 when  $f < 0.0$ , and it makes the restart begin from the beginning of STEP j. f is reset to 1.0 when  $f > 1.0$ , which makes the restart begin from the beginning of the next STEP (after STEP j).
7. In transient analysis, t is reset to  $t_0$  when  $t < t_0$ , and it makes the restart begin from the beginning of STEP j. t is reset to  $t_n$  when  $t > t_n$ , which makes the restart begin from the beginning of the next STEP (after STEP j).
8. The NLRESTART Case Control command must contain all of the commands used in the original execution up to the point where the restart is requested.
9. All data contained on the database from the restart point will be deleted when the restart begins.
10. When using NLRESTART, its corresponding COLD START input model should not use "PARAM,DBALL,SCRATCH", which will remove the database required for NLRESTART after completion of COLD START run.
11. NLRESTART is used for Nonlinear Restart Request. The restart step must be a nonlinear analysis, NLSTAT or NLTRAN. NLRESTART from perturbation step or thermal analysis step is not supported in SOL400.
12. DBSAVE in the Case Control Section is used to control saving of datablocks of advanced nonlinear elements for static and transient nonlinear analysis in SOL 400.
  - 1 No datablocks of advanced nonlinear elements is saved
  - 0 Saving datablocks of advanced nonlinear elements at the end of each loadcase (default)
  - >0 Saving datablocks of advanced nonlinear elements at the every nth output request of results

In DBSAVE=0, NLRESTART may start at the end of the loadcase (load step) for advanced nonlinear elements. If DBSAVE=n (>0), NLRESTART may start at the every INTOUT\* n th output point.

For NLTRAN analysis in SOL 400, the output to the file, i.e., NLRESTART usage, is also influenced by the NLPACK Param. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```



It has total 2000 increments, and is asked output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, you will have only one OP2 file.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With intermediate output request, you will have 100 OP2 files.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With intermediate output request, you will have 50 OP2 files.

Similarly, NLRESTART can be started at those output points ONLY.

13. When NLRESTART is used with Advanced Nonlinear Elements in a linear perturbation analysis, if the cold start run is not a linear perturbation analysis itself, then MDLPRM,NLRSTRTM,1 must be used in the cold start run to allow proper memory allocation between the two runs.

## NLSTEP

### Nonlinear Control Parameters for Mechanical, Thermal, and Coupled Analysis Selection

Selects integration and output time steps for static and transient nonlinear analysis in SOL 400.

#### Format:

NLSTEP=*n*

#### Example:

```
NLSTEP=10
SUBSTEP=1
ANALYSIS=HSTAT
SUBSTEP=2
ANALYSIS=NLSTAT
```

Describer	Meaning
<i>n</i>	Identification number of a NLSTEP Bulk Data entry. (Integer > 0).

#### Remarks:

1. An NLSTEP entry can be selected to execute a nonlinear static or nonlinear transient analysis in SOL 400. A NLSTEP is used in lieu of a NLPARM or both.
2. If a NLSTEP is present *anywhere* in a SUBCASE, then any NLPARM or entries in the SUBCASE *will be ignored*.
3. When used for coupled analysis, the NLSTEP must be above the first SUBSTEP command. A *single* NLSTEP entry is *used for all SUBSTEPS* of the STEP. (See the above example for use with coupled analysis.)

**NLSTRESS**

## Nonlinear Element Stress Output Request

Requests the form and type of nonlinear element stress output in SOLs 106 and 400.

**Format:**

$$\text{NLSTRESS} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right), [\text{NLOUT} = m] \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{See Remark 3} \end{array} \right\}$$

**Examples:**

```
NLSTRESS=5
NLSTRESS (SORT1, PRINT, PUNCH, PHASE)=15
NLSTRESS (PLOT)=ALL
NLSTRESS (NLOUT=23)=ALL
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load.
SORT2	Output will be presented as a tabular listing of load for each element type.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Stresses for all nonlinear elements will be output.
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No nonlinear element stress will be output (Default).
NLOUT	For SOL 400 only. Allows the selection of additional types of nonlinear output.
m	Identification of a NLOUT Bulk Data entry. (Integer > 0)

**Remarks:**

1. ALL should not be used in a transient problem due to potentially excessive output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. If there is a stress request, the default output set is that of the STRESS request.
4. For NLTRAN analysis, NLSTRESS does not support SORT1 selection.

**NONLINEAR**

## Nonlinear Dynamic Load Set Selection

Selects nonlinear dynamic load set for transient response or nonlinear harmonic response problems.

**Format:**

NONLINEAR = n

**Example:**

NONLINEAR=75

Describer	Meaning
n	Set identification of NOLINi, NLRGAP or NLRSFD Bulk Data entry (Integer > 0).

**Remark:**

1. Nonlinear force Bulk Data entries (NOLINi, NLRGAP or NLRSFD) will be ignored unless selected in the Case Control Section.
2. At least one degree of freedom must be defined on a nonlinear force entry and called up by the NONLINEAR Case Control command in nonlinear harmonic response.

## NOUTPUT

### Normal Output Request in Cyclic Symmetry Problems

Requests physical output in cyclic symmetry problems.

#### Format:

$$\text{NOUTPUT} \left\{ k, \begin{matrix} R \\ L \end{matrix} \right\} = \left\{ \begin{matrix} \text{ALL} \\ m \end{matrix} \right\}$$

#### Examples:

```
NOUTPUT (R) =ALL
NOUTPUT (2) =5
NOUTPUT (4, L) =10
```

Describer	Meaning
ALL	Output for all segments is desired.
m	Output for segments specified in SET m is desired (Integer > 0).
k	Used in eigenvalue analysis to request eigenvector and internal force output for harmonics specified in SET k (Integer > 0).
R, L	Output for only the right- or left-half of segments specified as ALL or in SET m. R and L are used in dihedral symmetry only.

#### Remarks:

1. Sets k and m are defined on SET commands.
2. In cyclic symmetry analysis, this command, or the HOUTPUT command, is required to obtain data recovery.

**NSM****Selects Nonstructural Mass Set Entries**

Selects nonstructural mass (NSM) set for mass generation.

**Format:**

NSM = n

**Example:**

NSM = 5

Describer	Meaning
n	Set identification number of a nonstructural mass that appears on a NSM, NSML, NSM1, NSML1, or NSMADD Bulk Data entry (Integer > 0).

**Remark:**

1. Different NSM sets may be selected for superelements and residuals but within a superelement or residual it may not change within the subcase structure.

## NVELOCITY

### Velocity Normal Output Request

Request output of velocity normal for structural grids in SOLs 108 and 111 only.

#### Format:

$$\text{NVELOCITY} \left[ \left( \begin{array}{l} \text{[SORT1]} \\ \text{[SORT2]} \end{array} \right), \left( \begin{array}{l} \text{[PRINT, PUNCH]} \\ \text{PLOT} \end{array} \right), \left( \begin{array}{l} \text{[REAL or IMAG]} \\ \text{[PHASE]} \end{array} \right), \text{ERP} \right] = \left. \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

NVELOCITY (PLOT, ERP) = ALL

Describer	Meaning												
SORT1	Output will be presented as tabular listing of grid points for each excitation frequency (Default).												
SORT2	Output will be presented as tabular listing of excitation frequencies for each grid point.												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											

\* The .op2 binary database file will be generated with "PARAM,POST, X" (or the POST Case Control command), while the .h5 binary database file will be generated with "MDLPRM,HDF5, X" specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAGE yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is In degrees.
ERP	Flag to accept ERP set convention.
ALL	Velocity normal will be computed for all surface structural grid points.
n	Set identification of a previously defined set of grid points. Velocity Normal will be computed for the grid points in this set only.
NONE	Velocity normal will not be processed.



Describer	Meaning
-----------	---------

**Remarks:**

1. NVELOCITY = NONE overrides an overall request.
2. The PLOT option is used if results are requested for post-processing but no printed or punched output is desired.
3. Even with NVELOCITY=all, Velocity Normal will be computed only for structural grids at the surface of the model. Fluid grids, if exist, are not included in NVELOCITY output.
4. With ERP flag in NVELOCITY command, structural grids of all ERPPNL will be considered as candidates for NVELOCITY output and subjected to further screening of NVELOCITY set.
5. For 3D model, ERPPNL points to SET3 with all PSOLID IDs in the model is the easiest way to compute NVELOCITY of structural surface GRIDs.
6. This Case Control command can be used in SOL 108 and SOL 111 only

**OFREQUENCY**

## Output Frequency Set

Selects a set of frequencies for output requests.

**Format:**

$$\text{OFREQUENCY} = \left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$$

**Examples:**

OFREQUENCY=ALL  
OFREQUENCY=15

Describer	Meaning
ALL	Output for all frequencies will be computed.
n	Set identification of a previously appearing SET command. Output for frequencies closest to those given on this SET command will be output (Integer > 0).

**Remarks:**

1. In real eigenvalue, buckling, and complex eigenvalue analyses, the OMODES Case Control command allows for an alternate way of selecting the modes to be output based on their mode numbers. In these cases, if both the OMODES and OFREQUENCY requests appear, the OMODES request takes precedence.
2. If this command is not specified in the Case Control Section (or, in the case of real eigenvalue, buckling, and complex eigenvalue analyses, if neither the OMODES nor the OFREQUENCY request is specified), then output will be generated for all frequencies.
3. The number of solutions selected will always be equal to the number of quantities in the selected set. The closest values are used.
4. In flutter analysis (SOL 145), the selected set refers to the imaginary part of the complex eigenvalues. The physical interpretation of this quantity depends on the method of flutter analysis as follows:
  - K- or KE-method: velocity (input units).
  - PK-method: frequency.
5. In aeroelastic response analysis (SOL 146) with RLOAD selection, the selected set refers to the frequency (cycles per unit time).
6. In complex eigenvalue analysis (SOLs 107 and 110), the selected set refers to the imaginary part of the complex eigenvalues.
7. If this command is specified in more than one subcase, then it is recommended that the first subcase contain OFREQ=ALL, and that subsequent subcases contain OFREQ = n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
OFREQ = ALL $ 0.0 through 0.5
SUBCASE 2
SET 10 = 0.0 0.1 0.3
OFREQ = 10
DISP = ALL
SUBCASE3
SET 20 = 0.4 0.5
OFREQ = 20
STRESS = ALL
```

## OIMPERFECT

### Request Output of Imperfection

Request output imperfection vectors in SOL 400. Imperfection vectors are coordinate variations applied on grid points to express geometric imperfection.

#### Format:

$$\text{OIMPERFECT} \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array}, \text{GEOM} \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

#### Examples:

OIMPERFECT(GEOM) = 5

OIMPERFECT(PLOT)=ALL

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

GEOM	To output grid points with updated coordinates in punch file.
ALL	Imperfection vectors for all points will be output.
n	Set identification of a previously appearing SET command. Only imperfection vectors of points with identification numbers that appear on this SET command will be output.
NONE	Imperfection vectors for no points will be output.

#### Remark:

1. This command must be above all subcases.

## OLOAD

### Applied Load Output Request

Requests the form and type of applied load vector output.

#### Format:

$$\text{OLOAD} \left( \left( \begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix} \right), \left[ \begin{matrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{matrix} \right], \left[ \begin{matrix} \text{REAL or IMAG} \\ \text{PHASE} \end{matrix} \right], \left[ \begin{matrix} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{matrix} \right], \right. \\ \left. \left[ \begin{matrix} \text{RPRINT} \\ \text{NORPRINT} \end{matrix} \right], \text{RPUNCH}, [\text{CID}] \right) = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix} \right\}$$

#### Examples:

```
OLOAD=ALL
OLOAD (SORT1, PHASE)=5
OLOAD (SORT2, PRINT, PSDF, CRMS, RPUNCH=20
OLOAD (PRINT, RALL, NORPRINT)=ALL
```

Describer	Meaning												
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.												
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.												
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.												

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 10.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 10.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level and RANDOM must be selected in the Case Control Section. See Remark 10.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in the Case Control. See Remark 10.
RPRINT	Writes random analysis results to the print file (Default).
NORPRINT	Disables the writing of random analysis results to the print file.
RPUNCH	Writes random analysis results to the punch file.
CID	Requests printing of output coordinate system ID to printed output file (.f06).
ALL	Applied loads for all points will be output. See Remarks 1. and 7.
NONE	Applied load for no points will be output.
n	Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all requested points (zero and nonzero) to be output.
3. OLOAD=NONE overrides an overall output request.
4. In the statics superelement solution sequences, and in the dynamics solution sequences (SOLs 107 through 112, 118, 145, 146, and 200). OLOADs are available for superelements and the residual structure. Only externally applied loads are printed. Loads transmitted from upstream superelements are not printed. Transmitted loads can be obtained with GPFORCE requests.

In the nonlinear transient analysis solution sequences (SOLs 129 and 159), OLOADs are available only for residual structure points and include loads transmitted by upstream superelements.

5. In nonlinear analyses SOL 106 and 129, OLOAD output will not reflect changes due to follower forces. But SOL 400 OLOAD output reflects the follower force change correctly. Notes, in SOL 400, the first OLOAD output before starting nonlinear iteration process, only reflects total non-follower force without thermal load and follower force.
6. Loads generated by the SPCD Bulk Data entry do not appear in OLOAD output.
7. In SORT1 format, OLOADs recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and it is zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
8. OLOAD results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
9. In static inertia relief analysis, the OLOAD output includes both the inertia loads and applied loads.
10. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
11. Note that the CID keyword affects only grid point related output, such as DISP, VELO, ACCE, OLOAD, SPCF and MPCF. In addition, CID the keyword needs to appear only once in a grid point related output request anywhere in the Case Control Section to turn on the printing algorithm.
12. OLOAD output is not available in SOL 600.

## OMODES

## Output Modes Set

Selects a set of modes for output requests.

### Format:

$$\text{OMODES} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

### Examples:

```
OMODES = ALL
OMODES = 20
```

Describer	Meaning
ALL	Output for all extracted modes will be computed (Default).
n	Set identification of a previously appearing SET command. Output for those extracted modes appearing on this SET command will be computed.

### Remarks:

1. This command is valid only in SOLs 103, 105, 107, 110, 111, 112, 145, 146, 200 and 400. It is ignored in all other analyses.
2. In contrast to the OFREQUENCY Case Control command, which provides an alternate way of selecting the modes to be output based on their frequencies, the OMODES command allows mode selection based on integer mode ID. For example:

```
SUBCASE 10
...
SET 11 = 1, 3, 5, 7
OMODES = 11
DISP = ALL
...
SUBCASE 20
...
SET 21 = 25., 28., 31.
OFREQ = 21
DISP = ALL
...
```

3. If both the OMODES and the OFREQUENCY requests appear, the OMODES request takes precedence.
4. If neither the OMODES nor the OFREQUENCY request is specified, output will be generated for all modes.
5. Note that the OMODES command has no effect on the number of modes computed. It only selects a subset of the computed modes for which output is to be generated.



6. In superelement analysis, the set definition, using an OMODES command, for an upstream superelement, will not be recognized unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OMODES default is ALL. Note that the program does not check to see if this condition is satisfied.
7. If OMODES is used in conjunction with a FATIGUE case control command, the OMODES request must be for contiguous modes from 1 through n and the FTGLOAD bulk data entries cannot invoke modes greater than n.
8. OMODES is intended to be specified above all subcases or in the first subcase. In SOLs 200 and 400, the first subcase would be the first subcase with ANALYSIS=MODES, MTRAN, MFREQ, MCEIG, SEAERO, and FLUTTER. See the MODES command to allow different output requests for individual modes.

**OTIME**                      Output Time Set

Selects a set of times for output requests.

**Format:**

$$\text{OTIME} = \left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$$

**Examples:**

```
OTIME =ALL
OTIME =15
```

Descriptor	Meaning
ALL	Output for all times will be computed.
n	Set identification number of a previously appearing SET command. Output for times closest to those given on this SET command will be computed (Integer > 0).

**Remarks:**

1. If the OTIME command is not supplied in the Case Control Section, then output for all times will be computed.
2. This command is particularly useful for requesting a subset of the output (e.g., stresses only at peak times, etc.).
3. This command can be used in conjunction with the MODACC module to limit the times for which modal acceleration computations are performed.
4. If this command is specified in more than one subcase in the modal solution sequences, then it is recommended that the first subcase contain OTIME=ALL, and that subsequent subcases contain OTIME=n. Also, data recovery requests should be specified only in the subsequent subcases. For example:

```
SUBCASE 1
  OTIME ALL
SUBCASE 2
  OTIME = 10
  SET 10 = . . .
  DISP = ALL
SUBCASE 3
  OTIME = 20
  SET 20 = . . .
  STRESS = ALL
```

5. The OTIME command is not effective in nonlinear transient analysis (SOL 129) or in SOL 400, ANALYSIS=NLTRAN. However, the OTIME command can be used in the nonlinear transient thermal analysis (SOL 159) and in SOL400, ANALYSIS=HTRAN to limit the output to specified output times.

6. In superelement analysis, the set definition, using an OTIME command for an upstream superelement, will not be recognized unless a similar request appears for its downstream neighbor. The downstream request should either be comprised of the union of all upstream requests, or left blank, as the OTIME default is ALL. Note that the program does not check to see if this condition is satisfied.

## OUTPUT

### Case Control Delimiter

Delimits the various types of commands for the structure plotter, curve plotter, grid point stress.

#### Format:

$$\text{OUTPUT} \left[ \left( \left\{ \begin{array}{c} \text{PLOT} \\ \text{POST} \\ \text{XYOUT} \\ \text{XYPLOT} \end{array} \right\} \right) \right]$$

#### Examples:

OUTPUT (POST)  
 OUTPUT (PLOT)  
 OUTPUT (XYOUT)

Describer	Meaning
PLOT	Beginning of the structure plotter request. This command must precede all structure plotter control commands. Plotter commands are described in <a href="#">OUTPUT(PLOT) Commands</a> .
POST	Beginning of grid point stress SURFACE and VOLUME commands. This command must precede all SURFACE and VOLUME commands. These commands are described in <a href="#">OUTPUT(PLOT) Commands</a> .
XYOUT or XYPLOT	Beginning of curve plotter request. This command must precede all curve plotter control commands. XYPLOT and XYOUT are equivalent. Curve plotter commands are described in <a href="#">X-Y PLOT Commands</a> .

#### Remarks:

1. The structure plotter request OUTPUT(PLOT), the curve plotter request OUTPUT(XYOUT or XYPLOT), and the grid point stress requests (OUTPUT(POST)) must follow the standard Case Control commands.
2. If OUTPUT is specified without a describer, then the subsequent commands are standard Case Control commands.
3. Case Control commands specified after OUTPUT(POST) are SURFACE and VOLUME.
4. This command must appear at the end of normal Case Control just above the Begin Bulk Command. Any Case Control command, controlling, say, selection and flow of the MSC Nastran run such as TEMP(LOAD), that occurs after this entry, will be ignored.

## P2G Direct Input Load Matrix Selection

Selects direct input load matrices.

### Format:

P2G=name

### Example:

```
P2G = LDMIG
P2G = LDMIG1, LDMIG2, LDMIG3
SET 100 = LDMIG, L1, L8
P2G = 100
P2G = 1.25*LDMIG1, 1.0*LDMIG2, 0.82*LDMIG3
```

Describer	Meaning
name	Name of a $[P_g^2]$ matrix to be input on the DMIG Bulk Data entry, or name list with or without factors. See Remark 4. (Character).

### Remarks:

- Terms are added to the load matrix before any constraints are applied.
- The matrix must be columnar in form (e.g., Field 4 on DMIG entry, IFO, must contain the integer 9.)
- A scale factor may be applied to this input using the user parameter PARAM,CP2. See [Parameters](#).
- The formats of the name list:
  - Names without factor.  
Names separated by a comma or blank.
  - Names with factors.  
Each entry in the list consists of a factor, followed by a star, followed by a name. The entries are separated by a comma or blank. The factors are real numbers. Each name must be paired with a factor including 1.0.
- SOL 101: P2G should be selected above all subcase. The number of columns specified for NCOL on the DMIG Bulk Data entry must equal the number of subcases.

SOL 106 and SOL 400: There are two choices. P2G may be selected in every subcase (or step in SOL 400), with NCOL=1. Otherwise, P2G may be selected above the subcases and PARAM,CP2 selected in every subcase.

## PACCELERATION

## Particle Acceleration Output Request for SOL 108/111

Request the form and frequency steps of particle acceleration output.

### Format:

$$PACCELERATION \left( \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setf} \end{array} \right\} \right] \right) =$$

$$\left\{ \begin{array}{c} \text{ALL} \\ \text{setg} \\ \text{NONE} \end{array} \right\}$$

### Example:

```
PACCELERATION = ALL
SET 20 = 104 THRU 204, 1005 THRU 1901
SET 50 = 105.0, 250.0, 310.0
PACCE (PUNCH, SOLUTION=50) = 20
```

Descriptor	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG	Requests rectangular format (real and imaginary). Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase). Phase output is in degrees.
SOLUTION	Selects a set of excitation frequencies for which the particle accelerations will be processed. The default is all excitation frequencies. See Remark 3.
setf	Set identification of excitation frequencies.
setg	Set identification of grid points on wetted surface. See Remark 2.

**Remarks:**

1. This entry will be available only for fluid-structure coupling problem. The particle acceleration is input force vector to each domain via wetted surface in frequency response analysis.
2. Both fluid and structure grid points can be selected. The particle accelerations of the grid points not on wetted surface will be zero.
3. The selected frequency must be part of the excitation frequencies. If not, the nearest excitation frequency will be selected.
4. Only SORT1 form is supported.

## **PAGE**

## Page Eject

Causes a page eject in the echo of the Case Control Section.

### **Format:**

PAGE

### **Example:**

PAGE

### **Remarks:**

1. PAGE appears in the printed echo prior to the page eject.
2. PAGE is used to control paging in large Case Control Sections.



## PARAM Parameter Specification

Specifies values for parameters. Parameters are described in [Parameters](#).

### Format:

PARAM,n,V1,V2

### Examples:

PARAM,GRDPNT,0  
PARAM,K6ROT,1.0

Descriptor	Meaning
n	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single precision	Real	Blank
Character	Character	Blank
Real, double precision	Real, Double Precision	Blank
Complex, single precision	Real or Blank	Real or Blank
Complex, double precision	Real, Double Precision	Real, Double Precision

### Remarks:

1. The PARAM command is normally used in the Bulk Data Section and is described in the [Bulk Data Entries](#).
2. The parameter values that may be defined in the Case Control Section are described in [Parameters](#). Case Control PARAM commands in user-written DMAPs requires the use of the PVT module, described in the [MSC Nastran DMAP Programmer's Guide](#).

## **PARTN** Partitioning Vector Specifications

Specifies a list of grid point identification numbers that will be partitioned with the DMAP module MATMOD (Option 17).

### **Format:**

PARTN=n

### **Example:**

PARTN=10

<b>Describer</b>	<b>Meaning</b>
n	Set identification number of a previously appearing SET command (Integer > 0).

### **Remarks:**

1. The PARTN command and the DMAP module MATMOD provide a convenient method for building a partitioning vector for use in DMAP modules such as PARTN and MERGE.
2. The PARTN command is no longer applicable to coupled fluid-structure analysis. It has been replaced by the FSLPOUT command.

**PEAKOUT**

Control peaks identification process for frequency response analysis

**Format:**

PEAKOUT NPEAK = p, NEAR = q, LFREQ = r, HFREQ = s,

$$RTYPE = \left\{ \begin{array}{l} \text{DISP} \\ \text{VELO} \\ \text{ACCE} \end{array} \right\}, PSCALE = \left\{ \begin{array}{l} \text{NONE} \\ \text{DB} \\ \text{DBA} \end{array} \right\}$$

**Example:**

PEAKOUT NPEAK=4, NEAR=2.0, LFREQ=10.0, HFREQ=200.0, RTYPE=DISP, PSCALE=DBA

Describer	Meaning
NPEAK	Requests the desired number of peaks to extract. See remark 2. (Integer, Default=5).
NEAR	Minimum allowed frequency between two peaks. If two peaks are closer than this value, the lower frequency peak will be ignored. (Real, Default=0.01 Hz)
LFREQ	Lowest frequency used in peak identification. (Real, Default=lowest forcing frequency or 0.0 Hz)
HFREQ	Highest frequency used in peak identification. (Real, Default=highest forcing frequency or 1.0e10 Hz)
RTYPE	Results type for peak identification in structural domain. (Default=DISP)
PSCALE	Scaling method for acoustic pressure results in fluid domain. See remark 3. (Default=NONE)

**Remarks:**

1. This command is meaningful only in frequency response analysis (SOLs 108 and 111) and SOL 200 with DFREQ and MFREQ. It is ignored in all other analyses.
2. The actual number of peaks found may be fewer than the desired number.
3. When requesting dB or dBA, the parameter PREFDB is used as a reference pressure for dB calculation, see ACOUT parameter.
4. All GRIDs referenced on any setdof of PFMODE, PFGRID and PFPANEL requests are required (but not limited to) on a DISP output command. The DISP command must be placed above the SUBCASE level.
5. If non sparse data recovery is requested using PARAM,SPARSEDR,NO, then also PARAM,DDRRMM,-1 is required. Sparse Data recovery is default.
6. External super elements are not supported
7. PARAM,PTHRES can be used to request the PEAKOUT result that exceeds a prescribed frequency-dependent threshold value. Refer the User Parameter Description of [PTHRES](#) for more details.

## PFGRID

## Acoustic Grid Participation Factor Output Request

Requests the form and type of acoustic grid participation factor output.

### Format:

$$PFGRID \left[ \left( \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \text{GRIDS} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setg} \end{array} \right\} \right] \right), \right. \\ \left. \left[ \begin{array}{c} \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setf} \\ \text{PEAK} \\ \text{NONE} \end{array} \right\} \right] \right] = \left\{ \begin{array}{c} \text{setdof} \\ \text{NONE} \end{array} \right\}$$

### Example:

```
SET 10 = 11217
SET 20 = 25., 30., 35.
PFGRID (PHASE, SOLUTION=20) = 10
SET 40 = 11217
PEAKOUT NPEAK=4, NEAR=2.0,
PFGRID (PLOT, PHASE, SOLUTION=PEAK) = 40
```

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

- REAL or IMAG Real and imaginary part of complex results will be output (Default).
- PHASE Magnitude and phase of complex results will be output.
- GRIDS Keyword selecting the structural grid points to be processed; the default is all structural grid points.
- setg Identifier of a set containing the identifiers of the structural grid points to be processed.

Describer	Meaning
SOLUTION	Keyword selecting a set of excitation frequencies for which the participation factors will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters.
setf	Identifier of a set of excitation frequencies.
setdof	Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.

**Remarks:**

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of fluid degrees of freedom.
2. Acoustic grid participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.

## PFMODE

### Modal Participation Factor (MPF) Output Request

Requests the form and type of modal participation factor (MPF) output.

#### Format:

$$\begin{aligned}
 & \text{PFMODE} \left[ \left( \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \end{array} \right), \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] [\text{PRTMSG}], \right. \\
 & \left. [\text{SORT} = \text{sorttype}], [\text{KEY} = \text{sortitem}], \left[ \text{ITEMS} = \left\{ \begin{array}{c} \text{ALL} \\ (\text{itemlist}) \end{array} \right\} \right], \right. \\
 & \left[ \text{FLUIDMP} = \left\{ \begin{array}{c} \text{ALL} \\ m_f \\ \text{NONE} \end{array} \right\} \right], \left[ \text{STRUCTMP} = \left\{ \begin{array}{c} \text{ALL} \\ m_s \\ \text{NONE} \end{array} \right\} \right], \\
 & \left[ \text{PANELMP} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setp} \\ \text{NONE} \end{array} \right\} \right], \left[ \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setf} \\ \text{PEAK} \\ \text{NONE} \end{array} \right\} \right], [\text{FILTER} = \text{fratio}], \\
 & \left. [\text{NULL} = \text{ipower}]] = \left\{ \begin{array}{c} \text{setdof} \\ \text{NONE} \end{array} \right\}
 \end{aligned}$$

#### Examples:

SET 20 = 11/T3, 33/T3, 55/T3

SET 30 = 420., 640., 660.

PFMODE (STRUCTURE, SOLUTION = 30, FILTER = 0.01, SORT = ABSD) = 20

SET 40 = 1222, 1223

SET 50 = 10., 12.

PFMODE (FLUID, STRUCTMP=ALL, PANELMP=ALL, SOLUTION=50, SORT=ABSD) = 40

PEAKOUT NPEAK=4, NEAR=2.0,

PFMODE (FLUID, STRUCTMP=ALL, PANELMP=ALL, SOLUTION=PEAK) = 40

Describer	Meaning
STRUCTURE	Requests output of MPFs for the response of structural degrees of freedom. See Remark 8. (Default).
FLUID	Requests output of MPFs for the response of acoustic grid points (one degree of freedom per point). See Remark 9.

Describer	Meaning
-----------	---------

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

PRMSG	Request generation of diagnostic messages for badly-defined output sets during PF computations. These badly defined sets generally produce no output. See Remark 15.
SORT	Keyword selecting the sort type. Default is sorting by increasing natural mode number. See Remark 7.
sorttype	Sort option:  ABSA    output will be sorted by absolute value in ascending order. ABSD    output will be sorted by absolute value in descending order. ALGA    output will be sorted by algebraic value in ascending order. ALGD    output will be sorted by algebraic value in descending order.
KEY	Keyword selecting the output item to be used for sorting; default is FRACTION. See Remark 7.
sortitem	Item from the item list, see the following table, on which the sort operation is performed.
ITEMS	Keyword specifying data selected for output to the .pch file
itemlist	One or more of the items in the following table:

Table 5-1 Item List Table

Item Identifier	Description
RESPONSE	Modal participation factor.
PROJECTION	Projected modal participation factor.
FRACTION	Normalized projected modal participation factor.
SCALED	Projected modal participation factor divided by largest magnitude of all modal participation factors.
MODEDISP	Real and imaginary part of modal participation factors. See Remark 7.
MODERESP	Magnitude and phase relative to total response of modal participation factors. See Remark 7.

	If more than one item is selected, the list must be enclosed in parentheses.
FLUIDMP	Keyword to select output of fluid MPFs. See Remarks 2. and 9.
$m_f$	Number of lowest fluid modes for which MPFs will be computed. See Remark 9.
STRUCTMP	Keyword to select output of structural MPFs. See Remarks 8. and 9.
$m_s$	Number of lowest structural modes for which MPFs will be computed. See Remarks 8. and 9.
PANELMP	Keyword to select output of panel MPFs. See Remark 2.
setp	Identifier of a set of panels.
SOLUTION	Selects a set of excitation frequencies for which MPFs will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters. See Remark 10.
setf	Identifier of a set of excitation frequencies. See Remark 10.
FILTER	Keyword specifying the value of a filter to be applied to the printed output. See Remark 11.
fratio	Filter value (Default is 0.001). See Remark 11.
NULL	Keyword specifying the power of ten used to detect a null response. See Remark 12.
ipower	Power of ten used to detect a null response (Default is 12). See Remark 12.
setdof	Identifier of a set of structural degrees of freedom or acoustic grid points (one degree of freedom per point) for which MPFs are to be processed. See Remark 3.



**Remarks:**

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of acoustic grid points.
2. Keywords FLUIDMP and PANELMP are only valid if FLUID is specified.
3. If STRUCTURE is specified, *setdof* must reference a set of structural degrees of freedom. If FLUID is specified, *setdof* must reference a set of acoustic grid points.
4. Acoustic MPFs are available in a coupled modal frequency response analysis (SOL 111) only.
5. Printed output includes results for ALL the data items described in the Item List Table.
6. Punched output includes results for only the data items selected by the ITEMS keyword.
7. MPFs are sorted by increasing order of mode number unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.
8. The STRUCTURE option selects MPF calculations for structural degrees of freedom. The STRUCTMP value defines the number of lowest structural modes used in final output preparation prior to any filtering. The default is ALL. If STRUCTMP=NONE, no structural MPF is generated.
9. The FLUID option selects MPF calculations for acoustic grid points (one degree of freedom per point). The STRUCTMP value defines the number of lowest structural modes used in final output preparation of acoustic structure MPFs prior to any filtering. The default is NONE. The FLUIDMP value defines the number of lowest fluid modes used in final output preparation of acoustic fluid MPFs prior to any filtering. The default is NONE.
10. The SOLUTION keyword can be used to select a subset of solutions available. If *setf* is specified, the items in the set are excitation frequencies.
11. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
12. If the magnitude of the total response at a selected response degree of freedom is less than  $10^{-ipowr}$ , then no MPFs are processed. If *ipowr* is not in the range of 1 to 31, the default of 12 is used.
13. Acoustic panel MPFs are normalized using the panel response instead of the total response.
14. If present, only the output of acoustic structural MPFs will include the load participation factor. The load participation factor has a mode number of 0 and a resonance frequency of 0. Structural MPFs will not include the load participation factor.
15. Output sets that define no valid model degrees of freedom for data recovery produce no PF output. This condition is usually discovered prior to actual computations and an informational message is generated. Occasionally, when multiple subcases are involved and one of the subcases references an invalid set, the PF calculations are still performed for any valid combinations. PRTMSG can be used to aid in identification of invalid set data under these conditions.

## PFPANEL

## Acoustic Panel Participation Factor Output Request

Requests the form and type of acoustic panel participation factor output.

### Format:

$$\begin{aligned}
 & \text{PFPANEL} \left( \left( \left[ \begin{array}{c} \text{FLUID} \\ \text{STRUCTURE} \end{array} \right] \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \text{PRTMSG} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \right. \\
 & \left. \left[ \text{PANEL} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setp} \end{array} \right\} \right], \left[ \text{SORT} = \text{sorttype} \right], \left[ \text{KEY} = \text{sortitem} \right], \left[ \text{ITEMS} = \left\{ \begin{array}{c} \text{ALL} \\ (\text{itemlist}) \end{array} \right\} \right], \\
 & \left. \left[ \begin{array}{c} \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{setf} \\ \text{PEAK} \\ \text{NONE} \end{array} \right\}, \left[ \text{FILTER} = \text{fratio} \right], \left[ \text{NULL} = \text{ipower} \right] \right) \right. \\
 & \left. = \left\{ \begin{array}{c} \text{setdof} \\ \text{NONE} \end{array} \right\} \right)
 \end{aligned}$$

### Example:

```

SET 10 = 10., 12.
SET 20 = 1222, 1223
PFPANEL (SOLUTION=10, FILTER=0.01, SORT=ABSD) = 20
SET 30 = 5001/T2, 6502/T3
PFPANEL (STRUCTURE, SOLUTION=10) = 30
SET 40 = 11217
PEAKOUT NPEAK=4, NEAR=2.0,
PFPANEL (SOLUTION=PEAK, FILTER=0.01, SORT=ABSD) = 40
    
```

Describer	Meaning
FLUID	Request output of MPFs for the response of acoustic grid points (one degree of freedom per point). See Remark 11. (Default)
STRUCTURE	Request output of MPFs for the response of structure degrees of freedom. See Remark 12.
PRINT or (blank)	
PUNCH	
PLOT	

Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
X		X*
	X	X*
		X*

Describer	Meaning
	* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.
PRTMSG	Request generation of diagnostic messages for badly-defined output sets during PF computations. These badly defined sets generally produce no output. See Remark 12..
REAL or IMAG	Real and imaginary part of complex results will be output (Default).
PHASE	Magnitude and phase of complex results will be output.
PANEL	Keyword to select the panels to be processed; default is all panels.
setp	Identifier of a set of panels.
SORT	Keyword selecting the sort type. Default is alphabetic sorting by panel name.
sorttype	Sort option:  ABSA     Output will be sorted by absolute value in ascending order.  ABSD     Output will be sorted by absolute value in descending order.  ALGA     Output will be sorted by algebraic value in ascending order.  ALGD     Output will be sorted by algebraic value in descending order.
KEY	Keyword selecting the output item to be used for sorting; default is FRACTION.
sortitem	Item from the item list, see the following table, on which the sort operation is performed.
ITEMS	Keyword specifying data selected for output to the .pch file.
itemlist	One or more of the items in the following table:

Table 5-2 Item List Table

Item Identifier	Description
RESPONSE	Modal participation factor.
PROJECTION	Projected modal participation factor.
FRACTION	Normalized projected modal participation factor.
SCALED	Projected modal participation factor divided by largest magnitude of all modal participation factors.
MODEDISP	Real and imaginary part of modal participation factors.
MODERESP	Magnitude and phase relative to total response of modal participation factors.

	If more than one item is selected, the list must be enclosed in parentheses.
SOLUTION	Keyword selecting a set of excitation frequencies for which the participation factors will be processed. Default is all excitation frequencies. PEAK activates the PEAKOUT automatic peak frequency extraction. See PEAKOUT Case Control Command for detailed description of PEAK parameters.
setf	Identifier of a set of excitation frequencies.
FILTER	Keyword specifying the value of a filter to be applied to the printed output.
fratio	Filter value (Default is 0.001), see Remark 7.
NULL	Keyword specifying the power of ten used to detect a null response, see Remark 8.
ipower	Power of ten used to detect a null response (Default is 12), see Remark 8.
setdof	Identifier of a set of fluid degrees of freedom for which the participation factors are to be processed.

**Remarks:**

1. All PFMODE(FLUID), PFPANEL, and PFGRID Case Control commands must reference the same set of fluid degrees of freedom.
2. Acoustic panel participation factors are available in a coupled frequency response analysis (SOL 108 and SOL 111).
3. Printed output includes results for all the data items described in the itemlist table.
4. Punched output includes results for only the data items selected by the ITEMS keyword.
5. Panel participation factors are alphabetically sorted by panel names unless the SORT keyword specifies a different sorting order. If a sorting order is specified, the KEY keyword selects the item that is used for sorting. When MODEDISP is selected, sorting is based on the magnitude. When MODERESP is selected, sorting is based on the real part.

6. The SOLUTION keyword can be used to select a subset of solutions available. If set setf is specified, the items in the set are excitation frequencies.
7. The filter is applied to the real part of the normalized projected participation factors. Only participation factors that pass the filter are output.
8. If the magnitude of the total response at a selected response degree of freedom is less than  $10^{-ipowr}$ , then no modal participation factors are processed. If ipower is not in the range of 1 to 31, the default of 12 is used.
9. If present, output includes the load participation factor. The panel name of the load participation factors is -LOAD-.
10. The FLUID option selects panel PF calculation for acoustic grid points (one degree of freedom per point) and setdof should be identification numbers.
11. The STRUCTURE option selects panel PF calculation for structure grid points and setdof should be identification numbers and component codes.
12. Output sets that define no valid model degrees of freedom for data recovery produce no PF output. This condition is usually discovered prior to actual computations and an informational is generated. Occasionally, when multiple subcases are involved and one of the subcases references an invalid set, the PF calculations are still performed for any valid combinations. PRTMSG can be used to aid in identification of invalid set data under these conditions.

**PLOTID**

## Plotter Identification

Defines a character string that will appear on the first frame of any plotter output.

**Format:**

PLOTID=title

**Example:**

```
PLOTID=BLDG. 125 BOX 91
```

Describer	Meaning
title	Any character string.

**Remarks:**

1. PLOTID must appear before the OUTPUT(PLOT) or OUTPUT(XYOUT) Case Control commands.
2. The presence of PLOTID causes a special header frame to be plotted, with the supplied identification plotted several times. The header frame allows plotter output to be identified easily.
3. If no PLOTID command appears, no ID frame will be plotted.
4. The PLOTID header frame will not be generated for the table plotters.

## POST Postprocessor Data Specifications

Controls selection of data to be output for postprocessing functions via the OUTPUT2 module interface for selected commercial postprocessor products.

### Format:

$$\text{POST} \left\{ \begin{array}{l} \text{TOFILE} \\ \text{TOCASE} \end{array} \right\} \left\{ \begin{array}{l} \text{furn} \\ \text{filename} \end{array} \right\} [\text{ppname}][\text{ENHOP2}][\text{oplist}]$$

### Examples:

```
POST PATRAN TOFILE 51 NOSTRESS
POST TOFILE SUBCASE8
POST TOCASE SUFNAME1
```

Descriptor	Meaning
TOFILE	Keyword to specify the destiny of .op2 output files (No default if it appears above all subcases).
TOCASE	Keyword to specify the destiny of subcase results to user-defined .f06 output files. (No default if it appears above all subcases.)
furn	Fortran file unit reference number where data will be written (Integer > 0).
filename	Suffix filename (see Remark 8., 9. and 11.) (Char8)
ppname	Name of the target post-processor program for TOFILE option (Default = PATRAN).
ENHOP2	If ppname='PATRAN' then request enhanced op2 format-same as with PARAM,POST,1. The enhanced op2 format includes Nastran version information and qualifier values.
oplist	Names of output items to be processed.

### Remarks:

1. The POST Case Control command controls the placement of output data on external FORTRAN files for use by commercial postprocessors. Use of the POST command generates the proper value for the POST DMAP parameter associated with the particular postprocessor. All of the other parameter controls related to the POST DMAP parameter remain in effect, and are described in [Parameters, 793](#). The products supported are identified in the following table. PATRAN is the default postprocessor name used for ppname. DBC output (POST=0) cannot be controlled by the POST command.

ppname	Product	PARAM,POST,Value
PATRAN	MSC Patran V3	-1
PATRAN with ENHOP2	MSC Patran V3	+1
SDRC	Siemens I-DEAS	-2
NF	MSC/LMS NF	-4
FEMTOOLS	DDS/FemTools	-5
UNIGRAHICS	Siemens/Unigraphics	-6

- The TOFILE descriptor is followed by the specification of either a FORTRAN unit reference number, or a file name associated with the external file that receives the output data. If a FORTRAN unit number is used, the file must be associated with it via the ASSIGN File Management Statement. If POST appears above all subcases, TOFILE must be used to specify either a FORTRAN unit reference number or a file name. The default value of TOFILE, which appears under a subcase, will inherit from the value given in the POST above all subcases. If the unit reference number is associated with a form=formatted file, changes in unit numbers across subcases are not allowed.
- The data that can be controlled for each postprocessor product is limited, and is identified under the description of the POST and related DMAP parameters as described in [Parameters, 793](#). The keywords that can be used for the oplist options are shown in the following table. If an output item supported by a particular postprocessor is described in [Parameters, 793](#) but is not listed here, then the POST command cannot be used to control its output to the external file.

Output Item	oplist Keyword	Case Command
Displacements	[NO]DISPLACE	DISP
Forces of single point constraint	[NO]SPCFORCE	SPCFORCE
Element forces	[NO]FORCES	ELFO/FORCE
Element stresses	[NO]STRESS	ELST/STRESS
Element strain energy	[NO]ESE	ESE
Grid point force balance	[NO]GPFORCE	GPFORCE
Stress at grid points	[NO]GPSIGMA	STRESS
Strain/curvature at grid points	[NO]GPEPSILON	STRAIN
Composite element failure indices	[NO]PLYFAILURE	STRESS
Element kinetic energy	[NO]EKE	EKE
Element energy loss	[NO]EDE	EDE
Multi-point constraint forces	[NO]MPCFORCE	MPCFORCE
Composite lamina stresses	[NO]PLYSIGMA	STRESS
Composite lamina strains	[NO]PLYEPSILON	STRAIN



Output Item	oplist Keyword	Case Command
Element strains	[NO]STRAIN	STRAIN
Grid point stresses	[NO]GPSTRESS	GPSTRESS
Grid point strains	[NO]GPSTRAIN	GPSTRAIN
Applied loads	[NO]LOAD	OLOAD
No items to be output	NONE	-----
Structure mode participation factors	[NO]SMPF	PFMODE

- Output data items must have been generated via the appropriate Case Control command for the data to be available for postprocessing options. For example, the specification of SPCF in the oplist of the POST command will not produce forces of single point constraint on the POST output file unless there is a SPCF Case Control command present. Refer to the tables under the POST parameter description in [Parameters, 793](#) for a list of the output items supported by each postprocessor.
- Any data generated by a Case Control command is automatically included in the oplist of the POST command. If output data is not wanted for a particular case, then the characters “NO” should be the first two characters of the keyword in the oplist. For example, NODISP specifies that displacements are not to be posted to the output file, even though they have been requested via the DISP Case Control command. Alternatively, the related POST parameters may be used. For example, to avoid outputting any displacements whatsoever to the .op2 file, use a PARAM, OUG, NO Bulk Data entry.
- Certain data (e.g., geometry) is always generated and is not dependent upon the presence of a Case Control command in the input data. The POST command affects the placement of this data on the external file only insofar as the selection of the postprocessor defines the value of the POST DMAP parameter value. The actions described in [Parameters, 793](#) under the POST parameter description will prevail for the particular value of POST associated with the selected postprocessor. The primary purpose of the POST command is to give the user more control over subcase-dependent output data being stored on the external OUTPUT2 file.
- If a POST command is present within any subcase, a POST command must also be present above the subcase level. The placement of the POST command above the subcase level causes a cumulative effect on POST commands in subsequent subcases. Any options specified above the subcase level propagate down into the POST command within a subsequent subcase. Thus, if a POST command specifies NODISP (no displacement output wanted) above the subcase level, then a POST command with the DISP option would be required within a subcase to generate any output to the OUTPUT2 file for displacements. This also implies that changing the OUTPUT2 file unit reference number with the TOFILE option in a subcase causes all output quantities currently scheduled for output to be switched to the new unit number, not just those in the oplist for the current POST command.
- When the name of an output file is specified by keyword TOFILE, the ASSIGN statement in the File Management Section (FMS) can be used to specify the full path of its root name. The logical-keyword for the root name is OUTPUT2F. The default root name is the Nastran job name. FORTRAN unit reference number 19 has been reserved by Nastran for OUTPUT2F, although the user can assign other FORTRAN unit number to it. The full file name is in the form of <root name>.<suffix filename>.

9. When the name of an output file is specified by keyword TOCASE, the ASSIGN statement in the File Management Section can be used to specify the full path of its root name. The logical-keyword for the root name is OPCASE. The default root name is the Nastran job name. FORTRAN unit reference number 22 has been reserved by Nastran for OPCASE, although the user can assign other FORTRAN unit numbers to it. The full file name is in the form of <root name>.<suffix filename>. Also, ppname and oplist are not required. If ppname and oplist are specified, they will be ignored. Suffix filename must be specified with keyword TOCASE.
10. POST commands using TOCASE for structure mode participation factor output (SMPF) are not supported and will be ignored.
11. The TOFILE option lets you control the output results (per the table in Remark 3) that are placed on the specified file in OP2 format. The "POST TOFILE SUBASE8" example directs output to the file named <root name>.subcase8.op2. The TOCASE option directs standard printed output requested for a subcase to the specified file in .f06 format. The "POST TOCASE SUFNAM1" example directs output to the file named <root name>.sufnam1.f06. The TOCASE and TOFILE descriptors are independent of one another. POST commands using each descriptor may be present in a subcase to specify different output destinations for both .op2 and .f06 outputs at the same time. For both the TOCASE and TOFILE options, the resulting file name will not have the .op2 or the .f06 file extension automatically appended if the file suffix name contains a period (".").
12. The TOCASE keyword is not supported for printout in the f06 file coming from X-Y Plot commands.

**PRESSURE**

## Pressure Output Request

Requests form and type of pressure output. Analogous to the DISPLACEMENT Case Control command. See the description of the DISPLACEMENT Case Control command, [DISPLACEMENT \(Case\)](#).

## RANDOM

### Random Analysis Set Selection

Selects the RANDPS and RANDT1 Bulk Data entries to be used in random analysis.

#### Format:

$$\text{RANDOM} = \left\{ \begin{array}{c} n \\ i \end{array} \right\}$$

#### Examples:

```
RANDOM=177
```

```
SET 10=100 110 120  
RANDOM=10
```

Describer	Meaning
n	Set identification number of a previously appearing SET command, which in turn references multiple RANDPS/RANDT1 Bulk Data entries with different set identification numbers.
i	Set identification number of RANDPS and RANDT1 Bulk Data entries to be used in random analysis (Integer > 0).

#### Remarks:

1. RANDOM must select RANDPS Bulk Data entries to perform random analysis.
2. RANDOM must appear in the first subcase of the current loop. RANDPS Bulk Data entries may not reference subcases in a different loop. Loops are defined by a change in the FREQUENCY command, or changes in the K2PP, M2PP, or B2PP commands.
3. If RANDPS entries are used in a superelement analysis, the RANDOM command may be specified above the subcase level if a condensed subcase structure (SUPER=ALL) is used. If a condensed subcase structure is not used, then a unique RANDOM selection of a unique RANDPS entry must be specified within each of the desired superelement subcases.
4. If a SET is referenced by n, then the SET identification number must be unique with respect to all RANDPS/RANDT1 set identification numbers.

**RCROSS**

## Cross-Power Spectral Density and Cross-Correlation Function Output Request

Requests computation and output of cross-power spectral density and cross-correlation functions in random analysis.

**Format:**

$$\text{RCROSS} \left( \left( \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) \left[ \begin{array}{c} \text{PRINT} \\ \text{NOPRINT} \end{array} \right], [\text{PUNCH}], [\text{PSDF}, \text{CORF}, \text{RALL}] \right) = n$$
**Example:**

```
RCROSS (PHASE, PSDF, CORF) = 10
RCROSS (RALL, NOPRINT, PUNCH) = 20
RCROSS = 30
```

Descriptor	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output for cross-power spectral density function. Use of either REAL or IMAG yields the same output (Default).
PHASE	Requests polar format (magnitude and phase) of complex output for cross-power spectral density function. Phase output is in degrees.
PRINT	Write output to print file (Default).
NOPRINT	Do not write output to print file.
PUNCH	Write output to punch file.
PSDF	Requests the cross-power spectral density function be calculated and output for random analysis postprocessing (Default).
CORF	Requests the cross-correlation function be calculated and output for random analysis postprocessing.
RALL	Requests both the cross-power spectral density function and cross-correlation function be calculated and output for random analysis postprocessing.
n	Identification number of the RCROSS Bulk Data entry to be used in random analysis (Integer > 0).

**Remarks:**

1. Case Control command RCROSS must be used along with Case Control command RANDOM. See Remarks under the [RANDOM \(Case\)](#) Case Control command description.

2. Response quantities such as DISPLACEMENT, STRESS, and FORCE must be requested by corresponding Case Control commands in order to compute cross-power spectral density and cross-correlation functions between the two response quantities specified by the RCROSS Bulk Data entry. It is recommended that those requests be put above the subcase level to avoid the situation that some response quantities are missing when it comes to the random analysis postprocessing.
3. The response quantities must belong to the same superelement. The cross-power spectral density and cross-correlation functions between the two responses, which belong to the different superelements, are not supported.

**RELDISP**

## Relative Displacement Output Request

Requests the form and type of relative displacement output with respect to a structural grid point in SOL 101, 103, 105, 108, 109, 111, 112, 200 and SOL 400 only.

**Format:**

$$\text{RELDISP} \left[ \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], [\text{REFSET} = m] \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ \text{NONE} \\ n \end{array} \right\}$$

**Example:**

```
RELDISP (REAL, PUNCH, PRINT, REFSET=10) =20
```

```
RELDISP (REFSET=10) =20
```

Describer	Meaning		
SORT1	Output will be presented as a tabular listing of grid points for each frequency.		
SORT2	Output will be presented as a tabular listing of frequency for each grid point.		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with "PARAM,POST, X" (or the POST Case Control command), while the .h5 binary database file will be generated with "MDLPRM,HDF5, X" specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REFSET (=m)	Set number of the previously appearing SET command having Reference Structural Grid ID present in the main bulk data w.r.t relative displacement is requested.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Relative displacement for all points will be output. See Remarks 2.

Describer	Meaning
NONE	No relative displacement will be output
n	Set identification of a previously appearing SET command. Only relative displacements of points with identification numbers that appear on this SET command will be output. (Integer >0).

**Remarks:**

1. RELDISP is available for SOL 101, 103, 105, 108, 109, 111, 112, 200 and SOL400 only.
2. RELDISP can be requested against only one grid for all the different subcases. The reference set should be same for all the subcases.
3. REFSET entry needs to be defined for getting the relative displacement.
4. See Remark 1. under [DISPLACEMENT \(Case\)](#), for a discussion of SORT1 and SORT2.
5. RELDISP (refset =m) =NONE suppresses the generation of relative displacement output.
6. Relative Displacement between two grids is the algebraic difference of the displacement of both the grids.
7. S points or E points are not supported as reference point against which relative displacement can be calculated.
8. For superelements, the reference grid can be only in residual structure and relative displacement output can be requested for residual structure only.
9. The reference set grid should be a structural grid only with respect to which the relative displacement is requested.
10. The reference set grid should be present in the main bulk data section only.



**REPCASE**

## Repeat Output Subcase Delimiter

Delimits and identifies a repeated output subcase.

**Format:**

REPCASE=*n*

**Example:**

REPCASE=1 3 7

Describer	Meaning
<i>n</i>	Subcase identification number. (Integer > 1)

**Remarks:**

1. *n* must be strictly increasing (i.e., must be greater than all previous subcase identification numbers).
2. REPCASE defines a subcase that is used to make additional output requests for the previous real subcase. This command is required because multiple output requests for the same item are not permitted in the same subcase.
3. One or more repeated subcases (REPCASEs) must immediately follow the subcase (SUBCASE) to which they refer.
4. REPCASE may be used only in statics and normal modes analysis. In normal modes analysis, output for only one mode is obtained. This output corresponds to the SUBCASE preceding the REPCASE.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the REPCASE with a TEMP(Load) Case Control command, or the element deformation state with a DEFORM command.
6. Repcase is not applicable to SOLs 400, 600, and 700.

## RESVEC

### Controls Residual Vectors

Specifies options for and calculation of residual vectors.

#### Format:

$$\text{RESVEC} \left( \left( \begin{array}{c} \text{INRLOD} \\ \text{NOINRL} \end{array} \right), \left( \begin{array}{c} \text{APPLOD} \\ \text{NOAPPL} \end{array} \right), \left( \begin{array}{c} \text{ADJLOD} \\ \text{NOADJLOD} \end{array} \right) \left( \begin{array}{c} \text{RVDOF} \\ \text{NORVDO} \end{array} \right), \left( \begin{array}{c} \text{DAMPLOD} \\ \text{NODAMP} \end{array} \right), \left( \begin{array}{c} \text{DYNRSP} \\ \text{NODYNRSP} \end{array} \right) \right) =$$

$$\left\{ \begin{array}{c} \text{SYSTEM/NOSYSTEM} \\ \text{COMPONENT/NOCOMPONENT} \\ \text{BOTH or YES} \\ \text{NO} \end{array} \right\}$$

$$\text{RESVEC}(\text{FLUID}) = \{\text{YES|NO}\}$$

#### Examples:

```
RESVEC=SYSTEM
RESVEC (NOINRL) =COMPONENT
RESVEC=NO
RESVEC (FLUID) =NO
```

Descriptor	Meaning
INRLOD/ NOINRL	Controls calculation of residual vectors based on inertial forces due to rigid-body motion (Default =INRLOD).
APPLOD/ NOAPPL	Controls calculation of residual vectors based on applied loads (Default = APPLD).
ADJLOD/ NOADJLOD	Controls calculation of residual vectors based on adjoint load vectors (SOL 200 only; Default = ADJLOD).
RVDOF/ NORVDOF	Controls calculation of residual vectors based on RVDOFi entries (Default = RVDOF).
DAMPLOD/ NODAMP	Controls calculation of residual vectors based on viscous damping (Default = DAMPLOD).
DYNRSP/ NODYNRSP	Controls whether the residual vectors will be allowed to respond dynamically in the modal transient or frequency response solution. See Remark 5. (Default = DYNRSP).
SYSTEM/ NOSYSTEM	Controls calculation of residual vectors for system (a-set) modes. For NOSYSTEM, descriptors inside the parentheses are ignored. See Remark 2. for default.
COMPONENT/ NOCOMPONENT	Controls calculation of residual vectors for component (superelement or o-set) modes. For NOCOMPONENT, descriptors inside the parentheses are ignored. See Remark 2. for default.

Describer	Meaning
BOTH or YES	Requests calculation of residual vectors for both system modes and component modes. See Remark 2. for default.
NO	Turns off calculation of residual vectors for both system and component modes, and describers inside the parentheses are ignored. See Remark 2. for default.
FLUID	Requests residual vector calculation for the fluid model (Default = YES).

**Remarks:**

1. RESVEC=SYSTEM/NOSYSTEM and RESVEC=COMPONENT/NOCOMPONENT may be specified in the same subcase.
2. RESVEC=BOTH is the default in all solution sequences except SOLs 103, 106, (with PARAM,NMLOOP), and 115, wherein RESVEC=COMPONENT is the default.
3. If the RESVEC command is specified then the user parameters PARAM,RESVEC and PARAM,RESVNER are ignored.
4. The lower frequency cutoff on the EIGR or EIGRL Bulk Data entries should be left blank or set to a value below the minimum frequency. Residual vectors may not be calculated if all modes below the maximum frequency cutoff are not determined. If low frequency modes are to be excluded from the analysis, use the MODESELECT Case Control command or PARAM,LFREQ.
5. Caution needs to be exercised when allowing the residual vectors to respond dynamically in a modal solution. The best approach is to always include enough normal modes to capture the dynamics of the problem, and rely on the residual vectors to help account for the influence of the truncated modes on the quasistatic portion of the response. This is not the default setting for this capability. When choosing to allow the residual vectors to respond dynamically, it is important to be aware of the frequency content of the excitation, as it will have the ability to excite these augmentation modes. If this is undesirable, then the forcing function should be filtered in advance to remove any undesired frequency content, or specify the NODYNRSP keyword.
6. Residual vectors are normalized with respect to MASS if possible. Massless residual vectors are normalized with respect to STIFFNESS. Residual vectors are not normalized by the requested method of the NORM field of the selected EIGR or EIGRL entry.
7. The maximum number of applied loads that may be used for residual vector calculations is controlled by PARAM,MAXAPL. Note that if the number of applied loads is large, the cost can become prohibitive. If the number of applies loads exceeds MAXAPL, residual vectors are disabled.

## RGYRO

### Activates Gyroscopic Effects and Selects RGYRO or UNBALNC Entries

The RGYRO Case Control command activates the rotodynamics capability, and selects the RGYRO Bulk Data entry for use in complex modes, frequency response, and static analysis. For transient response, the RGYRO command selects the UNBALNC Bulk Data entry. If the UNBALNC entry is not required, setting RGYRO to YES will include the gyroscopic effects in the transient response calculation. Setting RGYRO to NO will deactivate gyroscopic effects in all solutions.

#### Format:

For complex modes, frequency response, and static analysis:

RGYRO = n or YES/NO

#### Examples:

RGYRO = 100

#### For Transient Response:

RGYRO = YES

or

RGYRO = 200

**RIGID**

## Rigid Element Method

Selects rigid elements processing method for RBAR, RBAR1, RJOINT, RROD, RTRPLT, RTRPLT1, RBE1, RBE2, and RBE3.

**Format:**

$$\text{RIGID} = \left\{ \begin{array}{l} \text{LINEAR} \\ \text{LAGRAN} \\ \text{LGELIM} \end{array} \right\}$$

**Example:**

RIGID=LAGRAN

Describer	Meaning
LINEAR	Selects the linear elimination method.
LAGRAN	Selects the Lagrange multiplier method.
LGELIM	Selects the Lagrange multiplier method with elimination.

**Remarks:**

1. The RIGID command must be above the SUBCASE level.
2. The RIGID command can be used in SOLs 101, 103, 105, and 400 only. For all other solution sequences, only RIGID=LINEAR is available.
3. If the RIGID command is not specified in the Case Control Section, RIGID=LINEAR is used for all solution sequences except SOL 400. For SOL 400, RIGID=LAGRAN is used. If the RIGID command is specified, the full command must be specified, including the right hand side.
4. RIGID=LGELIM is not available for SOL 400.
5. LINEAR processing will not compute thermal loads. Also, for SOLs 103 and 105, LINEAR processing will not compute differential stiffness. In order to compute thermal load or differential stiffness, the LAGRAN or LGELIM methods must be used.
6. For SOL 400, the LINEAR rigid elements are valid for small rotation only. The LAGRAN method is valid for both small and large rotation (parameter LGDISP=1).
7. For the LINEAR method, the dependent DOFs are eliminated and placed in the mp-set. For the LAGRAN method, both independent and dependent DOFs are placed in the l-set. Lagrange multiplier DOFs are created internally for the dependent DOFs and placed in l-set. For the LGELIM method, the LAGRAN rigid elements are created first. Then, both the Lagrange DOFs and the dependent DOFs are eliminated, and the dependent DOFs are placed in the mr-set. Both the mp-set and mr-set are subsets of the m-set. See [Degree-of-Freedom Set Definitions](#).

8. Between LAGRAN and LGELIM, LAGRAN is the preferred method. LGELIM is a backup method if difficulty is encountered using the LAGRAN method.
9. The parameters LMFACT and PENFN can be used as scale factor and penalty function, respectively, for the LAGRAN method of processing.
10. In a SOL 400 analysis with CWELD, CFAST, and/or CSEAM elements Nastran internally creates RBE3 elements. These internal RBE3 elements respond in the same way to the RIGID command as any other rigid body elements that may have been defined in the model. Therefore the RIGID command also has an effect on the behavior of CWELD and CFAST elements in a SOL 400 analysis. For "ANALYSIS=NLSTAT" or "ANALYSIS=NLTRAN", the generated RBE3 constraints become Lagrange elements and will undergo large rotation. For "ANALYSIS=NLTRAN" with initial conditions (IC=n) in case control that cause large initial stresses in the structure at time t=0, the case control entry RIGID needs to have the value "RIGID=LINEAR." If "PARAM, OLDWELD, YES" is specified, the CSEAM is considered a linear element.
11. For external superelements, both the Case Control Command EXTSEOUT and the PARAM, EXTOUT, REQUIRE a value of RIGID=LINEAR (the default for non SOL400 solution sequences) when running non SOL 400 jobs.
12. When creating superelements in SOL 400, RIGID=LAGRAN produces incorrect answers, RIGID=LINEAR should be used or put the rigid elements into the residual.
13. When RIGID=LAGRAN is used, one may observe SWM 4968, indicating negative terms on multiple DOF which is due to the Lagrange multiplier. This message may be ignored and an accurate solution will be obtained.
14. When RIGID=LAGRAN is used, the CASI iterative solver may fail to converge or terminate, one should switch to a direct solver.
15. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The average temperature of the connected grid points is used as the temperature of the rigid body element. In this case, for the Lagrange method, PARAM, BAILOUT, -1 is activated within the solver.
16. For coupled thermal-mechanical analysis, only RIGID=LINEAR is available.

**ROTBENT**

## Specifying rotor initial deformation

Specifies rotor initial deformation (kink and offset) parameters.

**Format:**

ROTBENT = n

**Example:**

ROTBENT = 10

**Remarks:**

1. ROTBENT option is only supported for SOL400 analysis. ROTBENT analysis will require 2 step analysis, with step 1 being NLSTATIC and step 2 being rotordynamics analysis.

## ROTSEKE

## Rotor Modal Strain and Kinetic Energies Output Request

Requests rotor modal strain and kinetic energies at selected rotors and sort method.

### Format:

$$\text{ROTSEKE} \left[ \left( \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{MODE} \\ \text{ROTOR} \end{array} \right] \right) \right] = \left\{ \begin{array}{l} \text{ALL} \\ \text{NONE} \\ n \end{array} \right\}$$

### Examples:

```
SET 50 = 10, 20, 25 - Rotor ID set
ROTSEKE = 50
ROTSEKE (PLOT, MODE) = 50
ROTSEKE (PUNCH, MODE) = ALL
ROTSEKE (PRINT, PUNCH, ROTOR] = 50
```

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X**		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously.

\*\* Rotor kinetic and strain energies .f06 in the output file will NOT be generated with PARAM,ROTSEKE, X, specified in Bulk Data Section, and an UFM will be printed. PARAM,ROTSEKE has been removed from MSC NASTRAN release 2021.3

MODE or (blank)	Output will be presented as a tabular listing by mode number.
ROTOR	Output will be presented as a tabular listing by rotor ID according to SET n.
All	Rotor energies for all rotors will be output.
n	Set identification number of a previously appearing SET command. Only rotor IDs that appear on this SET command will be included in the rotor energies output (Integer >0).



**Remarks:**

1. ROTSEKE is only available for direct or modal complex eigenvalue analysis in SOLs 107/110/200/400.
2. For a model with one rotor only, the strain and kinetic energy percentages will always be 100%.

## RSDAMP

### Specifying Damping for the Residual Structure

Requests parameter and hybrid damping for the residual structure.

#### Format:

$$\text{RSDAMP} \left[ \begin{array}{c} \text{STRUCTURE} \\ \text{FLUID} \\ \text{BOTH} \end{array} \right] = n$$

#### Example:

RSDAMP (STRUCTURE, FLUID, or BOTH) = n

Describer	Meaning
n	Identification number of a DAMPING Bulk Data entry (Integer > 0).

#### Remarks:

1. For modal solutions, this entry adds to the modal damping that may be specified by the SDAMPING Case Control command.
2. This command can be different in each residual structure subcase.

## SACCELERATION

## Solution Set Acceleration Output Request

Requests the form and type of solution set acceleration output.

### Format:

$$\text{SACCELERATION} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \text{PRINT}, \text{PUNCH}, \left( \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

### Examples:

SACCELERATION=ALL

SACCELERATION (PUNCH, IMAG) = 1 4 2

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Acceleration for all solution set points (modes) will be output.
NONE	Acceleration for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only accelerations of points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. Acceleration output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis, and are discussed in Remark 1 under the [DISPLACEMENT \(Case\)](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SACCELERATION=NONE allows an overall output request to be overridden.

**SDAMPING**

## Structural Damping Selection

Requests modal damping as a function of natural frequency in modal solutions or viscoelastic materials as a function of frequency in direct frequency response analysis.

**Format:**

$$\text{SDAMPING} \begin{bmatrix} \text{STRUCTURE} \\ \text{FLUID} \\ \text{COUPLED} \end{bmatrix} = n$$
**Example:**

```
SDAMPING=77
SDAMPING (COUPLED)=100
```

Describer	Meaning
STRUCTURE or FLUID	Modal damping is requested for the structural or fluid portion of the model.
COUPLED	Modal damping is requested for the structure and fluid coupled portion of the model.
n	Set identification number of a TABDMP1 or TABLEDi Bulk Data entry (Integer>0).

**Remarks:**

1. In the modal solutions (e.g., SOLs 110, 111, 112, 145, 146, and 200), SDAMPING must reference a TABDMP1 entry.
2. In direct frequency response analysis (e.g., SOL 108), SDAMPING must reference a TABLEDi entry which defines viscoelastic (frequency-dependent) material properties. See [Viscoelastic Material Properties](#) in the *MSC Nastran Dynamic Analysis User's Guide*. TABLEDi,n and TABLEDi,n+1 must be specified in the Bulk Data to define the complex moduli of the viscoelastic materials.
3. When SDAMPING is defined for a superelement, PARAM,SESDAMP must be placed either in the SUBCASE for the superelement or in the superelement's BEGIN SUPER section in order to activate the modal damping for the superelement. The default for superelements is to place all boundary points in the B set; in this case, PARAM,SESDAMP,AUG is recommended. Alternatively, place all boundary points in the C set and set PARAM,SESDAMP,YES.
4. When SDAMPING is used during FRF component generation, PARAM,KDAMP,-1 is recommended in order to place the modal damping terms in the imaginary part of the stiffness matrix. The default for KDAMP is 0, which places the modal damping terms in the damping matrix, leading to a marginally under or over damped response. When hybrid damping is used during FRF component generation, it is recommended to set the KDAMP field to YES on the HYBDAMP entry.

5. When SDAMPING is used with METHOD(COUPLED), SDAMPING(COUPLED) should be used. For the real coupled mode computation, structural modes and fluid modes are not separated. So SDAMPING(STRUCTURE) or SDAMPING(FLUID) cannot be used properly. When SDAMPING(STRUCTURE/FLUID) is used with METHOD(COUPLED), it returns fatal out.

## SDISPLACEMENT

## Solution Set Displacement Output Request

Requests the form and type of solution set displacement output.

### Format:

$$\text{SDISPLACEMENT} \left[ \left( \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right), \text{PRINT}, \text{PUNCH}, \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

### Examples:

SDISPLACEMENT=ALL

SDISPLACEMENT (SORT2, PUNCH, PHASE) =NONE

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Displacements for all solution set points (modes) will be output.
NONE	Displacements for no solution set points (modes) will be output.
n	Set identification number of a previously appearing SET command. Only displacements on points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. The defaults for SORT1 and SORT2 depend on the type of analysis, and is discussed in Remark 1 under the [DISPLACEMENT \(Case\)](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL then the remaining commands will also be SORT1.
2. SDISPLACEMENT=NONE allows an overall output request to be overridden.
3. The SDISPLACEMENT command is required to output normalized complex eigenvectors.

**SEALL**

## Superelement Generation and Assembly

Specifies the superelement identification numbers of Phase 1 processing in which all matrices and loads are generated and assembled. Controls execution of the solution sequence.

**Format:**

$$\text{SEALL} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

**Examples:**

```
SEALL=ALL
SEALL=7
```

Descriptor	Meaning
ALL	Generate and assemble all superelements.
n	Set identification number of a previously appearing SET command. Only superelements with identification numbers that appear on this SET command will be generated and assembled (Integer > 0).
i	Identification number of a single superelement that will be generated and assembled (Integer > 0).

**Remarks:**

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. This command combines, in one command, the functions of the SEMG, SELG, SEKR, SELR, and SEMR commands.
6. This command does not control superelement data recovery (Phase 3). See the [SEDR \(Case\) Case Control](#) command description.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

## SEDAMP

### Specifying Damping for Superelements

Requests parameter and hybrid damping for superelements.

#### Format:

SEDAMP=n

#### Example:

Describer	Meaning
n	Identification number of a DAMPING Bulk Data entry (Integer > 0).

#### Remarks:

1. This command adds to the modal damping that may be specified by the Case Control command SDAMPING, used in conjunction with PARAM,SESDAMP,YES.
2. Multiple SEDAMP requests per superelement are not allowed. If more than one SEDAMP request is specified per superelement, the second and subsequent requests will be ignored.



## SEDR Superelement Data Recovery

Specifies the superelement identification numbers for which data recovery will be performed.

### Format:

$$\text{SEDR} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

### Examples:

SEDR=ALL  
 SEDR=7

Descriptor	Meaning
ALL	Performs data recovery for all superelements.
n	Set identification number of a previously appearing SET command. Data recovery will be performed for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which data recovery will be performed (Integer > 0).

### Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If this command is not present, data recovery is performed for all superelements for which there are output requests (i.e., the default for this command is SEDR=ALL).
6. The presence of the EXTDROUT Case Control command or the user PARAMeter EXTDROUT forces SEDR=ALL.

**SEDV**

## Superelement Design Variable Processing

Specifies the superelement identification numbers for which the design variables will be processed.

**Format:**

$$\text{SEDV} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

**Examples:**

```
SEDV=ALL
SEDV=18
```

Describer	Meaning
ALL	Requests design variable processing for all superelements. This is the default value if SEDV is missing in the file.
n	Set identification number of a previously appearing SET command. Design variable processing will be performed for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which design variable processing will be performed (Integer > 0).

**Remarks:**

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of superelement sensitivity analysis, see the *MSC.Nastran Design Sensitivity and Optimization User's Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

**SEEXCLUDE**

## Superelement Matrix and Load Assembly Exclusion

Specifies the superelement identification numbers for which all matrices and loads will not be assembled into the downstream superelement.

**Format:**

$$\text{SEEXCLUDE} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

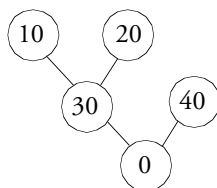
**Examples:**

```
SEEXCLUDE=ALL
SEEXCLUDE=18
```

Descriptor	Meaning
ALL	All upstream superelements will be excluded.
n	Set identification number of a previously appearing SET command. Only those superelements with identification numbers that appear on this SET command will be excluded (Integer > 0).
i	Identification number of a single superelement for which matrices will be excluded (Integer > 0).

**Remarks:**

1. This command, if present, must be located before the first SUBCASE command.
2. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
3. This command is not meaningful when applied to the residual structure.
4. For a further discussion of this command, see the *MSC Nastran Handbook for Superelement Analysis*.
5. If the SEEXCLUDE command is specified in a restart of SOLs 101 through 200, then PARAM,SERST,MANUAL must be specified. Also, the SEKR command must be specified for the superelement immediately downstream from the excluded superelement. For example, if superelement 10 is excluded in the following superelement tree:



then the user must specify the following commands in the Case Control Section:

```
SEKR = 30
```

PARAM, SERST, MANUAL

**SEFINAL**

## Final Superelement for Assembly

Specifies the superelement identification number for the final superelement to be assembled.

**Format:**

$$\text{SEFINAL} = \left\{ \begin{array}{c} \text{n} \\ \text{i} \end{array} \right\}$$

**Example:**

SEFINAL=14

Describer	Meaning
n	Set identification of a previously appearing SET command. Each superelement identification number appearing on the SET command must belong to a disjoint model (Integer > 0).
i	Identification number of the final superelement to be assembled (Integer > 0).

**Remarks:**

1. If this command is not present, the program selects the order of the superelements for assembly operations.
2. This command, if present, must be located before the first SUBCASE command.
3. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
4. This command can be used on restarts to ensure that minor modeling changes do not also change the processing order. For this usage, inspect the SEMAP table to determine which superelements were final superelements on the prior run.
5. See the *MSC Nastran Handbook for Superelement Analysis* for a further discussion of this command.

## SEKREDUCE

## Superelement Stiffness Matrix Assembly and Reduction

Specifies the superelement identification numbers for which stiffness matrices are assembled and reduced.

### Format:

$$\text{SEKREDUCE} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

### Examples:

SEKREDUCE=ALL  
 SEKREDUCE=9

Descriptor	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the stiffness matrix will be assembled and reduced (Integer > 0).

### Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. SEKREDUCE is an alternate form, and is entirely equivalent to the obsolete command SEMASSEMBLE.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

## SELGENERATE

## Superelement Load Generation

Specifies the superelement identification numbers for which static loads will be generated.

### Format:

$$\text{SELGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

### Examples:

```
SELGENERATE=ALL
SELGENERATE=1 8
```

Descriptor	Meaning
ALL	Generates static loads for all superelements.
n	Set identification number of a previously appearing SET command. Static load matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which load matrices will be generated (Integer > 0).

### Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

## SELREDUCE

## Superelement Load Assembly and Reduction

Specifies the superelement identification numbers for which the static load matrices will be assembled and reduced.

### Format:

$$\text{SELREDUCE} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

### Examples:

```
SELREDUCE=ALL
SELREDUCE=9
```

Descriptor	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will be assembled only for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the load matrices will be assembled and reduced (Integer > 0).

### Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only be appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. This command is used on restarts to selectively assemble and reduce load matrices.
5. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
6. In superelement static analysis, SELREDUCE is equivalent to SELASSEMBLE.
7. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
8. SEALL=ALL is the default, but can be overridden by specifying and of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.



**SEMGENERATE**

## Superelement Matrix Generation

Specifies the superelement identification numbers for which stiffness, mass, and damping matrices will be generated.

**Format:**

$$\text{SEMGENERATE} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

**Examples:**

```
SEMGENERATE=ALL
SEMGENERATE=7
```

Descriptor	Meaning
ALL	Generates structural matrices for all superelements.
n	Set identification number of a previously appearing SET command. Structural matrices will only be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which structural matrices will be generated (Integer > 0).

**Remarks:**

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. SEALL=ALL is the default, in the structured SOLs 101 through 200. This default can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

## SEMREDUCE

## Superelement Mass and Damping Assembly and Reduction

Specifies the superelement identification numbers for which the mass and damping matrices will be assembled and reduced. In buckling analysis, the differential stiffness matrices will be assembled and reduced.

### Format:

$$\text{SEMREDUCE} = \left\{ \begin{array}{c} \text{ALL} \\ n \\ i \end{array} \right\}$$

### Examples:

```
SEMREDUCE=ALL
SEMREDUCE=9
```

Descriptor	Meaning
ALL	Assembles and reduces matrices for all superelements.
n	Set identification number of a previously appearing SET command. Matrices will only be assembled for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the load matrices or the mass and damping matrices will be assembled and reduced (Integer > 0).

### Remarks:

1. This command, if present, must be located before the first SUBCASE command.
2. Zero (0) is the identification number for the residual structure, and can only appear as a member of a set.
3. This command is used on restart to selectively assemble and reduce mass and damping matrices. For a further discussion of this command, see [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. In dynamic analysis, SELASSEMBLE combines the functions of SELREDUCE and SEMREDUCE.
6. This command has no function in static analysis.
7. SEALL=ALL is the default but can be overridden by specifying any of the following Phase 1 commands: SEALL, SEMG, SELG, SEKR, SELR, or SEMR.

**SERESP**

## Superelement Response Sensitivity

Specifies the superelement identification numbers for which the design sensitivity matrices will be generated.

**Format:**

$$\text{SERESP} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{i} \end{array} \right\}$$

**Examples:**

```
SERESP=ALL
```

```
SERESP=18
```

Describer	Meaning
ALL	Requests design sensitivity matrix generation for all superelements. This is the default value if SERESP is missing.
n	Set identification number of a previously appearing SET command. Design sensitivity matrices will be generated for superelements with identification numbers that appear on this SET command (Integer > 0).
i	Identification number of a single superelement for which the design sensitivity matrix will be generated.

**Remarks:**

1. This command, if present, then it must be located before the first SUBCASE command.
2. Zero (0) is the identification number of the residual structure, and can only appear as a member of a SET.
3. For a further discussion of this command, see the [MSC Nastran Reference Guide](#).
4. If i is used, the superelement identification number must be unique with respect to any SET identification numbers used.
5. If both the SEDV and SERESP commands are not present, then the design variable processing and design sensitivity matrix generation will be performed for all superelements.

**SET** Set Definition, General Form

Sets are used to define the following lists:

1. Identification numbers (point, element, or superelement) for processing and output requests.
2. Frequencies for which output will be printed in frequency response problems, or times for transient response, using the OFREQ and OTIME commands, respectively.
3. Surface or volume identification numbers to be used in GPSTRESS or STRFIELD commands.
4. DRESP1 entries that are used in the spanning of subcases.
5. Grid point number and component type code to be used by the MCFRACTION command.

**Formats:**

$$\text{SET } n = \{i_1[,i_2, i_3, \text{THRU } i_4, \text{EXCEPT } i_5, i_6, i_7, i_8, \text{THRU } i_9]\}$$

$$\text{SET } n = \{r_1[,r_2, r_3, r_4]\}$$

$$\text{SET} = \text{ALL}$$

$$\text{SET } n = \{i_1/c_1[,i_2/c_2, i_3/c_3, i_4/c_4]\}$$

$$\text{SET } n = \{l_1, [l_2, l_3]\}$$
**Examples:**

```
SET 77=5
```

```
SET 88=5, 6, 7, 8, 9, 10 THRU 55 EXCEPT 15, 16, 77, 78, 79, 100 THRU 300
```

```
SET 99=1 THRU 100000
```

```
SET 101=1.0, 2.0, 3.0
```

```
SET 105=1.009, 10.2, 13.4, 14.0, 15.0
```

```
SET 1001=101/T1, 501/T3, 991/R3
```

```
SET 2001=M1, M2
```

Describer	Meaning
n	Set identification number. Any set may be redefined by reassigning its identification number. SETs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0).
$i_1, c_1$	Grid point identification numbers and component codes. The $c$ values must be of T1, T2, T3, R1, or R3.
$l_1, l_2$ etc.	Identification names of literals used for matrix or group selection.
$i_1, i_2$ etc.	Identification numbers. If no such identification number exists, the request is ignored (Integer $\geq 0$ ).

Describer	Meaning
$i_3$ THRU $i_4$	Identification numbers ( $i_4 > i_3$ ) (Integer > 0).
EXCEPT	Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.
$r_1, r_2, \text{ etc.}$	Frequencies or times for output. The nearest solution frequency or time will be output. EXCEPT and THRU cannot be used. If an OFREQ or OTIME command references the set then the values must be listed in ascending sequences, $r_1 < r_2 < r_3 < r_4 \dots$ etc., otherwise some output may be missing. If an OFREQ or OTIME command is not present, all frequencies or times will be output (Real > 0.0).
ALL	All members of the set will be processed.

**Remarks:**

1. A SET command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation. Place a number after the THRU.
2. Set identification numbers following EXCEPT within the range of the THRU must be in ascending order.
3. In SET 88 above, the numbers 77, 78, etc., are included in the set because they are outside the prior THRU range.
4. SET commands using the grid point/component code format cannot contain THRU. SETs using this format should be selected only by the MCFRACTION Case Control command.
5. SET commands using literals apply only to direct matrix input such as K2PP etc. or FLSPOUT panel grouping.

**SETP**

## Process Set Definition

Process sets are used to define lists of SET identifications to be processed individually for data recovery:

**Formats:**

$$\text{SETP } n = \{i_1[i_2, i_3 \text{ THRU } i_4 \text{ EXCEPT } i_5, i_6, i_7, i_8 \text{ THRU } i_9]\}$$
**Examples:**

```
SETP 77=5, 6
```

```
SETP 88=5, 6, 7, 8, 9, 10 THRU 55
```

Describer	Meaning
n	SETP identification number. Any SETP may be redefined by reassigning its identification number. SETPs specified under a SUBCASE command are recognized for that SUBCASE only (Integer > 0).
$i_1, i_2, \dots, i_n$	SET identification numbers $i_1, i_2$ , etc. If no such identification number exists, the request is ignored (Integer > 0).
EXCEPT	Set identification numbers following EXCEPT will be deleted from output list as long as they are in the range of the set defined by the immediately preceding THRU. An EXCEPT list may not include a THRU list or ALL.

**Remarks:**

1. A SETP command may be more than one physical command. A comma at the end of a physical command signifies a continuation command. Commas may not end a set. THRU may not be used for continuation—place a number after the THRU.
2. Set identification numbers following EXCEPT,k within the range of the THRU, must be in ascending order.
3. SETP usage is limited to the EDE, EKE and ESE Case Control commands.

## SETS DEFINITION

## Case Control Processing Delimiter

Delimites the various type of commands under grid point stress. This command is synonymous with OUTPUT(POST).

**Format:**

SETS DEFINITION

**Example:**

SETS DEFINITION

**Remark:**

1. Either SETS DEFINITION or OUTPUT(POST) may be specified, but not both.

## SKIP

### Case Control Processing Delimiter

Activates or deactivates the execution of subsequent commands in Case Control (including plot commands).

#### Format:

$$\text{SKIP} \left\{ \begin{array}{l} \text{ON} \\ \text{OFF} \end{array} \right\}$$

#### Example:

```
SKIPOFF
```

#### Remarks:

1. SKIPON and SKIPOFF commands may appear as many times as needed in the Case Control Section.
2. Commands that are skipped will be printed.
3. SKIPON ignores subsequent commands until either a SKIPOFF or BEGIN BULK command is encountered. This allows the user to omit requests without deleting them from the data. In the following example, plot commands will be skipped.

```
TITLE=EXAMPLE  
SPC=5  
LOAD=6  
SKIPON$SKIP PLOT REQUEST  
OUTPUT (PLOT)  
SET 1 INCLUDE ALL  
FIND  
PLOT  
BEGIN BULK
```



**SMETHOD**

## Iterative Solver Method Selection

Selects iterative solver method and parameters.

**Format:**

$$\text{SMETHOD} \left\{ \begin{array}{c} \text{ELEMENT} \\ n \\ \text{MATRIX} \end{array} \right\}$$
**Example:**

```
SMETHOD = ELEMENT $ selects element-based iterative solver defaults.
SMETHOD = MATRIX $ selects matrix based iterative solver defaults.
SMETHOD = 1000 $ specifies ID of ITER Bulk Data entry to select
iterative.
```

Describer	Meaning
ELEMENT	Selects the element-based iterative solver with default control values.
MATRIX	Selects the matrix-based iterative solver with default control values.
n	Sets identification of an ITER Bulk Data entry (Integer > 0).

**Remarks:**

1. The matrix-based iterative solver is available in SOLs 101, 106, 108, 111, 153, and 400 and allows use of all features.
2. The element-based iterative solver is only available in SOLs 101, 200 and 400. SMETHOD must be placed above all SUBCASEs in this case. It is intended primarily for very large solid element models. See the ITER Bulk Data entry for a list of restrictions in addition to details on setting the convergence parameter epsilon.
3. The element-based iterative solver can be used with SMP by setting smp=number on the command line. Please refer to [smp, 11](#) of this manual for correct usage.
4. For SOL 600, the iterative solver is activated using the MARCSOLV PARAM.
5. GPGPU devices are not supported for iterative methods.

## SOLUTION

### Solution ID Selection

Selects the solution ID for a 3<sup>rd</sup> step external superelement data recovery restart in SOL 400.

#### Format:

SOLUTION = n

#### Example:

SOLUTION=10

Describer	Meaning
n	Solution identification number.

#### Remarks:

1. SOLUTION command is used only in SOL 400 for 3<sup>rd</sup> step external superelement data recovery restart.
2. See Remark 8. under EXTDRIN Case Control command's description for a further explanation and example.

**SPC**

## Single Point Constraint Set Selection

Selects a single point constraint set to be applied.

**Format:**

SPC = n

**Example:**

SPC=10

Describer	Meaning
n	Set identification number of a single-point constraint that appears on an SPC, SPC1, SPC2 (SOL 700), FRFSPC1 (in FRF Based Assembly or FBA process) or SPCADD Bulk Data entry (Integer > 0).

**Remarks:**

1. In cyclic symmetry analysis, this command must appear above the first SUBCASE command.
2. Multiple boundary conditions are only supported in SOLs 101, 103, 105, 145, and 200. Multiple boundary conditions are not allowed for upstream superelements. The BC command must be specified to define multiple boundary conditions for the residual structure in SOLs 103, 105, 145, and 200.

## SPCFORCES

## Single Point Forces of Constraint Output Request

Requests the form and type of single point force of constraint vector output.

### Format:

$$\text{SPCFORCES} \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{c} \text{REAL or NOZPRINT} \\ \text{PHASE} \end{array} \right] \left[ \begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right], \right. \\ \left. \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \text{RPUNCH}, [\text{CID}] \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Examples:

```
SPCFORCES = 5
SPCFORCES (SORT2, PUNCH, PRINT, IMAG) = ALL
SPCFORCES (PHASE) = NONE
SPCFORCES (SORT2, PRINT, PSDF, CRMS, RPUNCH) = 20
SPCFORCES (PRINT, RALL, NORPRINT) = ALL
```

Descriptor	Meaning	
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.	
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.	
PRINT or (blank)	Printer File (.f06)	
PUNCH	Punch File (.pch)	
PLOT	Plot File (.op2/.h5)	
	X	X*
	X	X*
		X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
NOZPRINT	Print only nonzero SPC forces appearing in SORT2 output. This keyword does not affect SORT1 output.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
PRUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output file (.f06).
ALL	Single point forces of constraint for all points will be output. See Remarks 1. and 4.
NONE	Single point forces of constraint for no points will be output.
n	Set identification of a previously appearing SET command. Only single point constraint forces for points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2. In the SORT1 format, only nonzero values will be output.
2. In a statics problem, a request for SORT2 causes loads at all points (zero and nonzero) to be output.
3. SPCFORCES=NONE overrides an overall output request.
4. In SORT1 format, SPCFORCES recovered at consecutively numbered scalar points are printed in groups of six (sextets) per line of output. However, if a scalar point is not consecutively numbered, it will begin a new sextet on a new line of output. If a sextet can be formed and all values are zero, then the line will not be printed. If a sextet cannot be formed, then zero values may be output.
5. SPCFORCES results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
6. In SOLs 129 and 159, SPCFORCES results do not include the effects of mass and damping elements.

7. In all solution sequences except SOLs 129 and 159, SPCFORCES results do include the effects of mass and damping, except damping selected by the SDAMPING Case Control command. PARAM,DYNSPCF,OLD may be specified to obtain SPCFORCES results, which do not include mass and damping effects.
8. In inertia relief analysis, the SPCFORCES output is interpreted differently for SOLs 1, 101, and 200:
  - a. In SOL 1, the SPCFORCE output reflects the effects due to the applied loads only, and not the inertial loads.
  - b. In SOLs 101 and 200, the SPCFORCE output includes both the effects due to inertial loads and applied loads.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Note that the CID keyword affects only grid point related output such as DISP, VELO, ACCE, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid-related output command, anywhere in the Case Control Section, to turn on the printing algorithm.
11. For axisymmetric elements, SPCFORCES results are obtained by integrating over 1 radian of the circumference. Before 2012 release, it is total reaction over the whole circumference.

## SPLINOUT

### Export of Spline Matrix

Requests output of the spline matrix for external use.

#### Format:

$$\text{SPLINOUT} \left[ \left[ \begin{array}{c} \text{BOTH} \\ \text{DISP} \\ \text{FORCE} \end{array} \right], \left\{ \begin{array}{c} (\text{OP2} = \textit{unit}) \\ \text{DMIPCH} \end{array} \right\} \right]$$

#### Examples:

Descriptor	Meaning
BOTH	Requests output of the force and displacement splines (Default).
DISP	Requests output of only the displacement splines.
FORCE	Requests output of only the force splines.
OP2	Requests output to an .op2 file
unit	Unit the .op2 file is assigned to
DMIPCH	Requests output to a .pch file.

#### Remarks:

1. Matrices are output in external sort.
2. If displacement and force splines are identical, only displacement splines are output.
3. Option FACTORS is only allowed together with option OP2.
4. If OP2 = *unit* is specified, a table relating the matrix columns to structural degrees of freedom, and the matrix rows to aerodynamic degrees of freedom, will be written to the .op2 file.
5. If OP2 = *unit* is specified, an appropriate ASSIGN OP2 statement must be present in the File Management Section for this unit.
6. If DMIPCH is specified, DMI entries are written to the.pch file.

**STATSUB**

## Static Solution Selection for Differential Stiffness

Selects the static solution to use in forming the differential stiffness for static analysis, buckling analysis, normal modes, complex eigenvalue, frequency response and transient response analysis.

**Format:**

$$\text{STATSUB} \left( \begin{bmatrix} \text{BUCKLING} \\ \text{PRELOAD} \end{bmatrix} \right) = n$$

**Examples:**

```
STATSUB=23
```

```
STAT=4
```

```
STATSUB (PREL) =7
```

Describer	Meaning
BUCKLING	Subcase ID number corresponding to static subcase of buckling or varying load (Default in buckling analysis).
PRELOAD	Subcase ID number corresponding to static subcase of preload or constant load (Default in dynamic analysis).
n	Subcase identification number of a prior subcase specified for static analysis (Integer > 0).

**Remarks:**

1. STATSUB may be used in SOLs 101,103, 105, 107 through 112, 115, 116, 200 and SOL 400 (ANALYSIS = BUCKLING only in SOL 200 and SOL 400).
2. STATSUB must be specified in the same subcase that contains the METHOD selection for buckling or normal modes, CMETHOD for complex eigenvalue analysis, TSTEP for transient response, and FREQ for frequency response.
3. In SOL 105, if it is intended that results from the first static subcase are used to compute the differential stiffness, then the STATSUB command is not required. That is, the default for STATSUB is the first static subcase identification. In SOLs 101, 103 and 107 through 112, 115, and 116, STATSUB must reference a separate static subcase.
4. In dynamic analysis, only one STATSUB command may be specified in each dynamic subcase. In buckling analysis with a preload, both STATSUB (BUCKLING) and STATSUB(PRELOAD) must be specified in each buckling subcase. STATSUB(PRELOAD) is not supported in SOL 200 or SOL 400. Buckling Analysis with a preload is not supported in SOL 200 and SOL 400.
5. In dynamic analysis, any subcase that does not contain a CMETHOD command in SOLs 107 and 110, a FREQUENCY command in SOLs 108 and 111, and a TSTEP command in SOLs 109 and 112, will be treated as a static subcase.
6. SOL 200 and SOL 400 support linear buckling analysis only, but do not support post-buckling (nonlinear buckling analysis)



7. In versions prior to 2018, if it was desired to have a dynamic subcase **with** the effects of preload and another subcase **without** the effects of preload, then a static subcase with null load was required and its ID was referenced by STATSUB in the dynamic subcase **without** the effects of preload. In version 2018, STATSUB in the dynamic subcase **without** the effects of preload is no longer required. If the STATSUB is not removed and still references the static subcase with null load, then User Fatal Message 9244 will be issued.

**STEP****Step Delimiter**

Delimits and identifies a nonlinear analysis step for SOL 400.

**Format:**

STEP=n

**Examples:**

STEP=10

Describer	Meaning
n	Step identification number (Integer > 0).

**Remarks:**

1. The STEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
2. The STEP command is to be used below the SUBCASE Case Control command. If no SUBCASE is specified, MSC Nastran creates a default SUBCASE 1.
3. The STEP identification number n in a SUBCASE must be in increasing order, and less than 9999999.
4. The following example illustrates a typical application of SUBCASE and STEP:

```

SUBCASE 1
  STEP 1
    LOAD = 1
  STEP 2
    LOAD = 2
SUBCASE 2
  STEP 10
    LOAD = 10
  STEP 20
    LOAD = 20

```

5. The solutions of all SUBCASEs are independent of each other. However, the solution of any STEP is a continuation of the solution of the previous STEP.

## STOCHASTICS

### Randomization of Model Parameters

Request randomization of all or selected subsets of model parameters.

#### Format:

$$\text{STOCHASTICS} = \left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$$

#### Examples:

STOCHASTICS=10

Descriptor	Meaning
ALL	All real values of C-entries, M-entries, P-entries, loading entries, and SPCD entries are to be randomized.
n	Set identification number of a STOCHAS Bulk Data entry (Integer > 0).

#### Remarks:

1. Only one STOCHASTICS command may appear in the Case Control Section and should appear above all SUBCASE commands.
2. The STOCHASTICS = n command may be used to request randomizing a set of analysis model parameters with user specified statistics. (See Remark 1 of the STOCHAS Bulk Data entry.)
3. The default (STOCHASTICS = all) randomizes all scalar analysis model parameters that are real values on the C-entries, M-Entries, P-entries, all loading entries, and SPCD entries with default coefficients of variance (0.05) and multipliers of standard deviations (m=3.).
4. This command will only invoke a single Nastran randomization run. Separate runs can be submitted to achieve different randomizations.

## STRAIN Element Strain Output Request

Requests the form and type of strain output.

### Format:

$$\text{STRAIN} \left( \left( \begin{matrix} \text{SORT1} \\ \text{SORT2} \end{matrix} \right), \left[ \begin{matrix} \text{PRINT, PUNCH} \\ \text{PLOT} \end{matrix} \right] \left[ \begin{matrix} \text{REAL or IMAG} \\ \text{PHASE} \end{matrix} \right], \left[ \begin{matrix} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{matrix} \right], \left[ \begin{matrix} \text{STRCUR} \\ \text{FIBER} \end{matrix} \right], \right. \\ \left. \left[ \begin{matrix} \text{CENTER} \\ \text{CORNER or BILIN} \\ \text{SGAGE} \\ \text{CUBIC} \end{matrix} \right], \left[ \begin{matrix} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{matrix} \right] \left[ \begin{matrix} \text{RPRINT} \\ \text{NORPRINT} \end{matrix}, \text{RPUNCH} \right] \right) = \left\{ \begin{matrix} \text{ALL} \\ \text{n} \\ \text{NONE} \end{matrix} \right\}$$

### Examples:

```
STRAIN=5
STRAIN (CORNER) =ALL
STRAIN (PRINT, PHASE) =15
STRAIN (PLOT) =ALL
STRAIN (PRINT, PSDF, CRMS, RPUNCH) =20
STRAIN (PRINT, RALL, NORPRINT) =ALL
```

Describer	Meaning
-----------	---------

SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

Describer	Meaning
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. The request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
CRMS	Requests the cumulative root mean square function to be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 9.
VONMISES	von Mises strains are output.
MAXS or SHEAR	Maximum shear strains are output.
STRCUR	Strain at the reference plane and curvatures are output for plate elements.
FIBER	Strain at locations Z1, Z2 are computed for plate elements.
CENTER	Output CQUAD4 element strains at the center only.
CORNER or BILIN	Output CQUAD4 element strains at the center and grid points. Using strain gage approach with bilinear extrapolation.
SGAGE	Output CQUAD4 element strains at center and grid points using strain gage approach.
CUBIC	Output CQUAD4 element strains at center and grid points using cubic bending correction.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Strain for all elements will be output.
n	Set identification of a previously appearing SET command. Only strain for elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No element strain will be output.

**Remarks:**

1. Fully accurate nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output. In SOLs 106 and 129, if LGDISP > 0 the center, and if requested for CQUAD4 the corner, strains are computed from the displacements alone and are only approximate. If LGDISP = -1 and material nonlinear then no STRAIN output will be created. In SOL 400, if enhanced material options are used, the computed strains are accurate and in this case a CQUAD4 corner request is ignored.
2. In SOLs 106 and 129, the STRAIN request pertains only to linear elements and only if the parameter LGDISP is -1, which is the default. Nonlinear strains for nonlinear elements are requested by the STRESS command and appear in the nonlinear stress output.
3. STRAIN=NONE overrides an overall output request.
4. Definitions of stress, strain, curvature, and output locations are given in the [Structural Elements](#) in the *MSC Nastran Reference Guide*.
5. If the STRCUR option is selected, the values of Z1 will be set to 0.0. and Z2 will be set to -1.0 on the output.
6. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
7. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\)](#) Case Control command for further discussion.
8. See Remark 1 under the [DISPLACEMENT \(Case\)](#) Case Control command for a discussion of SORT1 and SORT2.
9. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
10. Random response for complex strain in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.
11. Element type CBEAM3 is not supported for random response.

**STRESS**

## Element Stress Output Request

Requests the form and type of element stress output. Note: ELSTRESS is an equivalent command.

**Format:**

$$\text{STRESS} \left( \left[ \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right], \left[ \begin{array}{l} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{l} \text{VONMISES} \\ \text{MAXS or SHEAR} \end{array} \right], \right. \\ \left. \left[ \begin{array}{l} \text{CENTER} \\ \text{CUBIC} \\ \text{SGAGE} \\ \text{CORNER or BILIN} \end{array} \right], \left[ \begin{array}{l} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right] \left[ \begin{array}{l} \text{RPRINT} \\ \text{NORPRINT, RPUNCH} \end{array} \right] \right) = \left\{ \begin{array}{l} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

```
STRESS=5
STRESS (CORNER) =ALL
STRESS (SORT1, PRINT, PUNCH, PHASE) =15
STRESS (PLOT) =ALL
STRESS (PRINT, PSDF, CRMS, RPUNCH) =20
STRESS (PRINT, RALL, NORPRINT) =ALL
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of elements for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each element type.
PRINT or (blank)	
PUNCH	
PLOT	

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

Describer	Meaning
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 11.
VONMISES	Requests von Mises stresses.
MAXS or SHEAR	Requests maximum shear in the plane for shell elements and octahedral stress for solid elements.
CENTER	Requests CQUAD4, CQUADR and CTRIAR element stresses at the center only. The default for CQUAD4 is CENTER. The default for CQUADR and CTRIAR is CORNER.
CUBIC	Requests CQUAD4 element stresses at the center and grid points using strain gage approach with cubic bending correction.
SGAGE	Requests CQUAD4 element stresses at center and grid points using strain gage approach.
CORNER or BILIN	Requests CQUAD4, CQUADR, and CTRIAR element stresses at center and grid points using bilinear extrapolation.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
ALL	Stresses for all elements will be output.
n	Set identification of a previously appearing SET command. Only stresses for elements with identification numbers that appear on this SET command will be output (Integer > 0).
NONE	No element stress will be output.



**Remarks:**

1. ALL should not be used in a transient problem due to excessive output.
2. See Remark 1 under the [DISPLACEMENT \(Case\)](#) Case Control command description for a discussion of SORT1 and SORT2.
3. ELSTRESS is an alternate form and is equivalent to STRESS.
4. STRESS=NONE overrides an overall output request.
5. Prior to MSC Nastran 2014, in nonlinear analysis, the nonlinear stresses will still be printed unless NLSTRESS(PLOT) is specified.
6. The VONMISES option is ignored for ply stresses.
7. The VONMISES, MAXS, and SHEAR options are ignored in the complex eigenvalue and frequency response solution sequences.
8. The options CENTER, CORNER, CUBIC, SGAGE, and BILIN are recognized only in the first subcase, and determine the option to be used in all subsequent subcases with the STRESS, STRAIN, and FORCE commands. (In superelement analysis, the first subcase refers to the first subcase of each superelement. Therefore, it is recommended that these options be specified above all subcases.) Consequently, options specified in subcases other than the first subcase will be ignored. See also Remark 8 under the [FORCE \(Case\)](#) Case Control command for further discussion.
9. For composite ply output, the grid point option for CQUAD4 elements will be reset to the default option (CENTER).
10. MAXS for shell elements is not an equivalent stress.
11. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
12. Random response for complex stress in composites plates or shells, and layered solid composites and layered solid shell composites will be available for each ply.
13. Element type CBEAM3 is not supported for random response.

**STRFIELD**

## Grid Point Stress Output Request

Requests the computation of grid point stresses for graphical postprocessing and mesh stress discontinuities.

**Format:**

$$\text{STRFIELD} = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \end{array} \right\}$$

**Examples:**

```
STRFIELD=ALL
STRFIELD=21
```

Describer	Meaning
ALL	Grid point stress requests for all surfaces and volumes defined in the OUTPUT(POST) Section will be saved for postprocessing.
n	Set identification number of a previously appearing SET command. Only surfaces and volumes with identification numbers that appear on this SET command, and in the OUTPUT(POST) Section, will be included in the grid point stress output request for postprocessing (Integer > 0).

**Remarks:**

1. The STRFIELD command is required for the graphical display of grid point stresses in postprocessors that use the .xdb file (PARAM,POST,0), or when the GPSDCON or ELSDCON commands are specified, and does not provide printed output. The GPSTRESS command can be used to obtain printed output.
2. Only grid points connected to elements used to define the surface or volume are output. See the SURFACE and VOLUME Case Control commands.
3. Element stress output (STRESS) must be requested for elements referenced on requested SURFACE and VOLUME Case Control commands.
4. In nonlinear static and transient analysis, grid point stresses are computed only if parameter LGDISP is -1, which is the default. Also, in nonlinear transient analysis, grid point stresses are computed only for elements with linear material properties.

**SUBCASE**

## Subcase Delimiter

Delimits and identifies a subcase.

**Format:**

SUBCASE=*n*

**Example:**

SUBCASE=101

Descriptor	Meaning
<i>n</i>	Subcase identification number (9999999 > Integer > 0).

**Remarks:**

1. The subcase identification number, *n*, must be greater than all previous subcase identification numbers.
2. Plot requests and RANDPS requests refer to *n*.
3. See the MODES Case Control command for use of this command in normal modes analysis.
4. If a comment follows *n*, then the first few characters of the comment will appear in the subcase label in the upper right-hand corner of the output.
5. Note that in nonlinear statics (SOL106/129), SUBCASE's are not stand-alone solutions like in other solution sequences. They act as a load-progression, and the ending conditions of one SUBCASE become the initial conditions of the next SUBCASE.

**SUBCOM**

## Combination Subcase Delimiter

Delimits and identifies a combination subcase.

**Format:**

SUBCOM = n

**Example:**

SUBCOM = 125

Describer	Meaning
n	Subcase identification number (Integer > 2).

**Remarks:**

1. The subcase identification number, n, must be greater than all previous subcase identification numbers.
2. A SUBSEQ command must follow this command.
3. SUBCOM may only be used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
4. Output requests above the subcase level will be used.
5. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SUBCOM with a TEMP(Load) command, or the element deformations with a DEFORM command.
6. SUBCOMs may be specified in superelement analysis with the following recommendations:
  - a. For each superelement, specify its SUBCASE(s) consecutively, directly followed by its SUBCOM(s).
  - b. Specify a SUPER command with a new load sequence number under each SUBCOM command.

The following example demonstrates a model with one superelement and one load combination:

```

SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SUBCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SUBSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SUBCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SUBSEQ=1.,1.

```

7. For static aeroelasticity, only the displacement/element responses are combined. Trim, stability derivative and monitor point results are not combined.
8. SUBCOMs are not allowed in external superelement creation runs and are ignored in external superelement data recovery in assembly runs.
9. For SOL 200, a compliance response is not combined. SUBCASE/DRSPAN must be used for combining compliance response.

**SUBSEQ**

## Subcase Sequence Coefficients

Gives the coefficients for forming a linear combination of the previous subcases.

**Format:**

SUBSEQ=R1 [, R2, R3, ..., Rn]

**Example:**

SUBSEQ=1.0, -1 .0, 0.0, 2.0

Descriptor	Meaning
Ri	Coefficients of the previously occurring subcases. See Remark 4. (Real).

**Remarks:**

1. The SUBSEQ command can only appear after a SUBCOM command.
2. SUBSEQ may be only used in SOL 101 (statics) or SOL 144 (static aeroelasticity) and in SOL 200 with ANALYSIS=STATICS or ANALYSIS=SAERO.
3. This command list is limited to a maximum of 200 numbers.
4. R1 to Rn refer to the immediately preceding subcases. In other words, Rn is applied to the most recently appearing subcase, R(n - 1) is applied to the second most recently appearing subcase, and so on. The embedded comments (\$) describe the following example:

```
DISPL = ALL
SUBCASE 1
SUBCASE 2
SUBCOM 3
SUBSEQ = 1.0, -1.0 $ SUBCASE 1 - SUBCASE 2
SUBCASE 11
SUBCASE 12
SUBCOM 13
SUBSEQ = 0.0, 0.0, 1.0, -1 .0 $ SUBCASE 11 - SUBCASE 12
or
SUBSEQ = 1.0, - 1.0 $ EQUIVALENT TO PRECEDING COMMAND. USE ONLY ONE.
```

**SUBSEQ1**

## Subcase Factors for Combination

Gives the factors for linear combination of a specific group of SUBCASEs.

**Format:**

SUBSEQ1= s0, s1, sub1, s2, sub2, [sn, subn]

**Example:**

SUBSEQ1= 1.0, 1.0, 101, -1.0, 102

Describer	Meaning
S0	Factor for all SUBCASEs involved. (No default; Real<>0.).
Sn	Factor applicable to SUBn only (Real; No default).
SUBn	SUBCASE ID (No default; Integer>0).

**Remarks:**

1. The SUBSEQ1 command can only appear after a SUBCOM command.
2. SUBSEQ1 may only be used in SOL 101 (Statics) or SOL 144 (Static Aeroelasticity) and in SOL 200 with ANALYSIS=STATIC or ANALYSIS=SAERO.
3. SUBSEQ1 and SUBSEQ are mutually exclusive and can't both appear under a SUBCOM.
4. S0, S1 and SUB1 are required input for SUBSEQ1. S2, SUB2 to Sn, SUBn pair are optional.

## SUBSTEP

### SUBSTEP Delimiter

Delimits and identifies a nonlinear analysis SUBSTEP for COUPLED analysis in SOL 400.

#### Format:

SUBSTEP=n

#### Examples:

```
STEP=50
NLSTEP=10
SUBSTEP=1
ANALYSIS=HSTAT
SUBSTEP=2
ANALYSIS=NLSTAT
```

Describer	Meaning
n	Substep identification number. (Integer > 0)

#### Remarks:

1. The SUBSTEP command can only be used in nonlinear solution sequence SOL 400 (NONLIN).
2. The SUBSTEP command can only be used in a STEP command.
3. When used in a STEP command two or more SUBSTEP commands must occur.
4. Each SUBSTEP must contain a *unique* ANALYSIS=type statement. Currently there is a limitation of only two SUBSTEPS per STEP with the following options:
  - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLSTAT for the second SUBSTEP.
  - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLTRAN for the second SUBSTEP
  - ANALYSIS=HTRAN for the first SUBSTEP and ANALYSIS=NLSTAT for the second SUBSTEP
  - ANALYSIS=HSTAT for the first SUBSTEP and ANALYSIS=NLTRAN for the second SUBSTEP
5. Within a STEP the SUBSTEP identification number n must be in increasing order and not greater than 9999999.
6. The following example illustrates a typical application of SUBCASE, STEP, and SUBSTEPS

Subcase 100

```
STEP 10
STRESS= ALL
NLSTRESS=ALL
SET 1456 = list
NLSTEP=84
SubSTEP 1
```



```

ANALYSIS=HSTAT
THERMAL=ALL
FLUX=ALL
SPC=35
LOAD=11
SubSTEP 2
ANALYSIS=NLSTAT
SPC=2
LOAD=110
DISP (PLOT) =1456

```

7. The solutions of all SUBCASEs are independent of each other. The solution of any STEP is a continuation of the solution of the previous STEP. *The solutions of the SUBSTEPs occur simultaneously within a STEP.*
8. In coupled analysis, TEMP(LOAD)=m will be ignored as the temperature loading from the thermal substep is automatically transferred to the mechanical substep. Note that in this multi-physics coupled framework, both the heat transfer and structural physics are executed on the same mesh with the same time steps.
9. In coupled analysis, TEMP(INIT)=n will be honored. It should be noted that when a thermo-mechanical static analysis is defined, TEMP(INIT) defines the initial temperature for both the thermal and mechanical substeps. Note that when a transient thermal-static mechanical analysis is define, IC=n should be used to define the initial temperature for the thermal substep and TEMP(INIT)=n (the same ID n) should be used to define the initial temperature for the structural substep. Then an initial thermal strain is defined as:

$$\varepsilon T = A(T) \cdot (T - TREF) - A(T_o) \cdot (T_o - TREF)$$

where  $T_o$  comes from TEMP(INIT) and TREF comes from the material entry.

10. The LOADSET Case Control Command *is not allowed* with this command. All dynamic loading must be applied through the use of the DLOAD Case Control command.
11. For coupled analysis, i.e., when two or more sub-steps are defined, a *single* Bulk Data entry NLSTEP must be included in the STEP command containing the SUBSTEP commands. It must occur above or in the STEP command and above the first SUBSTEP command.

The loads and constraints can be independently defined for each physics under the relevant SUBSTEP. The commands that are normally used for single physics are applicable. For, e.g., for a HTRAN-NLSTAT coupled analysis, DLOAD should be used n the HTRAN substep while LOAD should be used in the NLSTAT substep.

12. Any case control command will apply to all SUBSTEPs when and only when it is above all the SUBSTEPs of the STEP.

**SUBTITLE**

## Output Subtitle

Defines a subtitle that will appear on the second heading line of each page of printer output.

**Format:**

SUBTITLE=subtitle

**Example:**

SUBTITLE=PROBLEM NO. 5-1A

Describer	Meaning
subtitle	Any character string.

**Remarks:**

1. SUBTITLE appearing under a SUBCASE command will appear in the output for that subcase only.
2. SUBTITLE appearing before all SUBCASE commands will appear in the output for all subcases except those in Remark 1.
3. If no SUBTITLE command is present, the subtitle line will be blank.
4. The subtitle also appears on plotter output.

**SUPER**

## Superelement Subcase Assignment

Assigns a subcase(s) to a superelement or set of superelements.

**Format:**

$$\text{SUPER} = \left\{ \begin{array}{c} \text{ALL} \\ \left\{ \begin{array}{c} n \\ i \end{array} \right\} [ , l ] \end{array} \right\}$$

**Examples:**

SUPER=17, 3

SUPER=15

SUPER=ALL

Descriptor	Meaning
i	Superelement identification number (Integer > 0).
ALL	The subcase is assigned to all superelements and all loading conditions (Default).
n	Set identification number of a previously appearing SET command. The subcase is assigned to all superelements with identification numbers that appear on this SET command (Integer > 0).
l	Load sequence number (Integer > 0; Default=1).

**Remarks:**

1. All subcases with requests for specific superelement(s) must contain the SUPER command. If no SUPER command is specified in the Case Control Section, then all subcases will be assigned to all superelements; i.e., SUPER=ALL is the default.
2. All subcases associated with superelements must precede those for the residual structure except when SUPER=ALL or SUPER=n and the selected set includes the residual structure.
3. The load sequence number is only used in static analysis and frequency response analysis when there are multiple loading conditions. Also, the residual structure must have a subcase specified for each unique load condition. This is required because the number of residual structure subcases is used to determine the number of load conditions for all superelements.
4. The load sequence number is associated with the order of the subcases for the residual structure; i.e., the third loading condition is associated with the third subcase for the residual structure.
5. Subcases are required for superelements when there is a load, constraint, or output request.
6. If a set is referenced by n, then the SET identification number must be unique with respect to any superelement identification numbers. In addition, the same sets must be used for all loading conditions.

7. If the ALL option is used, it must be used for all loading conditions.
8. If there are no superelements in the model then the SUPER command will be ignored except in 3-step external superelement data recovery restarts in SOL 400. See Remarks 9 and 10 under the EXTDRIN Case Control command description.

**SUPPORT1**

## Fictitious Support Set Selection

Selects the fictitious support set (SUPPORT1 or SUPPORT6 entries only) to be applied to the model.

**Format:**

SUPPORT1=n

**Examples:**

SUPPORT1=15

SUPO=4

Describer	Meaning
n	Set identification of fictitious support set defined on the SUPPORT1 or SUPPORT6 Bulk Data entries. See Remark 1. (Integer > 0).

**Remarks:**

1. SUPPORT1 or SUPPORT6 Bulk Data entries will not be used unless selected in the Case Control Section by the SUPPORT1 command.
2. SUPPORT entries will be applied in all subcases.
3. For SOL 600, Case Control command SUPPORT1 must reference a SUPPORT6 Bulk Data entry with ID = N.

## SVECTOR

## Solution Set Eigenvector Output Request

Requests the form and type of solution set eigenvector output.

### Format:

$$\text{SVECTOR} \left( \begin{array}{c} \text{PRINT,PUNCH} \\ \text{PLOT} \end{array} \right) = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Examples:

SVECTOR=ALL  
SVECTOR (PUNCH) =NONE

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Displacements for all points (modes) will be output.
NONE	Displacements for no points (modes) will be output.
n	Set identification of a previously appearing SET command. Only displacements of points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. SVECTOR=NONE overrides an overall output request.
2. Output will be presented as a tabular listing of grid points for each eigenvector.

**SVELOCITY**

## Solution Set Velocity Output Request

Requests the form and type of solution set velocity output.

**Format:**

$$\text{SVELOCITY} \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \text{PRINT}, \text{PUNCH}, \left( \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

SVELOCITY=5

SVELOCITY (SORT2, PUNCH, PRINT, PHASE) =ALL

Descriptor	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point (or mode number).
PRINT	The printer will be the output medium.
PUNCH	The punch file will be the output medium.
REAL or IMAG	Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.
PHASE	Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.
ALL	Velocity for all solution points (modes) will be output.
NONE	Velocity for no solution points (modes) will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output (Integer > 0).

**Remarks:**

1. Velocity output is only available for transient and frequency response problems.
2. The defaults for SORT1 and SORT2 depend on the type of analysis, and is discussed in Remark 1 under the [DISPLACEMENT \(Case\)](#) Case Control command. If SORT1 is selected for any of the commands SACC, SDIS, and SVEL, then the remaining commands will also be SORT1.
3. SVELOCITY=NONE overrides an overall output request.

## **SYM** Symmetry Subcase Delimiter

Delimits and identifies a symmetry subcase.

### **Format:**

SYM=n

### **Example:**

SYM=123

<b>Describer</b>	<b>Meaning</b>
n	Subcase identification number (Integer > 0).

### **Remarks:**

1. The subcase identification number n must be greater than all previous subcase identification numbers.
2. Plot commands should refer to n.
3. Overall output commands will not propagate into a SYM subcase (i.e., any output desired must be requested within the subcase).
4. SYM may only be used in statics or inertia relief problems.



**SYMCOM**

## Symmetry Combination Subcase Delimiter

Delimits and identifies a symmetry combination subcase.

**Format:**

SYMCOM=n

**Example:**

SYMCOM=123

Describer	Meaning
n	Subcase identification number (Integer > 2).

**Remarks:**

1. The subcase identification number n must be greater than all previous subcase identification numbers.
2. SYMCOM may only be used in statics problems.
3. If the referenced subcases contain thermal loads or element deformations, the user must define the temperature field in the SYMCOM by use of a TEMP(LOAD) command, or the element deformations by a DEFORM command.
4. An alternate command is the SUBCOM command.
5. SYMCOMs may be specified in superelement analysis with the following recommendations:
  - a. For each superelement, specify its SUBCASEs consecutively, directly followed by its SYMCOM(s).
  - b. Specify a SUPER command with a new load sequence number under each SYMCOM command.

The following example represents a model with one superelement and one load combination:

```

SUBCASE 101
SUPER=1,1
LOAD=100
SUBCASE 102
SUPER=1,2
LOAD=200
SYMCOM 110
LABEL=COMBINE SUBCASES 101 AND 102
SUPER=1,3
SYMSEQ=1.,1.
SUBCASE 1001
SUBCASE 1002
SYMCOM 1010
LABEL=COMBINE SUBCASES 1001 AND 1002
SYMSEQ=1.,1.

```

**SYMSEQ**

## Symmetry Sequence Coefficients

Specifies the coefficients for combining symmetry subcases into the total structure.

**Format:**

SYMSEQ=R1 [,R2,R3,..., Rn]

**Example:**

SYMSEQ=1.0, -2.0, 3.0, 4.0

Describer	Meaning
Ri	Coefficients of the previously occurring n SYM subcases. (Real)

**Remarks:**

1. SYMSEQ may only appear after a SYMCOM command.
2. The default value for the coefficients is 1.0 if no SYMSEQ command appears.
3. SYMSEQ may only be used in static analysis or inertia relief.
4. Ri is limited to a maximum of 200 numbers.

**TACCELERATION**

## Acceleration Output Request for trim components, TRMC

Requests the form and type of acceleration output for TRMC.

**Format:**

$$TACCELERATION \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

```
SET 20 = 3/103, 5/0, 12/ALL $
TACCELERATION=ALL
TACC (REAL, PUNCH, PRINT)=20
TACCE=20
TACCELE (SORT2, PRINT)=20
```

Describer	Meaning
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.
SORT2	Output will be presented as a tabular listing of frequency for each grid point.
PRINT or (blank)	
PUNCH	
PLOT	

Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
X		X*
	X	X*
		X*

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

ALL TRMC acceleration for all points of all TRMCs will be output. See Remarks 2.

Describer	Meaning
NONE	No TRMC acceleration will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TACC=NONE suppresses the generation of TRMC acceleration output.
4. TACCELERATION is supported only with PARAM,TRMBIM,PHYSICAL
5. TACCELERATION is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TACCELERATION is available for output only and cannot be utilized as design response.
6. For restart PEM job, TACCE request must remain the same as cold start PEM job.

**TDISPLACEMENT**

## Displacement Output Request for trim components, TRMC

Requests the form and type of displacement output for TRMC.

**Format:**

$$TDISPLACEMENT \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

```
SET 20 = 3/103, 5/0, 12/ALL $
TDISPLACEMENT=ALL
TDIS (REAL, PUNCH, PRINT)=20
TDISP=20
TDISPLACE (SORT2, PRINT)=20
```

Describer	Meaning																
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.																
SORT2	Output will be presented as a tabular listing of frequency for each grid point.																
	<table border="1"> <thead> <tr> <th></th> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td></td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td></td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td></td> <td>X*</td> </tr> </tbody> </table>		Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X		X*	PUNCH		X	X*	PLOT			X*
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)														
PRINT or (blank)	X		X*														
PUNCH		X	X*														
PLOT			X*														

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

ALL TRMC displacement for all points of all TRMCs will be output. See Remarks 2.

Describer	Meaning
NONE	No TRMC displacement will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TDISP=NONE suppresses the generation of TRMC displacement output.
4. TDISPLACEMENT is supported only with PARAM,TRMBIM,PHYSICAL
5. TDISPLACEMENT is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TDISPLACEMENT is available for output only and cannot be utilized as design response.
6. For each grid in a TRMC, it has 3 or 6 DOFs for solid phase and 1 DOF for fluid phase. TDISPLACEMENT output results for both solid phase DOFs and fluid phase DOF.
7. The 3 DOFs of solid phase takes T1, T2 and T3 locations of a regular 6 DOFs structural grid and set R1, R2 and R3 locations to 0.0. The 1 DOF of fluid phase is reported as acoustic pressure.
8. For restart PEM job, TDISP request must remain the same as cold start PEM job.

## TEMPERATURE

## Temperature Set Selection

Selects the temperature set to be used in either material property calculations or thermal loading in heat transfer and structural analysis.

### Format:

$$\text{TEMPERATURE} \left[ \left[ \begin{array}{c} \text{INITIAL} \\ \text{MATERIAL} \\ \text{LOAD} \\ \text{BOTH} \end{array} \right] \right], \text{HSCASE} = i \left[ \right], \text{HSTEP} = j \left[ \right], \text{HTIME} = \left[ \begin{array}{c} t \\ \text{ALL} \end{array} \right] \left[ \right] \left[ \right] \left[ \right]$$

$$\left[ \begin{array}{c} \text{GRID} \\ \text{ELEMENT} \\ \text{BOTH} \\ \text{NONE} \end{array} \right] = n$$

### Examples:

TEMPERATURE (LOAD) =15

TEMPERATURE (MATERIAL) =7

TEMPERATURE=7

TEMPERATURE (LOAD, HSCASE=20, HTIME=12.0) =39

Descriptor	Meaning
MATERIAL	The selected temperature set will be used to determine temperature-dependent material properties indicated on MATTi Bulk Data entries. See Remarks 6., 7., and 8.
LOAD	The selected temperature set will be used to determine an equivalent static load and to update material properties in a nonlinear analysis. See Remarks 2., 5., 6., 7. and 14.
BOTH	Both MATERIAL and LOAD will use the same temperature set.
n	Set identification number of TEMP, TEMPD, TEMPP1, TEMPB3, TEMPRB, or TEMPAX Bulk Data entries (Integer > 0).
INITIAL	The selected temperature table will be used to determine initial temperature distribution in nonlinear static analysis. See Remarks 4., 6., 7., 8., 9., and 12.
HSCASE	Specifies a SUBCASE executed in the selected thermal job. See Remark 14.
i	Identification number of a SUBCASE executed in the selected thermal job. (Integer ≥ 0, Default = 0 is the first SUBCASE) See Remark 14.
HSTEP	Specifies a STEP executed in the selected thermal job. See Remark 14.
j	Identification number of a STEP executed in the selected thermal job. (Integer ≥ 0, Default = 0 is the first STEP) See Remark 14.

Describer	Meaning
HTIME	Specifies the time of a time step executed in the selected nonlinear transient thermal job. See Remark 14.
t	Time of a time step executed in the selected nonlinear transient thermal job. (Real $\geq 0.0$ , Default is the last time of the specified SUBCASE and/or STEP) See Remark 14.
ALL	Selects all time steps executed in the selected nonlinear transient thermal job. See Remark 14.
VERIFY	Output temperature verification data output requested
GRID	Output grid temperature verification data
ELEMENT	Output element temperature verification data
BOTH	Output both grid and element temperature verification data
NONE	Do not output any verification data

### Remarks:

1. In linear analysis, only one TEMP(MATE) may be made in any problem and should be specified above the subcase level. If there are multiple subcase and TEMP(MATE) is not above the first subcase, then it must appear in the last subcase or it will be ignored. See also Remarks 6. and 7.
2. The total load applied will be the sum of external (LOAD command), thermal (TEMP(LOAD) command), element deformation (DEFORM command), and constrained displacement (SPC command) loads.
3. Static, thermal, and element deformation loads should have unique set identification numbers.
4. INITIAL is used in steady state heat transfer analysis for conduction material properties, and provides starting values for iteration. In structural analysis, TEMP(INIT) is used to specify an initial temperature. It may appear above or in the first subcase. In SOL 400, it may appear above or in the first STEP.
5. In superelement data recovery restarts, TEMPERATURE(LOAD) requests must be respecified in the Case Control Section.
6. In linear static analysis, temperature strains are calculated by

$$\varepsilon_T = A(T_o) \cdot (T - T_o)$$

where  $A(T_o)$  is the thermal expansion coefficient defined on the MATi Bulk Data entries,  $T$  is the load temperature defined with TEMPERATURE(LOAD), and  $T_o$  is the initial temperature defined as follows. The following rules apply for TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), and TREF on the MATi entries:

- a. If TEMPERATURE(INITIAL) and TREF are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties.



- b. If TEMPERATURE(MATERIAL) and TREF are specified, then TREF will be used as the initial temperature in calculating the load and the TEMPERATURE(MATERIAL) set will be used for the calculation of material properties.
  - c. If neither TEMPERATURE(INITIAL), TEMPERATURE(MATERIAL), nor TEMPERATURE(BOTH) is present, TREF will be used to calculate both the load and the material properties and will be obtained from the MATi entry. The MATi is not used in this case.
7. In nonlinear static analysis, temperature strains are calculated with

$$\varepsilon_T = A(T) \cdot (T - \text{TREF}) - A(T_o) \cdot (T_o - \text{TREF})$$

where  $A(T)$  is the thermal expansion coefficient defined on the MATi Bulk Data entries,  $T$  is the load temperature defined with TEMPERATURE(LOAD), and  $T_o$  is the initial temperature defined with TEMPERATURE(INITIAL). The following rules apply:

- a. The specification of TEMPERATURE(MATERIAL) or TEMPERATURE(BOTH) will cause a fatal error.
- b. If a subcase does not contain a TEMPERATURE(LOAD) request, then the thermal load set will default to the TEMPERATURE(INITIAL) set.
- c. TEMPERATURE(LOAD) will also cause the update of temperature-dependent material properties due to the temperatures selected in the thermal load set. Temperature-dependent material properties are specified with MATi, MATTi, MATS1, and/or TABLEST Bulk Data entries.
- d. If TREF and TEMPERATURE(INITIAL) are specified, then the TEMPERATURE(INITIAL) set will be used as the initial temperature to calculate both the loads and the material properties. Both are used in the definition of thermal strain.

For SOL 600, TREF and TEMP(INIT) must be consistent (the same values) or unexpected results may occur.

- 8. TEMPERATURE(MATERIAL) and TEMPERATURE(INITIAL) cannot be specified simultaneously in the same run.
- 9. TEMP(INIT) is not used with TEMPAX.
- 10. Temperature loads cause incorrect element forces and stresses in all the dynamic analysis except SOL 400 nonlinear transient analysis with the elements having nonlinear capability.

It should be noted that:

- a. For nonlinear dynamics analysis with SOL 400, TEMPERATURE/THERMAL load can't be applied by TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data), this load should be applied by TEMP(Case Control) and TTEMP(Bulk Data);
  - b. For pure linear analysis with SOL 400, the TEMP/DLOAD (Case Control) and TEMPD/TLOADi (Bulk Data) has to be used to apply the temperature load.
11. In linear analysis, TEMPERATURE(MATERIAL) is not supported for hyperelastic elements (MATHP). TEMP(INIT) must be placed above the subcase level, and TEMP(LOAD) placed within the subcase.

12. For layered composites, neither the TREF specified on the material entries, nor TEMP(INIT) nor TEMP(MATE) are used to determine ply reference temperature. The TREF on the PCOMP or PCOMPG entries is used for all plies of the element. This is true for both linear and nonlinear analysis. If TEMP(INIT) is defined, the TREF on the PCOMP or PCOMPG entries is only used for determining the material properties and TEMP(INIT) is used for thermal strains. If TEMP(INIT) is not defined, the TREF on the PCOMP or PCOMPG entries is used for determining the material properties and thermal strains.
13. For SOL 600, in a thermal stress analysis where the temperatures were produced in a previous SOL 600 simulation, the use of MCHSTAT is preferred.
14. For TEMPERATURE(LOAD) requests in SOL 400, HSUBCASE, HSTEP, and HTIME are used to retrieve the temperature results from an existing thermal database. This feature allows user to select either steady state or transient thermal results for nonlinear structural analysis, with the flexibility of different time steps and dissimilar mesh sizes between thermal and structural runs. The following rules apply for using this capability:
  - HSUBCASE, HSTEP, and HTIME keywords must follow a LOAD keyword.
  - Although all three keywords have default values, at least one keyword must exist to apply this uncoupled multi-physics feature in analysis.
  - HTIME=ALL is used in nonlinear transient structural analysis to perform real time temperature interpolations. In this case, the nodal temperatures of nonlinear elements are updated at each time step. These temperatures are equal to the temperature results of the selected thermal database at current time.
  - The set IDs of TEMP(LOAD) must be different from the set IDs of TEMP(INIT).
  - To save temperature results of thermal analysis in MASTER nastran database, use the following command.

```
nastran thermal_job_name scratch=no
```

In addition, the user must specify the following Bulk Data entry:

```
PARAM,NLPACK,-1
```

for transient thermal models with number of time steps greater than the default NLPACK output time steps.

- The following File Management Statements are required in the current structural model to select the thermal database.

```
ASSIGN hrun='thermal_job_name.MASTER'
```

```
DBLOC DATABLEK=(HEATDB) LOGI=hrun
```

- Contact body integrity is not maintained while doing the thermal mapping. While this would not be a restriction for bodies that are not physically in contact or in glued thermal contact, it could lead to undesirable temperature mapping for bodies that are in regular thermal contact with temperature gradients across the contact interface.

**TERMIN**

## Conditions to Terminate in SOL 600

Selects a TERMIN Bulk Data entry which specifies criteria such that a SOL 600 analysis can be terminated, for example, if the displacement at a certain grid exceeds a specified value.

**Format:**

TERMIN=N

**Example:**

TERMIN=5

Describer	Meaning
N	ID of a matching TERMIN Bulk Data entry specifying the termination conditions for a particular analysis.

**Remarks:**

1. This entry may only be used within subcases (it may not be placed above the first subcase entry). If there are no subcase entries, it may be placed anywhere in the Case Control Section.
2. Most SOL 600 analyses do not require TERMIN entries.
3. If some subcases have TERMIN entries and others do not, only those that do will check for termination conditions.
4. TERMIN criteria may be different for different subcases.

## TFL Transfer Function Set Selection

Selects the transfer function set(s) to be added to the direct input matrices.

### Format:

TFL=n

### Example:

TFL=77

TFL = 1, 25, 77

Describer	Meaning
n	Set identification of a TF Bulk Data entry (Integer > 0).

### Remarks:

1. Transfer functions will not be used unless selected in the Case Control Section.
2. Transfer functions are supported in dynamics problems only.
3. Transfer functions are described in the [MSC Nastran Dynamic Analysis User's Guide](#).
4. It is recommended that PARAM,AUTOSPC,NO be specified when using transfer functions. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide*.
5. The transfer functions are additive if multiple TF values are referenced on the TFL command.

**THERMAL**

## Temperature Output Request

Requests the form and type of temperature output.

**Format:**

$$\text{THERMAL} \left( \left( \begin{array}{l} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) \right) = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Examples:**

THERMAL=5

THER (PRINT, PUNCH) =ALL

Descriptor	Meaning
SORT1	Output is presented as a tabular listing of point temperatures for each load or time step.
SORT2	Output is presented as a tabular listing of loads or time steps for each.

	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Temperatures for all points will be output.
NONE	Temperatures for no points will be output.
n	Set identification of a previously appearing SET command. Only temperatures of points with identification numbers that appear on this SET command will be output (Integer > 0).

**Remarks:**

1. The THERMAL output request is designed for use with the heat transfer option. The printed output will have temperature headings. The PUNCH option produces TEMP Bulk Data entries, and the SID on the entries will be the subcase number (=1 if no SUBCASES are specified).

2. SORT1 is the default in steady state heat transfer analysis. SORT2 is the default in transient heat transfer analysis.
3. In a transient heat transfer analysis, the SID on the punched TEMP Bulk Data entries equal the time step number.

**TITLE**                      Output Title

Defines a character string to appear on the first heading line of each page of MSC Nastran printer output.

**Format:**

TITLE=title

**Example:**

TITLE=RIGHT WING, LOAD CASE 3.

Describer	Meaning
title	Any character string.

**Remarks:**

1. If this command appears under a SUBCASE command, then the title appears in the output for that subcase only.
2. If this command appears before all SUBCASE commands, then the title is used in all subcases without a TITLE command.
3. If no TITLE command is present, then the title line will contain data and page numbers only.
4. The title also appears on plotter output.

**TIRE**

## Tire Set Selection

The TIRE case control entry defines the tires to be included in the analysis by selecting the TIRE bulk data entry. This case control command can be used in solutions sequences SOLs 103, 107-112, 200 and 400.

**Format:**

TIRE = n

**Example:**

TIRE = 100

Descriptor	Meaning
n	Set ID of a TIRE bulk data entry (Integer > 0).

**Remarks:**

1. The TIRE case control command must occur above subcase level.
2. Using a particular combination of tires in the analysis only requires selection of the tires by referencing the TIRE bulk data entry.



**TRIM**

## Aerodynamic Trim Variable Constraint Selection

Selects trim variable constraints in static aeroelastic response.

**Format:**

TRIM = n

**Example:**

TRIM=1

Describer	Meaning
n	Set identification number of a TRIM Bulk Data entry (Integer > 0).

**Remark:**

1. Aerodynamic extra points (trim variables) not constrained by a TRIM Bulk Data entry will be free during the static aeroelastic response solution.

## TRIMF

### Trim Load Output Specification

Specifies options for the output of trim loads from a static aeroelastic analysis as FORCE/MOMENT Bulk Data entries.

#### Format:

TRIMF([UNIT = i], [LOADSET = n], [LARGE], [INERTIA], [APPLIED], [AIR]

[NOSUM][RIGID],[NOELASTIC], [QNORM] )=  $\left\{ \begin{array}{c} \text{ALL} \\ n \end{array} \right\}$

#### Example:

TRIMF (LOADSET=10001, LARGE) =ALL  
TRIMF (UNIT=59, INERTIA, NOSUM) =1

Descriptor	Meaning
UNIT	Fortran unit to which data are written. (Optional; Default = 7) (punch file).
LOADSET	Load set id for output bulk data entries. If the TRIMF specification results in multiple load sets, then the defined ID will be used for the first and each subsequent load set has an ID incremented by 1. (Optional; Default = 1)
LARGE	Write the output data in large field format (16 characters per field). The default is 8 characters per field.
INERTIA	Write out inertial loads as a separate load set. By default, the separate load set will not be written.
APPLIED	Write out applied loads as a separate load set. By default, the separate load set will not be written.
AIR	Write out aerodynamic loads as a separate load set. By default, the separate load set will not be written.
NOSUM	By default, the sum of the inertial, applied, and aerodynamic loads will be written as a separate load set. This option suppresses the writing of that set of loads.
RIGID	Write out rigid instances of the selected loads (Inertial, Applied, Air and/or Sum) as separate load sets. By default, the separate load set will not be written.
NOELASTIC	By default, the sum of the rigid and elastic increment loads will be written as a separate load set. This option suppresses the writing of that set of loads.
QNORM	Normalize the load by the dynamic pressure used in the trim analysis. By default, the loads are not normalized.
ALL	Loads for all points will be output
n	Set identification of a previously appearing SET command. Only loads on points with identification numbers that appear on this SET command will be output (Integer > 0)

**Remark:**

1. By default, the loads are written to the punch file (Fortran unit 7). If the user specifies an alternate Fortran unit number on the TRIMF entry, by default the loads will be written to a file name that is machine specific (i.e. 'fort.53' on many LINUX platforms). The user may connect the Fortran unit to a user-defined file name by using an ASSIGN entry in the FMS Section of the input file. For example:

```
ASSIGN USERFILE='load13.inc', STATUS=UNKNOWN, FORMATTED, UNIT=53
```

2. Up to eight loads sets are available: Rigid Inertial, Rigid Applied, Rigid Air, Rigid Sum and four more with the sum of the rigid and elastic increment. This table indicates how the describers invoke each of these sets:

	INERTIAL	APPLIED	AIR	SUM
RIGID	O	O	O	O
ELASTIC	O	O	O	DEFAULT

where O = Optional. For a load to appear, both the row and column in the table above have to be set.

3. Care must be taken if LOADSET is specified in a run with multiple subcases. There are no checks that the load set IDs which are generated by one subcase are not also used for another subcase. For example, consider the following Case Control commands:

```
SUBCASE 1
  TRIM = 1
  TRIMF(RIGID) = ALL $
SUBCASE 2
  TRIM = 2
  TRIMF(LOADSET=2) = ALL
```

Subcase 1 will generate two load sets with set IDs 1 and 2. Subcase 2 will also output a load set ID 2.

4. The LOADSET option should not be specified above the subcase level when there are multiple subcases. If it is, each subcase will start numbering its load ID's from LOADSET.

**TRIMGRP**

## Selection of Trim Component(s)

Selects a set of trim components for analysis.

**Format:**

$$\text{TRIMGRP} \left[ \begin{matrix} (ALLTRMC) \\ (SLTTRMC) \end{matrix} \right] = \begin{bmatrix} sid \\ ALL \\ NONE \end{bmatrix}$$

**Example:**

TRIMGRP=101

Describer	Meaning
ALLTRMC	Process all TRMCs defined under 'BEGIN BULK TRMC=trimid' regardless if it is been selected in sid. This is the default option.
SLTTRMC	Process only those TRMCs whose ID is selected in sid. It can be equal to or less than what are been defined under 'BEGIN BULK TRMC=trimid'.
sid	Set identification of a previously appearing SET command or TRIMID. (Integer>0).
ALL	All trim components present in the model are included in the analysis.
NONE	This is the default of TRIMGRP. No trim component is involved in the analysis.

**Remark:**

1. TRIMGRP is used to select a group of trim components for analysis, including the calculation of trim component boundary coupling matrices.:
2. PEM capability is available in SOL 108, SOL 111 and SOL 200 with analysis=mfreq.
3. SUBCASEs without TRIMGRP will have the equivalent effect of TRIMGRP=none even with the presence of 'BEGIN TRMC' in the deck.
4. To run PEM job with multiple processors, additional information is available in 'Running PEM jobs' section of [Using PEM Functions in MSC Nastran](#) in the *MSC Nastran Installation and Operations Guide*.
5. For restart PEM job, TRIMGRP must remain the same as cold start PEM job and no changes to any TRMC should be observed. Otherwise, the PEM job should be run as cold start.
6. Collapsed logic is introduced to improve performance of large PEM jobs satisfy following requirements:
  - a. Same TRIMGRP for all SUBCASEs and
  - b. same master frequencies for all TRMCs. The activation of collapsed logic is automatic if requirements are met. With collapsed logic activated, restart job is restricted to the same TRIMGRP as cold start.

7. Following TRIMGRP case control commands have same effect which is to process all defined TRMCs:
  - a. TRIMGRP = ALL
  - b. TRIMGRP(alltrmc)=ALL
  - c. TRIMGRP(sltrmc)=ALL
8. If TRIMGRP is used in multiple SUBCASEs, SLTTRMC/ALLTRMC on TRIMGRP of above subcase level or first subcase will be utilized. SLTTRMC/ALLTRMC on TRIMGRP of second subcase and onwards will be ignored. Note that SLTTRMC on TRIMGRP of first subcase may be ignored if the union of all TRIMGRP means ALL TRMCs.

## TSTEP

### Transient Time Step Set Selection

Selects integration and output time steps for linear or nonlinear transient analysis.

#### Format:

TSTEP=n

#### Example:

TSTEP=7 3 1

Describer	Meaning
n	Set identification number of a TSTEP or Bulk Data entry (Integer > 0).

#### Remarks:

1. A TSTEP entry must be selected to execute a linear transient analysis (SOLs 109 or 112) and for a nonlinear transient analysis (SOLs 129 and 159).
2. A entry must be selected in each subcase to execute a nonlinear transient problem.
3. For the application of time-dependent loads in modal frequency response analysis (SOLs 111 and 146), the TSTEP entry must be selected by the TSTEP command. The time-dependent loads will be recomputed in frequency domain by a Fourier transform.
4. In one subcase or STEP for SOL 400, users should only specify one of TSTEP, TSTEPNL or NLSTEP.

**TSTEPNL**

## Transient Time Step Set Selection for Nonlinear Analysis

See the description of the [TSTEP \(Case\)](#). (The TSTEP Case Control entry can be used to select a TSTEPNL Bulk Data entry, however, a Bulk Data TSTEP is completely different from a Bulk Data Entry TSTEPNL.)

## TSTRU

## Temperature Set ID for a Structures Run

Defines a temperature set ID for a structures run based on a heat transfer subcase.

### Format:

TSTRU=n

### Example:

TSTRU=999

Describer	Meaning
n	Set identification for use on TEMP(LOAD)=n or TEMP(INIT)=n

### Remarks:

1. TSTRU should be placed in a heat transfer subcase.
2. If TSTRU does not explicitly appear in the heat transfer subcase, it is defaulted to TSTRU=heat transfer subcase ID.
3. In a structures run, a temperature set generated from a heat transfer run will override an existing temperature set with identical set ID defined with TEMP, TEMPD, TEMPP1, TEMPRB, or any combination.
4. TSTRU may be placed in the first subcase of a PARAM,HEATSTAT,YES run.
5. TSTRUs may be placed in each subcase of an APPHEAT run. The associated structural analysis then requires the following:

```
ASSIGN heat_run='heat transfer job name.MASTER'  
DBLOC DATABLK=(UG,EST,BGPDTS,CASECCR/CASEHEAT) LOGICAL=heat_run
```

6. Heat transfer runs and structural runs must have the same mesh.
7. For nonlinear heat transfer SOL 106 or SOL 153, the INOUT field on the NLPARM Bulk Data entry must be blank or NO if the results of the run are to be transferred to a linear structures run.

```
PARAM,NLHTLS,-1
```

The above parameter should be placed in the nonlinear heat run. This will place UG heat transfer on the database.



**TVELOCITY**

## Velocity Output Request for trim components, TRMC

Requests the form and type of velocity output for TRMC.

**Format:**

$$TVELOCITY \left( \left( \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right), \left[ \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right] \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$

**Example:**

```
SET 20 = 3/103, 5/0, 12/ALL $
TVELOCITY=ALL
TVEL (REAL, PUNCH, PRINT)=20
TVELO=20
TVELOCI (SORT2, PRINT)=20
```

Describer	Meaning												
SORT1	Output will be presented as a tabular listing of grid points for each load and frequency.												
SORT2	Output will be presented as a tabular listing of frequency for each grid point.												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

ALL TRMC velocity for all points of all TRMCs will be output. See Remarks 2.

Describer	Meaning
NONE	No TRMC velocity will be output.
n	Set identification of a previously appearing SET command. Only points with identification numbers that appear on this SET command will be output (Integer > 0). SET 20 in above examples section is to request SET 103(not present) for TRMC 3, none for TRMC 5 and ALL for TRMC 12.

**Remarks:**

1. Refrain from using ALL which may produce voluminous output.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. TVELO=NONE suppresses the generation of TRMC velocity output.
4. TVELOCITY is supported only with PARAM,TRMBIM,PHYSICAL
5. TVELOCITY is available for PEM jobs of SOL 108, 111 and SOL 200 with ANALYSIS=DFREQ and MFREQ. Note that TVELOCITY is available for output only and can not be utilized as design response.
6. For restart PEM job, TVELO request must remain the same as cold start PEM job.

## UNGLUE

### Contact Body Unglue Selection

Selects the grids should use standard contact instead of glued contact in glued bodies in SOL 400.

**Format:**

UNGLUE=*n*

**Example:**

UNGLUE=10

Describer	Meaning
<i>n</i>	Set identification number of the UNGLUE Bulk Data entry (Integer > 0).

**Remarks:**

1. This command is used only in SOL 400 for 3D Contact analysis.
2. The default SID of UNGLUE Bulk Data entry is defined on BCONTACT Case Control command if applicable; however, the SID on UNGLUE Case Control command can overwrite it.

## VCCT

## Virtual Crack Closure Technique for SOLs 400/600 Analysis

Selects grid sets to be used for virtual crack closure analysis in SOL 600 and SOL 400.

### Format:

VCCT=N

### Example:

VCCT=0

VCCT=1

Describer	Meaning
N	ID of a matching Bulk Data VCCT entry specifying the crack.

### Remarks:

1. This entry can only be used in SOLs 400/600.
2. Different sets of cracks can be selected for different subcases using this option.
3. For SOL 600, N=0 may be entered above any subcases; then, the Bulk Data entry VCCT with ID=0 will be used in the Marc model definition section. The fracture mechanics calculations will be performed for all subcases. Otherwise, if  $N > 0$ , the matching Bulk Data entry VCCT will be used in Marc's history definition section for the applicable subcase, and all subsequent subcases, until a new VCCT is activated.

**VECTOR**

## Displacement Output Request

Requests the form and type of displacement vector output.

See the description of the [DISPLACEMENT \(Case\)](#).

## VELOCITY

### Velocity Output Request

Requests the form and type of velocity vector output.

#### Format:

$$\text{VELOCITY} \left( \left[ \begin{array}{c} \text{SORT1} \\ \text{SORT2} \end{array} \right], \left[ \text{PRINT}, \text{PUNCH}, \text{PLOT} \right], \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], \left[ \begin{array}{c} \text{PSDF, ATOC, CRMS} \\ \text{or RALL} \end{array} \right], \right. \\
 \left. \left[ \begin{array}{c} \text{RPRINT} \\ \text{NORPRINT} \end{array} \right], \text{RPUNCH} \right], [\text{CID}] \\
 = \left\{ \begin{array}{c} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

#### Examples:

VELOCITY=5

VELOCITY (SORT2, PHASE, PUNCH) =ALL

VELOCITY (SORT2, PRINT, PSDF, CRMS, RPUNCH) =20

VELOCITY (PRINT, RALL, NORPRINT) =ALL

Descriptor	Meaning												
SORT1	Output will be presented as a tabular listing of grid points for each load, frequency, eigenvalue, or time, depending on the solution sequence.												
SORT2	Output will be presented as a tabular listing of frequency or time for each grid point.												
	<table border="1"> <thead> <tr> <th>Printer File (.f06)</th> <th>Punch File (.pch)</th> <th>Plot File (.op2/.h5)</th> </tr> </thead> <tbody> <tr> <td>PRINT or (blank)</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PUNCH</td> <td>X</td> <td>X*</td> </tr> <tr> <td>PLOT</td> <td></td> <td>X*</td> </tr> </tbody> </table>	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)	PRINT or (blank)	X	X*	PUNCH	X	X*	PLOT		X*
Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)											
PRINT or (blank)	X	X*											
PUNCH	X	X*											
PLOT		X*											

\* The .op2 binary database file will be generated with "PARAM,POST, X" (or the POST Case Control command), while the .h5 binary database file will be generated with "MDLPRM,HDF5, X" specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

REAL or IMAG Requests rectangular format (real and imaginary) of complex output. Use of either REAL or IMAG yields the same output.

PHASE Requests polar format (magnitude and phase) of complex output. Phase output is in degrees.

Describer	Meaning
PSDF	Requests the power spectral density function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
ATOC	Requests the autocorrelation function be calculated and stored in the database for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
CRMS	Requests the cumulative root mean square function be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
RALL	Requests all of PSDF, ATOC, and CRMS be calculated for random analysis postprocessing. Request must be made above the subcase level, and RANDOM must be selected in Case Control. See Remark 5.
RPRINT	Writes random analysis results in the print file (Default).
NORPRINT	Disables the writing of random analysis results in the print file.
RPUNCH	Writes random analysis results in the punch file.
CID	Request to print output coordinate system ID in printed output (.f06) file.
ALL	Velocity for all solution points will be output.
NONE	Velocity for no solution points will be output.
n	Set identification of a previously appearing SET command. Only velocities of points with identification numbers that appear on this SET command will be output (Integer > 0).

### Remarks:

1. Velocity output is only available for transient and frequency response problems.
2. See Remark 1 under [DISPLACEMENT \(Case\)](#) for a discussion of SORT1 and SORT2.
3. VELOCITY=NONE overrides an overall output request.
4. Velocity results are output in the global coordinate system (see field CD on the GRID Bulk Data entry).
5. The option of PSDF, ATOC, CRMS, and RALL, or any combination of them, can be selected for random analysis. Here PSDF, ATOC and CRMS output is always in SORT2 format.
6. Note that the CID keyword affects only grid point related output, such as DISP, VELO, ACCEL, OLOAD, SPCF and MPCF. In addition, the CID keyword needs to appear only once in a grid point-related output request anywhere in the Case Control Section to turn on the printing algorithm.

## VINTENSITY

## Vibration Intensity, VI, Output Request

Request output of vibration intensity for structural elements in SOLs 108 and 111 only.

### Format:

$$\text{VINTENSITY} \left( \begin{array}{l} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right) = \left\{ \begin{array}{l} \text{ALL} \\ \text{n} \\ \text{NONE} \end{array} \right\}$$

### Examples:

VINTENSITY = ALL

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST, X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5, X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

ALL	Vibration Intensities will be computed for all supported structural elements.
n	Set identification of a previously defined set of structural elements. Vibration Intensities will be computed for the structural elements in this set only.
NONE	Vibration intensity will not be processed.

### Remarks:

1. VINTENSITY = NONE overrides an overall request.
2. This Case Control command can be used in SOL 108 and SOL 111 only
3. Structural element types covered for VI computations are BAR, BEAM, QUAD4, TRIA3, QUADR, TRIAR, QUAD8, TRIA6, HEXA, PENTA and TETRA.
4. For BAR/BEAM element types, VI is computed in element axial direction using following equation



$$VI_x = \text{Real}[-(F_x v_x^* + V_1 v_y^* + V_2 v_z^* + T \omega_x^* - M_2 \omega_y^* + M_1 \omega_z^*)] / \text{Area},$$

- Where
- $F_x$  = axial force in element x direction,
  - $V_1$  = shear force in element y direction,
  - $V_2$  = shear force in element z direction,
  - $T$  = torsion about element x direction,
  - $M_2$  = Bending moment in element y direction,
  - $M_1$  = Bending moment in element z direction,
  - $v_i$  = Translational velocity about element i direction,
  - $\omega_i$  = Rotational velocity about element i direction,

'\*' superscript denotes complex conjugate of the complex value.  $VI_x$  is computed at both end of BAR/BEAM element and the average of ends VI is presented as element  $VI_x$ .

5. For 2D element types, such as QUAD4/TRIA3, VI is computed in element coordinate system using following equations

$$VI_x = \text{REAL}[-(V_x v_z^* - M_x \omega_y^* + M_{xy} \omega_x^* + F_x v_x^* + F_{xy} v_y^*)] / \text{Areax},$$

$$VI_y = \text{REAL}[-(V_y v_z^* + M_y \omega_x^* - M_{xy} \omega_y^* + F_y v_y^* + F_{yx} v_x^*)] / \text{Areay},$$

- Where
- $V_x, V_y$  = transverse shear forces,
  - $M_x, M_y$  = Bending moments,
  - $M_{xy}$  = twisting moment,
  - $F_x, F_y$  = Membrane forces,
  - $F_{xy}, F_{yx}$  = Membrane shear,
  - $\text{Areax}, \text{Areay}$  = Area in the corresponding element axes,
  - $v_i$  = Translational velocity about element i direction,
  - $\omega_i$  = Rotational velocity about element i direction,

'\*' superscript denotes complex conjugate of the complex value.  $VI_x, VI_y$  are computed in element coordinate system using average corner velocities and the element forces at the element center.

6. For 3D element types, VI is computed in basic coordinate system using following equations

$$VI_x = \text{REAL}[-(\sigma_x v_x + \tau_{xy} v_y + \tau_{xz} v_z)]$$

$$VI_y = \text{REAL}[-(\sigma_y v_y + \tau_{yx} v_x + \tau_{yz} v_z)]$$

$$VI_z = \text{REAL}[-(\sigma_z v_z + \tau_{zx} v_x + \tau_{zy} v_y)]$$

Where  $VI_x, VI_y, VI_z = VI$  in basic coordinate system,  
 $\sigma_x, \sigma_y, \sigma_z =$  Normal stresses,  
 $\tau_{xy}, \tau_{yz}, \tau_{xz} =$  Shear stresses,  
 $v_i =$  Translational velocity in basic i direction,

7. VI for all element types is presented in basic coordinate system in print and/or punch output.

**WEIGHTCHECK**

## Rigid Body Mass Reduction Check

At each stage of the mass matrix reduction, compute rigid body mass and compare with the rigid body mass in the g-set.

**Format:**

$$\text{WEIGHTCHECK} \left( \left( \begin{array}{l} \text{PRINT} \\ \text{NOPRINT} \end{array} \right) \text{PUNCH}, \text{SET} = \left( \begin{array}{l} \text{G}, \text{N}, \text{N} + \text{AUTOSPC}, \text{F}, \text{A}, \text{V} \\ \text{ALL} \end{array} \right) \right) = \begin{cases} \text{YES} \\ \text{NO} \end{cases}$$

$$\left[ \text{GRID} = \text{gid}, \text{CGI} = \begin{array}{l} \text{YES} \\ \text{NO} \end{array}, \begin{array}{l} \text{WEIGHT} \\ \text{MASS} \end{array} \right)$$

**Examples:**

WEIGHTCHECK=YES

WEIGHTCHECK (GRID=12, SET=(G, N, A), MASS)=YES

Describer	Meaning
PRINT	Write output to the print file (Default).
NOPRINT	Do not write output to the print file.
PUNCH	Write output to the punch file.
SET	Selects degree of freedom set(s) (Default SET=G).
gid	Reference grid point for the calculation of rigid body motion. The default is the origin of the basic coordinate system.
CGI	For SET ≠ G, CGI = YES requests output of center of gravity and mass moments of inertia (Default: CGI = NO).
WEIGHT/MASS	Selects output in units of weight or mass (Default = WEIGHT).

**Remarks:**

1. WEIGHTCHECK must be specified above the subcase level.
2. For SET=N, N+AUTOSPC, F, or A, the WEIGHTCHECK command also outputs a percentage loss or gain in the reduced rigid body mass matrix (e.g., MAA) as compared to the g-set rigid body mass matrix (e.g., MGG). G must also be requested to obtain this comparison; e.g., WEIGHTCHECK(SET=(G,A))=YES.
3. SET=N+AUTOSPC uses the mass matrix for the n-set with the rows corresponding to degrees of freedom constrained by the PARAM, AUTOSPC operation zeroed out. If AUTOSPC was not performed, then this check is redundant with respect to SET=N.
4. WEIGHTCHECK is available in all SOLs. However for the residual structure in SOLs 101, 105, 114, and 116, because no mass reduction is performed, only WEIGHTCHECK(SET=J) is available. The 'J' set does not include upstream superelements.

5. If Lagrange multipliers are present via RIGID=LAGRAN or LGELIM then for degree-of-freedom sets N, N+AUTOSPC, F, and A the check will be performed on degree-of-freedom sets NL, NL+AUTOSPC, FL, and AL. The output will also be labeled accordingly.

**WETSENS**

## Sensitivity Wetted Grids for SOL 108/111

Select SOLUTION frequencies and RESPONSE DOFs for the generation of sensitivity for wetted grids.

**Format:**

$$WETSENS \left( \begin{array}{c} \text{PRINT, PUNCH} \\ \text{PLOT} \end{array} \right), \left[ \begin{array}{c} \text{REAL or IMAG} \\ \text{PHASE} \end{array} \right], [\text{THRESH} = P], \text{RESPONSE} = r,$$

$$\left[ \text{SOLUTION} = \left\{ \begin{array}{c} \text{ALL} \\ \text{self} \end{array} \right\} \right],$$

$$[\text{WETTED, SQWETT}] = \left\{ \begin{array}{c} \text{ALL} \\ n \\ \text{NONE} \end{array} \right\}$$
**Example:**

```
SET 81 = 100.0, 120.0
SET 91 = 11240/T3, 4001/T1
SET 95 = 9000000 THRU 9000050
$
WETSENS (RESPONSE=91,solution=81,WETTED) = 95
```

Describer	Meaning		
	Printer File (.f06)	Punch File (.pch)	Plot File (.op2/.h5)
PRINT or (blank)	X		X*
PUNCH		X	X*
PLOT			X*

\* The .op2 binary database file will be generated with “PARAM,POST,X” (or the POST Case Control command), while the .h5 binary database file will be generated with “MDLPRM,HDF5,X” specified in Bulk Data Section. Both .op2 and .h5 file can be created simultaneously. Note .xdb file is being deprecated.

Descriptor	Meaning
THRESH	The magnitude of element sensitivity less than p will be suppressed in all output files: print, punch, plot, .op2, and .xdb. (Default = 0.0).
RESPONSE	Adjoint load response will be computed for unit load applied at grid point components in SET r.
SOLUTION	Frequency responses at these forcing frequencies, defined in setf, will be used for element sensitivity computation. (Default=all forcing frequencies)
WETTED	Sensitivity for wetted grids will be computed and output.
SQWETT	Squared sensitivity for wetted grids will be computed and output.
ALL	Sensitivities for all elements will be calculated.
n	Set identification number. Sensitivity for all elements specified on the SET n command will be calculated. The SET n command must be specified in the same subcase as the ELSENS command, or above all subcases ( Integer > 0 ). The IDs in set n must be GID (grid ID).
NONE	Elemental sensitivity will not be output.

**Remarks:**

1. Set r for RESPONSE on WETSENS is default to set r on ELSSENS. If no ELSSENS in the deck, set r for WETSENS must be provided.
2. The equations for various options of WETSENS

$$\text{WETSENS(WETTED)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r]$$

$$\text{WETSENS(SQWETT)} = [U_{\text{setf}}]^t [\text{AGG}] [U_r] + [U_{\text{setf}}]^{*t} [\text{AGG}]^* [U_r]^*$$

where  $[U_{\text{setf}}]$  is the displacement of SOLUTION  
 $[U_r]$  is the displacement of RESPONSE  
 $[\text{AGG}]$  is Fluid/Structure Coupling matrix  
 superscript \* means complex conjugate of the term.

## Case Control Applicability Tables

The following tables describe the applicability of Case Control commands to Solution Sequences:

<a href="#">Table 5-3</a> and <a href="#">Table 5-4</a>	SOLs (101 through 200) -- Subcase Definition, Superelement Control, and Auxiliary Model Control
<a href="#">Table 5-5</a> and <a href="#">Table 5-6</a>	SOLs (101 through 200) -- Data Selection
<a href="#">Table 5-7</a> and <a href="#">Table 5-8</a>	SOLs (101 through 200) -- Output Selection

Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ADACT	X	X								
ANALYSIS										
AUXCAS										
AUXMODEL										
BEGIN BULK	X	X	X	X	X	X	X	X	X	X
CAMPBELL					X			X		
EXTDRIN	X	X			X	X	X	X	X	X
EXTDROUT	X	X			X	X	X	X	X	X
EXTSEOUT	X	X			X	X	X	X	X	X
MASTER	X	X	X	X	X	X	X	X	X	X
MCFRACTION								X	X	X
MODES		X								
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X
OUTPUT (XYPLOT)	X	X	X	X	X	X	X	X	X	X
REPCASE	X	X								
SEALL	X	X	X	X	X	X	X	X	X	X
SEDR	X	X	X	X	X	X	X	X	X	X
SEDV										
SEEXCLUD	X	X	X	X	X	X	X	X	X	X
SEFINAL	X	X	X	X	X	X	X	X	X	X
SEKR	X	X	X	X	X	X	X	X	X	X
SELG	X		X	X		X	X		X	X
SELR	X		X	X		X	X		X	X
SEMR	X	X	X	X	X	X	X	X	X	X
SEMR		X	X		X	X	X	X	X	X
SERE										



Table 5-3 Case Control Commands in SOLs 101 Through 112 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
STOCHASTICS	X	X								
SUBCASE	X	X	X	X	X	X	X	X	X	X
SUBCOM	X									
SUBSEQ	X									
SUPER	X	X	X	X	X	X	X	X	X	X
SYM	X									
SYMCOM	X									
SYMSEQ	X									

Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
ADACT												
ANALYSIS									X		X	
AUXCAS											X	
AUXMODEL											X	
BEGIN BULK	X	X	X	X	X	X	X	X	X	X	X	
CAMPBELL											X	X
EXTDRIN												X
EXTDROUT												X
EXTSEOUT												X
MASTER	X	X	X	X	X	X	X	X	X	X	X	
MODES		X	X								X	
OUTPUT(blank)	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT(PLOT)	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT (POST) or SETS DEFINITION	X	X	X	X	X	X	X	X	X	X	X	
OUTPUT(XYPLOT)	X	X	X	X	X	X	X	X	X	X	X	
REPCASE											X	

Table 5-4 Case Control Commands in SOLs 114 Through 400 -- Subcase Definition, Superelement Control, and Auxiliary Model Control

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SEALL	X	X	X	X	X	X	X	X	X	X	X	
SEDR	X	X	X	X	X	X	X	X	X	X	X	
SEDV											X	
SEEXCLUD	X	X	X	X	X	X	X	X	X	X	X	
SEFINAL	X	X	X	X	X	X	X	X	X	X	X	
SEKR	X	X	X	X	X	X	X	X	X	X	X	
SELG	X		X	X	X	X		X	X	X	X	
SELR	X		X	X	X	X		X	X	X	X	
SEMR	X	X	X	X	X	X	X	X	X	X	X	
SEMR		X	X	X	X	X	X	X		X	X	
SERE											X	
STOCHASTICS											X	X
SUBCASE	X	X	X	X	X	X	X	X	X	X	X	
SUBCOM											X	
SUBSEQ											X	
SUPER	X	X	X	X	X	X	X	X	X	X	X	
SYM											X	
SYMCOM											X	
SYMSEQ											X	

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
ADAPT	X	X								
AUTOSPC	X	X	X		X	X	X	X	X	X
AXISYMME	X	X	X		X	X	X	X	X	X
B2GG	X	X	X	X	X	X	X	X	X	X
B2PP					X	X	X	X	X	X
BC		X								

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
CLOAD				X						
CMETHOD					X			X		
DEFORM	X		X							
DEGLB										
DESOBJ										
DESSUB										
DLOAD					X	X	X	X	X	X
DSYM										
FMETHOD										
FREQUENC						X			X	
GUST										
HARMONIC	X	X	X		X	X	X	X	X	X
IC							X			
K2GG	X	X	X	X	X	X	X	X	X	X
K2PP					X	X	X	X	X	X
LOAD	X		X	X		X	X		X	X
LOADSET	X		X	X		X	X		X	X
M2GG	X	X	X	X	X	X	X	X	X	X
M2PP					X	X	X	X	X	X
METHOD		X		X	X	X	X	X	X	X
MFLUID		X			X	X	X	X	X	X
MODTRAK										
MPC	X	X	X	X	X	X	X	X	X	X
NLPARM				X						
NONLINEA							X			X
OMODES		X	X					X*	X*	X*
P2G	X		X	X		X	X		X	X
RANDOM						X			X	
RESVEC		X		X	X	X	X	X	X	X
SDAMPING								X	X	X

Table 5-5 Case Control Commands in SOLs 101 Through 112 -- Data Selection (continued)

Command Name	Structured Solution Number									
	101	103	105	106	107	108	109	110	111	112
SDENSITY	X	X	X							
SMETHOD	X			X		X			X	
SPC	X	X	X	X	X	X	X	X	X	X
STATSUB*	X	X	X		X	X	X	X	X	X
SUPPORT1	X	X	X	X	X	X	X	X	X	X
TEMPER(INIT)				X						
TEMPER(LOAD)	X		X	X		X	X		X	X
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X
TFL					X	X	X	X	X	X
TRIM										
TRIMGRP									X	
TSTEP							X			X
WEIGHTCHECK	X	X	X	X	X	X	X	X	X	X

\*If STATSUB is specified, then the Case Control commands that select static loads become applicable to the solution sequence supporting STATSUB.

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
ADAPT												
AUTOSPC	X	X	X	X		X	X	X		X	X	
AXISYMME											X	
B2GG	X	X	X	X	X	X	X	X	X	X	X	X
B2PP				X	X		X	X		X	X	
BC												
CLOAD									X			
CMETHOD							X					
DEFORM	X		X			X					X	
DESGLB											X	
DESMOD											X	

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
DESOBJ											X	
DESSUB											X	
DESVAR											X	
DLOAD				X	X		X	X		X	X	
DSYM	X	X	X	X								
FMETHOD							X				X	
FREQUENC				X				X			X	
GUST								X	X			
HARMONIC	X	X	X	X							X	
IC										X		
K2GG	X	X	X	X	X	X	X	X	X	X	X	X
K2PP				X	X		X	X		X	X	
LOAD	X		X	X	X	X		X	X		X	
LOADSET	X		X	X	X	X		X	X	X	X	
M2GG	X	X	X	X	X	X	X	X	X	X	X	X
M2PP				X	X		X	X		X	X	
METHOD		X	X	X	X	X	X	X	X	X	X	
MFLUID					X	X	X	X			X	
MODTRAK											X	
MPC	X	X	X	X	X	X	X	X	X	X	X	
NLBUCK												X
NLPARAM									X			
NONLINEA					X					X		
OMODES		X									X	
P2G	X		X	X	X	X		X	X		X	
RANDOM				X				X				
RESVEC		X		X	X	X	X	X	X	X	X	
SDAMPING							X	X			X	
SMETHOD												
SPC	X	X	X	X	X	X	X	X	X	X	X	
STATSUB				X							X	

Table 5-6 Case Control Commands in SOLs 114 Through 400 -- Data Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SUPPORT1	X	X	X	X	X	X	X	X	X	X	X	
TEMPER(INIT)												
TEMPER(LOAD)	X			X		X		X			X	
TEMPER(MATE)	X	X	X	X	X	X	X	X	X	X	X	
TFL				X	X		X	X		X	X	
TRIM						X					X	
TRIMGRP											X	
TSTEP					X			X		X	X	
WEIGHTCHECK	X	X	X	X	X	X	X	X			X	

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection

Command Name	Solution Number										
	101	103	105	106	107	108	109	110	111	112	
ACCELERA		X				X	X		X	X	
ACFPMRESULT						X				X	
ACPOWER						X				X	
AEROF											
APRESSURE											
BOUTPUT				X							
CMSENRGY		X		X				X	X	X	
DATAREC	X	X				X	X		X	X	
DISPLACE	X	X	X	X	X	X	X	X	X	X	
DSAPRT											
ECHO	X	X	X	X	X	X	X	X	X	X	
EDE		X			X	X	X	X*	X	X	
EKE		X			X	X	X	X*	X	X	
ELSDCON	X		X								
ELSUM	X	X	X	X	X	X	X	X	X	X	
ENTHALPY											
ERP						X			X		

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
ESE	X	X	X	X	X	X	X	X*	X	X
FLUX	X									
FORCE	X	X	X	X	X	X	X	X	X	X
GPFORCE	X	X	X	X			X+			X+
GPKE		X						X*	X*	X*
GPSDCON	X		X							
GPSTRAIN	X	X		X			X			X
GPSTRESS	X	X		X			X			X
GROUNDCHECK	X	X	X	X	X	X	X	X	X	X
HARMONIC	X	X	X		X	X	X	X	X	X
HDOT										
HOUTPUT										
INTENSITY						X				X
LABEL	X	X	X	X	X	X	X	X	X	X
LINE	X	X	X	X	X	X	X	X	X	X
MAXLINES	X	X	X				X			X
MAXMIN (old form)	X	X	X				X			X
MAXMIN (DEF)	X	X	X	X			X			X
MPCFORCE	X	X	X		X	X	X	X	X	X
MPRES		X			X	X	X	X	X	X
NLOAD							X			X
NOUTPUT										
OFREQUEN						X			X	
OLOAD	X		X	X		X	X		X	X
OTIME							X			X
PAGE	X	X	X	X	X	X	X	X	X	X
PARTN	X	X	X	X	X	X	X	X	X	X
PLOTID	X	X	X	X	X	X	X	X	X	X
POST	X									
PRESSURE		X			X	X	X			
ROTSEKE					X			X		

Table 5-7 Case Control Commands in SOLs 101 Through 112 -- Output Selection (continued)

Command Name	Solution Number									
	101	103	105	106	107	108	109	110	111	112
SACCELER						X	X		X	X
SDISPLAC		X	X		X	X	X		X	X
SET	X	X	X	X	X	X	X	X	X	X
SKIP	X	X	X	X	X	X	X	X	X	X
SPCFORCE	X	X	X	X	X	X	X	X	X	X
STRAIN	X	X	X	X	X	X	X	X	X	X
STRESS	X	X	X	X	X	X	X	X	X	X
STRFIELD	X	X	X				X			X
SUBTITLE	X	X	X	X	X	X	X	X	X	X
SURFACE	X	X		X			X			X
SVECTOR		X	X					X	X	X
SVELOCITY						X	X		X	X
THERMAL	X									
TITLE	X	X	X	X	X	X	X	X	X	X
VECTOR	X	X	X	X	X	X	X	X	X	X
VELOCITY						X	X		X	X
VOLUME	X	X		X			X			X
*For modal part of solution.										
†Forces limited to stiffness contributions.										



Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
ACCELERA				X	X			X			X	
ACFPMRESULT												
ACPOWER												
AEROF						X	X	X			X	
APRESSURE						X					X	
BOUTPUT					X				X	X		
CMSENRGY							X	X			X	
DATAREC											X	
DISPLACE	X	X	X	X	X	X	X	X	X	X	X	
DSAPRT											X	
ECHO	X	X	X	X	X	X	X	X	X	X	X	
EDE		X		X							X	
EKE		X		X							X	
ELSDCON						X					X	
ELSUM	X	X	X	X	X	X	X	X			X	
ENTHALPY										X		
ERP											X	X
ESE	X	X	X	X		X					X	
FLUX	X								X	X	X	
FORCE	X	X	X	X	X	X	X	X	X	X	X	
GPFORCE	X	X	X	X							X	
GPKE		X									X	
GPSDCON	X					X			X		X	
GPSTRAIN	X	X	X		X			X			X	
GPSTRESS	X	X	X		X	X					X	X
GROUNDCHCK	X	X	X	X	X	X	X	X			X	
HARMONY	X	X	X	X							X	
HDOT										X		
HOUTPUT	X	X	X	X								

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
INTENSITY												
LABEL	X	X	X	X	X	X	X	X	X	X	X	
LINE	X	X	X	X	X	X	X	X	X	X	X	
MAXLINES	X	X	X	X	X	X	X	X	X	X	X	
MAXMIN(DEF)												X
MPCFORCE	X	X	X	X		X	X	X			X	
MPRES		X		X		X	X	X			X	
NLLOAD										X		
NLOPRM												X
NLSTEP												X
NOUTPUT	X	X	X	X								
OFREQUEN				X				X			X	
OLOAD	X		X	X	X	X		X	X	X	X	
OTIME	X		X	X	X	X		X	X	X	X	X
PAGE	X	X	X	X	X	X	X	X	X	X	X	
PARTN	X	X	X	X	X	X	X	X	X	X	X	
PLOTID	X	X	X	X	X	X	X	X	X	X	X	
PRESSURE											X	
ROTSEKE												X
SACCELER				X	X			X			X	
SDISPLAC		X		X	X		X	X		X	X	
SET	X	X	X	X	X	X	X	X	X	X	X	
SKIP	X	X	X	X	X	X	X	X	X	X	X	
SPCFORCE	X	X	X	X	X	X	X	X	X	X	X	
STRAIN	X	X	X	X	X	X	X	X	X	X	X	
STRESS	X	X	X	X	X	X	X	X	X	X	X	
STRFIELD	X	X	X			X					X	
SUBSET												X
SUBTITLE	X	X	X	X	X	X	X	X	X	X	X	

Table 5-8 Case Control Commands in SOLs 114 Through 400 --Output Selection (continued)

Command Name	Solution Number											
	114	115	116	118	129	144	145	146	153	159	200	400
SURFACE	X	X	X								X	
SVECTOR		X	X	X			X		X		X	
SVELOCITY				X	X			X			X	
TEMPERATURE												
THERMAL	X								X	X	X	
TITLE	X	X	X	X	X	X	X	X	X	X	X	
TRIMF						X					X	
VECTOR	X	X	X	X	X	X	X	X	X	X	X	
VELOCITY				X	X			X			X	
VOLUME	X	X	X								X	

## X-Y PLOT Commands

The X-Y output request packet of the Case Control Section includes all commands between either OUTPUT(XYPLOT) or OUTPUT(XYOUT), and either BEGIN BULK or OUTPUT(PLOT). The remainder of this section describes the X-Y output commands.

A single set of plotted X-Y pairs is known as a curve. Curves are the entities to be plotted. The surface (paper, microfilm frame, etc.) on which one or more curves is plotted is known as a frame. Curves may be plotted on a whole frame, an upper-half frame, or a lower-half frame. Grid lines, tic marks, axes, axis labeling, and other graphic control options may be chosen by the user. The program will select defaults for parameters not selected by the user.

Only two commands are required for an X-Y output request.

1. Only one of OUTPUT(XYPLOT) or OUTPUT(XYOUT) at the beginning of the X-Y output command packet.
2. At least one of the commands XYPLOT, XYPEAK, XYPRINT, XYPUNCH, XYPAPLOT.

The commands OUTPUT(XYPLOT) and OUTPUT(XYOUT) are equivalent. If the X-Y output is to be printed and/or punched, a PLOTTER command is not required.

If only the required commands are used, the graphic control options will all assume default values. Curves using all default parameters have the following general characteristics.

1. Tic marks are drawn on all edges of the frame. Five spaces are provided on each edge of the frame.
2. All tic marks are labeled with their values.
3. Linear scales are used.
4. Scales are selected such that all points fall within the frame.
5. The plotter points are connected with straight lines.
6. The plotted points are not identified with symbols.

The above characteristics may be modified by inserting any of the parameter definition commands in the next section, ahead of the XY\_\_\_\_\_ command(s). The use of a parameter definition command sets the value of that parameter for all following command operation commands unless the CLEAR command is inserted. If grid lines are requested, they will be drawn at the locations of all tic marks that result from defaults or user request. The locations of tic marks (or grid lines) for logarithmic scales cannot be selected by the user. Values for logarithmic spacing are selected by the program. The values for the number of tic marks (or grid lines) per cycle depend on the number of logarithmic cycles required for the range of the plotted values.

The definition and rules for the X-Y output commands follow. The form of X-Y output commands differ in many instances from that of similar commands used in the OUTPUT(PLOT) section.

### X-Y Output Command Summary

	Commands Applied To All Curves
PLOTTER	Selects format of plot file for interpretation by plotter postprocessor.
CAMERA	Selects plotter media.

Commands Applied To All Curves	
PENSIZE	Selects pen number.
DENSITY	Selects the line density for microfilm plotters only.
XPAPER	Defines the size of the paper in x-direction.
YPAPER	Defines the size of the paper in y-direction.
XMIN	Specifies the minimum value on the x-axis.
XMAX	Specifies the maximum value on the x-axis.
XLOG	Selects logarithmic or linear x-axis.
YAXIS	Controls the plotting of the y-axis on all curves.
XINTERCEPT	Specifies the location of the x-axis on the y-axis.
UPPER TICS	Specifies how to draw tic marks on upper edge.
LOWER TICS	Specifies how to draw tic marks on lower edge.
CURVELINESYMBOL	Selects lines and/or symbols to be drawn through the x-y points.
XDIVISIONS	Specifies spacing of tic marks on the x-axis for all curves.
XVALUE PRINT SKIP	Specifies how often to print the x-values alongside the x-axis tic marks.
CLEAR	Resets X-Y Plot commands to their default value.
XTITLE	Defines a character string that will appear along the x-axis.
TCURVE	Defines a character string that will appear at the top of the plot frame.
LONG	Controls amount of curve's summary printout.
CSCALE	Defines scale factor for characters in the plot frame.

Commands Applied to Whole Frame Curves Only	
YMIN	Specifies the minimum value on the y-axis.
YMAX	Specifies the maximum value on the y-axis.
XAXIS	Controls the plotting of the x-axis.
YINTERCEPT	Specifies the location of the y-axis on the x-axis.
YLOG	Selects logarithmic or linear y-axis.
LEFT TICS	Specifies how to draw tic marks on left edge.
RIGHT TICS	Specifies how to draw tic marks on right edge of the frame.
ALLEDGE TICS	Specifies how to draw tic marks on all edges of the frame.
YDIVISIONS	Specifies spacing of tic marks on the y-axis.
YVALUE PRINT SKIP	Specifies how often to print the y-values alongside the y-axis tic marks applies.

Commands Applied to Whole Frame Curves Only	
XGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YTITLE	Defines a character string that will appear along the y-axis

Commands Applied to Upper Half Frame Curves Only	
YTMIN	Specifies the minimum value on the y-axis.
YTMAX	Specifies the maximum value on the y-axis.
YTAXIS	Controls the plotting of the y-axis.
YTINTERCEPT	Specifies the location of the y-axis on the x-axis.
YTLOG	Selects logarithmic or linear y-axis.
TLEFT TICS	Specifies how to draw tic marks on the left edge.
TRIGHT TICS	Specifies how to draw tic marks on the right edges.
TALL EDGE TICS	Specifies how to draw tic marks on all edges.
YTDIVISIONS	Specifies spacing of tic marks on the y-axis.
YTVALUE PRINT SKIP	Specifies how often to print the y-values alongside the y-axis tic marks.
XTGRID LINES	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
YTGRID LINES	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
YTTITLE	Defines a character string that will appear along the y-axis.

Commands Applied to Lower Half Frame Curves Only	
YBMIN	Specifies the minimum value on the y-axis.
YBMAX	Specifies the maximum value on the y-axis.
XBAXIS	Controls the plotting of the x-axis.
YBINTERCEPT	Specifies the location of the y-axis on the x-axis.
YBLOG	Selects logarithmic or linear y-axis.
BLEFT TICS	Specifies how to draw tic marks on left edge.
BRIGHT TICS	Specifies how to draw tic marks on right edge.
BALL EDGE TICS	Specifies how to draw tic marks on all edges.
YBDIVISIONS	Specifies spacing of tic marks on the y-axis.

Commands Applied to Lower Half Frame Curves Only	
<a href="#">YBVALUE PRINT SKIP</a>	Specifies how often to print the y-values alongside the y-axis tic marks.
<a href="#">XBGRID LINES</a>	Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks.
<a href="#">YBGRID LINES</a>	Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks.
<a href="#">YBTITLE</a>	Defines a character string that will appear along the y-axis.

X-Y Plot Generation Commands	
<a href="#">XYPAPLOT</a>	Generate X-Y plots for a printer.
<a href="#">XYPEAK</a>	Print only the summary for all curves.
<a href="#">XYPLOT</a>	Generate X-Y plots for a plotter.
<a href="#">XYPRINT</a>	Generate table of X-Y pairs for a printer.
<a href="#">XYPUNCH</a>	Generate table of X-Y pairs for the PUNCH file

## ALLEEDGE TICS

### Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the frame.

Format:

ALLEEDGE TICS tic

Example:

ALLEEDGE -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer, Default= 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. ALLEEDGE TICS applies to whole frame curves only.
2. To determine if on any given edge (a) tic marks will be drawn without values, (b) no tic marks or values will be drawn, or (c) tic marks with values will be drawn, the following sum must be computed by the user. Add the tic values of the edge in question to its associated ALLEEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS tic values. If the resulting value is less than zero, tic marks will be drawn without values. If the resulting value is zero, no tic marks or values will be drawn. If the resulting value is greater than zero, tic marks with values will be drawn. The user should be careful in the use of the ALLEEDGE TICS, TALL EDGE TICS, or BALL EDGE TICS commands. For example, the use of only the ALLEEDGE TICS = -1 command will result in no tic marks or values being drawn, since the default values for individual edges is +1. Tic values input may only be -1, 0, or 1.



## BALL EDGE TICS

## Controls Drawing of Tic Marks on Lower Half

Specifies how to draw tic marks on lower half of frame.

Format:

BALL EDGE TICS tic

Example:

BALL EDGE TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer, Default = 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. BALL EDGE TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 670](#).

## BLEFT TICS

### Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of lower half of frame.

Format:

BLEFT TICS tic

Example:

BLEFT TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. BLEFT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 670](#).
3. See related command [BRIGHT TICS, 673](#).

**BRIGHT TICS****Controls Drawing of Tic Marks on Right Edge**

Specifies how to draw tic marks on right edge of lower half of frame.

Format:

BRIGHT TICS tic

Example:

BRIGHT TICS -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. BRIGHT TICS applies to lower frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 670](#).

## CAMERA

### Plotter Media Selection

Selects plotter media.

Format:

CAMERA ctype

Example:

CAMERA 1

Describer	Meaning
ctype	Camera type (Integer 1, 2, or 3; Default= 2).
1	Film
2	Paper (Default)
3	Both

Remark:

1. If the CAMERA command is not specified then CAMERA 2 is assumed.

**CLEAR****Resets X-Y Plot Commands**

Resets X-Y Plot commands to their default values.

Format

CLEAR

Remark:

1. All commands except XTITLE, YTITLE, YTTITLE, YBTITLE, and TCURVE will revert to their default values.

## CSCALE

### Character Scale Factor

Defines scale factor for characters in the plot frame. See the command [CSCALE, 755](#) in the OUTPUT(PLOT) Section.

**CURVELINESYMBOL**

## Curve, Line and Symbol Selection

Selects lines and/or symbols to be drawn through the x-y points.

Format:

CURVELINESYMBOL symtype

Example:

CURV 4

Describer	Meaning
symtype	Specifies the symbol drawn at the x-y points. If symtype is 0 then only lines will be drawn through the points with no symbol. If symtype is less than zero then only the symbol and not the lines will be drawn. If symtype is greater than zero then both the symbol and the lines will be drawn ( $-9 \leq \text{Integer} \leq 9$ ; Default= 0).

symtype	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	×
7	[ ]
8	<>
9	/\

Remark:

1. If more than one curve is plotted per frame, then the symbol number is incremented by 1 for each curve.

## DENSITY

### Microfilm Plotter Line Density

Selects the line density for microfilm plotters only.

Format

DENSITY d

Example

DENS 3

Describer	Meaning
d	Specifies line density scale factor for microfilm plotters. A line density of d is d times heavier than a line density of 1 (Integer $\geq 0$ ; Default = 1).



## LEFT TICS

### Controls Drawing of Tic Marks on Left Edge

Specifies how to draw tic marks on left edge of whole frame curves.

Format:

LEFT TICS tic

Example:

LEFT -1

Describer	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. LEFT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEDGE TICS, 670](#).
3. See related command [RIGHT TICS, 684](#).

## LONG

## Summary Print Control

Controls amount of curve's summary printout.

Format:

LONG { YES }  
      { NO }

Describers	Meaning
YES	One page for each curve's summary (Default).
NO	Condensed curve summary.

Remark:

1. If LONG is not specified, then LONG=NO is assumed.

## LOWER TICS

## Controls Drawing of Tic Marks on Lower Edge

Specifies how to draw tic marks on lower edge.

Format:

LOWER TICS tic

Example:

LOWER -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. LOWER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS](#).

## PENSIZE

### Pen Selection

Selects pen number.

Format:

PENSIZE p

Example:

```
PENS 3
```

Describer	Meaning
p	Specifies pen number to be used to generate the plot (Integer > 0; Default = 1).

## PLOTTER

### X-Y Plot File Format

See the command [PLOTTER](#) in the OUTPUT(PLOT) Section.

## RIGHT TICS

### Controls Drawing of Tic Marks on Right Edge

Specifies how to draw tic marks on right edge of the frame.

Format:

RIGHT TICS tic

Example:

RIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. RIGHT TICS applies to whole frame curves only.
2. See Remark 2 under [ALLEDGE TICS](#).
3. See related command [LEFT TICS](#).

## SEPLOT

### Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

Format:

```
SEPLOT seid1 [ seid2 ... ]
```

Examples:

```
SEPLOT 5
SEPLOT 0 3 7 200
```

Describer	Meaning
seidi	Superelement identification number (Integer $\geq 0$ ).

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

## SEUPPLOT

### Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

Format:

SEUPPLOT seid

Example:

SEUPPLOT 7

Describer	Meaning
seid	Superelement identification number (Integer $\geq 0$ ).

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.



## TALL EDGE TICS

## Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TALL EDGE TICS tic

Example:

TALL -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 0).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values (Default).
1	Draw tic marks and associated values.

Remarks:

1. TALL EDGE TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS](#).

## TCURVE Curve Title

Defines a character string that will appear at the top of the plot frame.

Format:

TCURVE ctitle

Example:

```
TCUR RIGHT WING -- LOAD CASE 3
```

Describer	Meaning
ctitle	Any character string (Character; Default = blank).

Remark:

1. TCURVE may not be continued to the next command line.

## TLEFT TICS

### Controls Drawing of Tic Marks on All Edges

Specifies how to draw tic marks on the left edge of the upper half of the frame.

Format:

TLEFT TICS tic

Example:

TLEFT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default=1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. TLEFT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS](#).
3. See related command [TRIGHT TICS](#).

## TRIGHT TICS

## Controls Drawing of Tic Marks on the Right Edge

Specifies how to draw tic marks on all edges of the upper half of the frame.

Format:

TRIGHT TICS tic

Example:

TRIGHT -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default = 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. TRIGHT TICS applies to upper half frame curves only.
2. See Remark 2 under [ALLEDGE TICS](#).
3. See related command [TLEFT TICS](#).

## UPPER TICS

### Controls Drawing Of Tic Marks On Upper Edge

Specifies how to draw tic marks on upper edge.

Format:

UPPER TICS tic

Example:

UPPER -1

Describers	Meaning
tic	Specifies how to draw tic marks (Integer; Default= 1).
-1	Draw tic marks only.
0	Do not draw tic marks or associated values.
1	Draw tic marks and associated values (Default).

Remarks:

1. UPPER TICS applies to all curves.
2. See Remark 2 under [ALLEDGE TICS](#).
3. See related command [LOWER TICS](#).

## XAXIS

### X-Axis Plot Control

Controls the plotting of the x-axis on whole frame curves only.

Format:

XAXIS { YES  
NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remarks:

1. XAXIS applies to whole frame curves only.
2. See related command [YAXIS](#).

**XBAXIS****X-Axis Plot Control**

Controls the plotting of the x-axis on lower half frame curves only.

Format:

XBAXIS { YES }  
          { NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remark:

1. XBAXIS applies to lower half frame curves only.

## **XBGRID LINES**

### Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on lower half frame curves only.

Format:

XBGRID LINES { YES  
NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XBGRID applies to lower half frame curves only.
2. See related command [YBGRID LINES](#).



**XDIVISIONS**

## Tic Spacing on Y-Axis

Specifies spacing of tic marks on the x-axis for all curves.

Format:

XDIVISIONS *xd*

Example:

XDIV 10

Describer	Meaning
<i>xd</i>	Number of spaces between tic marks on x-axis (Integer > 0; Default = 5).

Remarks:

1. XDIVISIONS applies to all curves and to the commands: UPPER TICS, LOWER TICS, and YINTERCEPT.
2. XDIVISIONS is ignored for a logarithmic x-axes.

## XGRID LINES

### Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on whole frame curves only.

Format:

XGRID LINES { YES }  
                  { NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XGRID applies to whole frame curves only.
2. See related command [YGRID LINES](#).

## XINTERCEPT

### Location of X-Axis on Y-Axis

Specifies the location of the x-axis on the y-axis.

Format:

XINTERCEPT xi

Example:

XINT 50 .

Describer	Meaning
xi	Location of x-axis on the y-axis (Real; Default = 0.0).

## XLOG                      Logarithmic or Linear X-Axis

Selects logarithmic or linear x-axis.

Format:

XLOG { YES  
       NO }

Describers	Meaning
YES	Plot a logarithmic x-axis.
NO	Plot a linear x-axis (Default).

Remarks:

1. XLOG applies to all curves.
2. The default value for tic division interval depends on the number of log cycles. The default values for tic divisions are given as follows but will range over whole cycles:

Number of Cycles	Intermediate Values
1, 2	2., 3., 4., 5., 6., 7., 8., 9.
3	2., 3., 5., 7., 9.,
4	2., 4., 6., 8.,
5	2., 5., 8.
6, 7	3., 6.
8, 9, 10	3.

## XMAX      Maximum X-Axis Value

Specifies the maximum value on the x-axis.

Format:

XMAX xmax

Example:

XMAX 100 .

Describer	Meaning
xmax	Maximum value on the x-axis. (Real)

Remarks:

1. If XMAX is not specified, then the maximum value is set to the highest value of x.
2. See related commands [XMIN](#), [YMIN](#), and [YMAX](#).

## XMIN Minimum X-Axis Value

Specifies the minimum value on the x-axis.

Format:

XMIN xmin

Example:

XMIN 100 .

Describer	Meaning
xmin	Minimum value on the x-axis (Real).

Remarks:

1. XMIN applies to all curves.
2. If XMIN is not specified, then the minimum value is set to the lowest value of x.
3. See related commands [XMAX](#), [YMIN](#), and [YMAX](#).

**XPAPER**

## Paper Size in X-Direction

Defines the size of the paper in x-direction.

Format:

XPAPER *xsize*

Example:

XPAP 10 .

Describer	Meaning
<i>xsize</i>	Size of paper in x-direction and in inches (Real; Default = 20.0).

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command [YPAPER](#).

## XTAXIS

### X-Axis Plot Control

XYAXIS applies to upper half frame curves only.

Format:

XTAXIS { YES  
NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).



## XTGRID LINES

### Plot X-Axis Grid Lines

Controls the drawing of the grid lines parallel to the x-axis at the y-axis tic marks on upper half frame curves only.

Format:

XTGRID LINE { YES }  
                  { NO }

Describers	Meaning
YES	Plot the x-axis grid lines.
NO	Do not plot the x-axis grid lines (Default).

Remarks:

1. XTGRID applies to upper half frame curves only.
2. See related command [YTGRID LINES](#)

**XTITLE** X-Axis Title

Defines a character string that will appear along the x-axis.

Format:

```
XTITLE xtit
```

Example:

```
XTIT RIGHT WING CASE 3 - TIME
```

Describer	Meaning
xtit	Any character string (Character; Default = Blank).

Remarks:

1. XTITLE may not be continued to the next command line.
2. XTITLE applies to all curves.

## XYPAPLOT

### Generate X-Y Plots for a Printer

Generates X-Y plots for a printer.

See [XYPLOT](#) for format, describers, and additional remarks.

#### Remarks:

1. The x-axis moves vertically along the page and the y-axis moves horizontally along the page.
2. An asterisk (\*) identifies the points associated with the first curve of a frame, and then for successive curves on a frame the points are designated by symbols O, A, B, C, D, E, F, G, and H.

## XYPEAK

## Print Curve Summary

Print only the summary for all curves. The summary output is titled:

```
"X Y - O U T P U T S U M M A R Y"
```

and is also printed under XYPLOT, XYPUNCH, XYPRINT, and XYPAPLOT. This output contains the maximum and minimum values of y for the range of x.

See [XYPLOT](#) for format, descriptors, and additional remarks.

**XYPLOT****Generate X-Y Plots for a Plotter**

Generates X-Y Plots for a plotter.

**Format:**

```
XYPLOT          yvtype ptype [ i1, i2, i3,... ] /
                id11 (itemu11 [, iteml11] ) , id12 (itemu12 [, iteml12] ) , ... /
                id21 (itemu21 [, iteml21] ) , id22 (itemu22 [, iteml22] ) , ... / ....
```

**Examples:**

BEGIN BULK or OUTPUT(PLOT) commands are shown as a reminder to place X-Y output request packets properly in the Case Control Section; i.e., at the end of the Case Control Section, or just ahead of any structure plot requests.

**Example 1:**

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XYPLOT SDISP/16 (T1)
BEGIN BULK
```

This sequence causes a single whole frame to be plotted for the T1 displacement component of solution set point 16 using the default parameter values. If 16(T1) is not in the solution set, a warning message will be printed and no plot will be made. The plot will be generated for the NASTRAN plotter on file PLT, which must be available.

**Example 2:**

```
OUTPUT (XYOUT)
CSCALE = 1.8
XYPLOT, XYPRINT VELO RESPONSE 1,5 /3 (R1, ) , 5 (, R1)
OUTPUT (PLOT)
```

This sequence causes two frame plots (each consisting of an upper half frame and a lower half frame) to be plotted, one for subcase 1 and one for subcase 5, using the default parameter values. The velocity of the first rotational component of grid point 3 will be plotted on the upper half frame, and that of grid point 5 will be plotted on the lower half frame. Tabular printer output will also be generated for both curves.

**Example 3:**

```
OUTPUT (XYPLOT)
CSCALE = 1.8
YDIVISIONS = 20
XDIVISIONS = 10
SGRID LINES = YES
YGRID LINES = YES
XYPLOT DISP 2,5/10 (T1) , 10 (T3)
BEGIN BULK
```

This sequence causes two whole frame plots to be generated, one for subcase 2 and one for subcase 5. Each plot contains the T1 and T3 displacement component for grid point 10. The default parameters will be modified to include grid lines in both the x-direction and y-direction, with 10 spaces in the x-direction and 20 spaces in the y-direction. The plot will be generated for the NASTRAN plotter on file .plt.

#### Example 4

```
OUTPUT (XYPLOT)
CSCALE = 1.8
XAXIS = YES
YAXIS = YES
XPAPER = 40.
YPAPER = 20.
XYPLOT STRESS 3/ 15 (2) / 21 (7)
OUTPUT (PLOT)
```

This sequence causes two whole frame plots to be generated using the results from subcase 3. The first plot is the response of the axial stress for rod element number 15. The second plot is the response of the major principal stress at Z1 for CTRIA3 element number 21. The default parameters will be modified to include the x-axis and y-axis drawn through the origin. Each plot will be initially scaled to fit on 40 x 20 inch paper. The plots will be generated for the NASPLT postprocessor and NASTRAN file .plt2 which must be defined. NASPLT will redefine the plot to 14 x 7-inch paper (with default options).

#### Example 5

```
OUTPUT (XYPLOT)
CURVELINESYMBOL = -1
XYPLOT XYPAPLOT VG / 1 (G, F) 2 (G, F) 3 (G, F) 4 (G, F)
OUTPUT (PLOT)
```

This sequence is an example of plotting in a flutter analysis for which a split frame plot is made; the upper half is V-g and the lower half is V-f. Data from the first four loops will be plotted. Distinct symbols are used for data from each loop, and no lines are drawn between points (since the flutter analyst must sometimes exercise judgement about which points should be connected). The plots will also be printed in the normal output. These plots will not have all the features of the external plots, but can be very useful in getting a quick picture of the curves.

#### Example 5

```
XTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ SC 200
YTITLE=FLUID MODE PF AT FLUID POINT 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) MODE 200/204
```

#### Example 6

```
YTITLE=EXCITATION FREQUENCY FROM 2.5 TO 250 HERTZ
YTITLE=PSD MPF FOR FLUID GRID 204 FOR NATURAL MODE 2
XYPLOT,XYPEAK FMPF(2) PSDF /204
```

#### Example 7

```
XYPLOT CSIG PSDF / 4 (2003)
```

This is an example of plotting Composite Stress value of stress quantity given by Item Code 3 for layer with ID 2 of element with ID 4.

This can also be equivalently written as (see Remark 9 for explanation)

XYPLOT CSIG PPDF / 4 (-2,3)

Describers	Meaning
yvtype	Type of y-value to be plotted: (Character).
ACCE	Acceleration in the physical set.
BOUT	Slideline contact output.
DISP	Displacement in the physical set.
ELFORCE	Element force.
ENTHALPY	Enthalpy in the physical set.
FLUX	Element heat flux.
FMPF (mode_id or frequency_id)	Fluid mode participation factors. See Remark 8.
GMPF (mode_id or frequency_id, panel_name, panel_grid_id)	Panel grid mode participation factors. See Remark 8.
HDOT	Rate of change of enthalpy in the physical set.
MPCF	Multipoint force of constraint.
LMPF	Load mode participation factors. See Remark 8.
NONLINEAR	Nonlinear applied load.
OLOAD	Applied load.
PMPF (mode_id or frequency_id, panel_name)	Panel mode participation factors. See Remark 8.
PRESSURE	Pressure of fluid-structure body.
SACCE	Acceleration in the solution set.
SDISP	Displacement in the solution set.
SMPF (mode_id or frequency_id)	Structural mode participation factors. See Remark 8.
SPCF	Single-point force of constraint.
STEMP	Temperature in the solution set.

Describers	Meaning
	STRAIN Element strain.
	STRESS Element stress.
	SVELO Velocity in the solution set.
	TEMP Temperature in the physical set.
	VECTOR Displacement in the physical set.
	VELO Velocity in the physical set.
	VG Flutter analysis.
	CSIG Composite Stress
	CEPS Composite Strain
	CFAI Composite Failure Index
	CSRS Composite Strength Ratio
p <sub>type</sub>	Plot type defining the meaning of i <sub>1</sub> , i <sub>2</sub> , ... etc., i <sub>di</sub> , i <sub>temui</sub> , and i <sub>temli</sub> (Character, Default="RESPONSE").
	AUTO Autocorrelation function on whole frame curves only.
	FREQ Frequency—for given excitation frequency plot mode participation versus natural frequency—Oxx2E tables—point plot only. See Remark 8.
	MODE Mode - for given fluid mode plot mode participation versus excitation frequency - Oxx2M tables.
	PSDF Power spectral density function on whole frame curves only.
	RESPONSE Time or frequency in SORT2 format, or grid point identification numbers in SORT1 format (Default) .
	SPECTRAL Response spectrum on whole frame curves only.
i <sub>1</sub> , i <sub>2</sub> ,...	Subcase identification numbers for p <sub>type</sub> =RESPONSE. The list must be specified in ascending order. For p <sub>type</sub> =SPECTRAL, the subcase refers to the RECNO in the DTI,SPSEL Bulk Data entry. The list is ignored for p <sub>type</sub> =AUTO and PSDF (Integer ≥ 0, Default is all subcases).



Describers	Meaning
idij	Element, grid, scalar, or extra point identification number for y-value for frame i. For yvtype=VG, idij refers to the loop count of a flutter analysis (Integer > 0).
itemuij,	<p>Item code for y-value. itemuij is for upper half or whole itemlij curves on frame i, and itemlij is for lower half curves only on frame i. If itemlij is not specified, then whole frame curves will be plotted with itemuij. itemlij is ignored for ptype=“AUTO”, “PSDF”, and “SPECTRAL” (Character or Integer &gt; 0).</p> <p>For elements, the code represents a component of the element stress, strain, or force and is described in <a href="#">Table 7-1</a> and <a href="#">Table 7-5</a> of the Guide. For ptype=“AUTO” and “PSDF”, the complex stress or strain item codes need to be used. Since the output quantities are real, you can use either the real or the imaginary item code. Both will give the same result.</p> <p>For grid points and pty=“RESPONSE”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3, T1RM, T2RM, T3RM, R1RM, R2RM, R3RM, T1IP, T2IP, T3IP, R1IP, R2IP, or R3IP, where Ti stands for the i-th translational component, Ri stands for the i-th rotational component, RM means real or magnitude, and IP means imaginary or phase. For scalar or extra points, or heat transfer analysis, use T1, T1RM, or T1IP. The output format of results in PCH file is dictated by the case control request for DISP, STRESS. For example for SOL111:</p> <pre data-bbox="458 892 928 968">DISP (PLOT, phase, SORT2) = 1 OUTPUT (XYOUT) XYPUNCH DISP RESP /3 (T3RM)</pre> <p>This will punch results in PCH file in Mag/phase format. If the DISP command is missing results will shift to Real/Imaginary the default format for SOL111 displacement.</p> <p>For grid points and ptype=“AUTO” or “PSDF”, the code is one of the mnemonics T1, T2, T3, R1, R2, R3. For scalar or extra points, use T1.</p> <p>For yvtype=VG, itemui and/or itemli can be “F” for frequency or “G” for damping.</p>

#### Remarks:

1. Multiple XYPLOT, XYPUNCH, XYPRINT, XYPEAK, and/or XYPAPLOT commands may be specified in the OUTPUT(XYPLOT) section.
2. Solution set requests are more efficient, because the time-consuming recovery of the dependent displacements can be avoided.
3. The item codes also appear in printed summaries as “CURVE ID” for grid points as well as element data.

4. The information after each slash ( / ) specifies the curve(s) that are to be plotted on the same frame. The descriptor idij identifies the grid point j or element j associated with the frame number i. All plot requests on one command are sorted on idij to improve the efficiency of the plotting process. Symbols are assigned in order by idij.
5. If any of the item codes, itemlij or itemuij, are not specified; e.g., (8) or (5), the corresponding half frame curve is not plotted. If both the comma ( , ) and itemlij not specified; e.g., (8), then whole frame curves will be plotted. Also, for any single frame, the specifications of “(itemuij,itemlij)” must be consistently half frame (upper and/or lower) or whole frame. For example on half frame curves, if iteml11 and the comma is not specified then either iteml12 or itemu12 must not be specified and on whole frame curves, the commas, iteml11, and iteml12 must not be specified. In other words, the curves on each plot frame must be all whole or half (upper and/or lower).
6. The XYPLOT command may be continued on the next line as long as “XYPLOT yvtype ptype [ i1, i2, i3,... ] /” is specified on the first line.
7. Specifying a nonexistent grid point may cause the program to exit in the XYTRAN module and missing plots to occur.
8. mode\_id is used for natural frequency selection of Oxx2m participation versus excitation frequency output.  
frequency\_id is used for excitation frequency selection of Oxx2E participation versus natural frequency output. frequency\_id is an integer value; e.g., (2) would represent the second frequency calculated.
9. For yvtype = CSIG, CEPS, CFAI and CSRS, we can have (itemuij, itemlij) or (itemuij) formats. For the (itemuij, itemlij) format, itemuij will be a negative integer and its absolute value indicates the Ply ID; itemlij indicates the composite stress item code. For the (itemuij) format, itemuij will be an integer  $I = \text{PLYID} * 1000 + \text{ITEM\_CODE}$ .
10. Printout in the f06 file coming from X-Y plot commands is not supported by the POST TOCASE command.

## XYPRINT

### Generate Table of X-Y Pairs for a Printer

Generates tabular printer output of the X-Y pairs.

See [XYPLOT](#) for format, descriptors, and additional remarks.

## XYPUNCH

### Generate Table of X-Y Pairs for the PUNCH File

Generates tabular punch output of the X-Y pairs. Same as XYPRINT except the output is written to the PUNCH file.

See [XYPLOT](#) for format, describers, and additional remarks.

## XVALUE PRINT SKIP

### Print Values on X-Axis Tic Marks

Specifies how often to print the x-values alongside the x-axis tic marks.

Format:

```
XVALUE PRINT SKIP xyps
```

Example:

```
XVAL 5
```

Describer	Meaning
xyps	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ).

Remark:

1. XVALUE applies to all curves.

## YAXIS

### Y Axis Plot Control

Controls the plotting of the y-axis on all curves.

Format:

YAXIS { YES  
NO }

Describers	Meaning
YES	Plot the y-axis.
NO	Do not plot the y-axis (Default).

## YBDIVISIONS

## Tic Spacing on Y-Axis

Specifies spacing of tic marks on the y-axis for lower half frame curves only.

Format:

YBDIVISIONS ybd

Example:

YBDI 10

Describer	Meaning
ybd	Number of spaces between tic marks on y-axis (Integer > 0; Default = 5).

Remarks:

1. YBDIVISIONS applies to lower half frame curves only.
2. YBDIVISIONS is ignored for a logarithmic y-axis.

## YBINTERCEPT

### Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for lower half frame curves only.

Format:

YBINTERCEPT ybi

Example:

YBINT 50

Describer	Meaning
ybi	Location of y-axis on the x-axis (Real; Default = 0.0).

Remark:

1. YBINTERCEPT applies to lower half frame curves only.



## YBGRID LINES

### Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on lower half frame curves only.

Format:

YBGRID LINES { YES }  
                  { NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YBGRID applies to lower half frame curves only.
2. See related command [XBGRID LINES](#)

## YBLOG

## Logarithmic or Linear Y Axis

Selects logarithmic or linear y-axis for lower half frame curves only.

Format:

YBLOG { YES }  
          NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YBLOG applies to lower half frame curves only.
2. See Remark 2 under [XLOG](#).

**YBMAX**

## Maximum Y Axis Value

Specifies the maximum value on the y-axis for lower half frame curves only.

Format:

YBMAX ymax

Example:

YBMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis (Real).

Remarks:

1. YBMAX applies to lower half frame curves only.
2. If YBMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YBMIN](#).

## YBMIN Minimum Y Axis Value

Specifies the minimum value on the y-axis for lower half frame curves only.

Format:

YBMIN ymin

Example:

YBMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YBMIN applies to lower half frame curves only.
2. If YBMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YBMAX](#).

**YBTITLE****Y-Axis Title**

Defines a character string that will appear along the y-axis for lower half frame curves only.

Format:

YBTITLE ytit

Example:

YBTIT RIGHT WING LOADS - CASE 3

Describer	Meaning
ytit	Any character string (Character; Default= Blank).

Remarks:

1. YBTITLE may not be continued to the next command line.
2. YBTITLE applies to lower half frame curves only.

## YBVALUE PRINT SKIP

## Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on lower half frame curves only.

Format:

YBVALUE PRINT SKIP *yvps*

Example:

YBVAL 5

Describer	Meaning
<i>yvps</i>	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ).

Remark:

1. YBVALUE applies to lower half frame curves only.

## YDIVISIONS

## Tic Spacing on Y Axis

Specifies spacing of tic marks on the y-axis for whole frame curves only.

Format:

YDIVISIONS yd

Example:

YDIV 10

Describer	Meaning
yd	Number of spaces between tic marks on y-axis (Integer > 0; Default=5).

Remarks:

1. YDIVISIONS applies to whole frame curves only and to the commands: LEFT TICS, RIGHT TICS, and XINTERCEPT.
2. YDIVISIONS is ignored for a logarithmic y-axis.

## YINTERCEPT

### Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for whole frame curves only.

Format:

YINTERCEPT yi

Example:

YINT 50

Describer	Meaning
yi	Location of y-axis on the x-axis. (Real; Default = 0.0)

Remark:

1. YINTERCEPT applies to lower half frame curves only.



## YGRID LINES

### Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on whole frame curves only.

Format:

YGRID LINES { YES }  
                  { NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YGRID applies to whole frame curves only.
2. See related command [XGRID LINES](#)

## YLOG

## Logarithmic or Linear Y Axis

Selects logarithmic or linear y-axis for whole frame curves only.

Format:

YLOG { YES }  
      { NO }

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YLOG applies to whole frame curves only.
2. See Remark 2 under [XLOG](#).

**YMAX**      Maximum Y Axis Value

Specifies the maximum value on the  $y$ -axis.

Format:

YMAX  $y_{\max}$

Example:

YMAX 100

Describer	Meaning
$y_{\max}$	Maximum value on the $y$ -axis (Real).

Remarks:

1. If YMAX is not specified, then the maximum value is set to the highest value of  $y$ .
2. See related command [YMIN](#).

## YMIN Minimum Y Axis Value

Specifies the minimum value on the y-axis.

Format:

YMIN ymin

Example:

YMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YMIN applies to all curves.
2. If YMIN is not specified, then the minimum value is set to the lowest value of y.
3. See related command [YMAX](#).

**YPAPER**

## Paper Size in Y-Direction

Defines the size of the paper in y-direction.

Format:

YPAPER ysize

Example:

```
YPAP 10
```

Describer	Meaning
ysize	Size of paper in y-direction and in inches (Real; Default = 20.0).

Remarks:

1. The default paper size is 20 by 20 inches.
2. See related command [XPAPER](#).

## YTAXIS

### Y Axis Plot Control

Controls the plotting of the y-axis on upper half frame curves only.

Format:

YTAXIS { YES }  
          { NO }

Describers	Meaning
YES	Plot the x-axis.
NO	Do not plot the x-axis (Default).

Remark:

1. YTAXIS applies to upper half frame curves only.

## YTDIVISIONS

## The Spacing on Y Axis

Specifies spacing of tic marks on the y-axis for upper half frame curves only.

Format:

YTDIVISIONS ytd

Example:

YTDI 10

Describer	Meaning
ytd	Number of spaces between tic marks on y-axis (Integer > 0; Default = 5).

Remarks:

1. YTDIVISIONS applies to upper half frame curves only.
2. YTDIVISIONS is ignored for a logarithmic y-axis.

**YTGRID LINES**

## Plot Y Axis Grid Lines

Controls the drawing of the grid lines parallel to the y-axis at the x-axis tic marks on upper half frame curves only.

Format:

YTGRID LINES { YES }  
                  { NO }

Describers	Meaning
YES	Plot the y-axis grid lines.
NO	Do not plot the y-axis grid lines (Default).

Remarks:

1. YTGRID applies to upper half frame curves only.
2. See related command [XTGRID LINES](#)



**YTINTERCEPT**

## Location of Y Axis on X Axis

Specifies the location of the y-axis on the x-axis for upper half frame curves only.

Format:

YTINTERCEPT yti

Example:

YTINT 50

Describer	Meaning
yti	Location of y-axis on the x-axis (Real; Default= 0.0).

Remark:

1. YTINTERCEPT applies to upper half frame curves only.

**YTITLE**            Y Axis Title

Defines a character string that will appear along the y-axis for whole frame curves only.

Format:

YTITLE ytit

Example:

YTIT RIGHT WING LOADS - CASE 3

Describer	Meaning
ytit	Any character string (Character; Default = Blank).

Remarks:

1. YTITLE may not be continued to the next command line.
2. YTITLE applies to whole frame curves only.

**YTLOG**

## Logarithmic or Linear Y Axis

Selects logarithmic or linear y-axis for upper half frame curves only.

Format:

$$YT(LOG) \left\{ \begin{array}{l} YES \\ NO \end{array} \right\}$$

Describers	Meaning
YES	Plot a logarithmic y-axis.
NO	Plot a linear y-axis (Default).

Remarks:

1. YTLOG applies to upper half frame curves only.
2. See Remark 2 under [XLOG](#).

## YTMAX Maximum Y Axis Value

Specifies the maximum value on the y-axis for upper half frame curves only.

Format:

YTMAX ymax

Example:

YTMAX 100

Describer	Meaning
ymax	Maximum value on the y-axis (Real).

Remarks:

1. YTMAX applies to upper half frame curves only.
2. If YTMAX is not specified, then the maximum value is set to the highest value of y.
3. See related command [YTMIN](#).

**YTMIN**

## Minimum Y Axis Value

Specifies the minimum value on the y-axis for upper half frame curves only.

Format:

YTMIN ymin

Example:

YTMIN 100

Describer	Meaning
ymin	Minimum value on the y-axis (Real).

Remarks:

1. YTMIN applies to upper half frame curves only.
2. If YTMIN is not specified then the minimum value is set to the lowest value of y.
3. See related command [YTMAX](#).

**YTTITLE**

## Y-Axis Title

Defines a character string that will appear along the y-axis for upper half frame curves only.

Format:

YTTITLE ytit

Example:

```
YTTIT RIGHT WING LOADS - CASE 3
```

Describer	Meaning
ytit	Any character string (Character; Default = Blank).

Remarks:

1. YTTITLE may not be continued to the next command line.
2. YTTITLE applies to upper half frame curves only.

## YTVALUE PRINT SKIP

## Print Values on Y Axis Tic Marks

Specifies how often to print the  $y$ -values alongside the  $y$ -axis tic marks applies on upper half frame curves only.

Format:

YTVALUE PRINT SKIP  $yvps$

Example:

YTVAL 5

Describer	Meaning
$yvps$	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ).

Remark:

1. YTVALUE applies to upper half frame curves only.

## YVALUE PRINT SKIP

## Print Values on Y Axis Tic Marks

Specifies how often to print the y-values alongside the y-axis tic marks applies on whole frame curves only.

Format:

YVALUE PRINT SKIP *yvps*

Example:

YVAL 5

Describer	Meaning
<i>yvps</i>	Number of tic marks to be skipped between labeled tic marks with their corresponding values (Integer $\geq 0$ ).

Remark:

1. YVALUE applies to whole frame curves only.



## OUTPUT(POST) Commands

### SURFACE Surface Definition

Defines a surface for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

$$\text{SURFACE id SET sid, FIBRE } \left\{ \begin{array}{l} \text{ALL} \\ \text{Z1} \\ \text{Z2} \\ \text{MID} \end{array} \right\},$$

$$\left[ \text{SYSTEM } \left\{ \begin{array}{l} \text{ELEMENT} \\ \text{BASIC} \\ \text{CORD cid} \end{array} \right\} \right], \left[ \text{AXIS } \left\{ \begin{array}{l} \text{X1} \\ \text{X2} \\ \text{X3} \end{array} \right\} \right], \left[ \text{NORMAL [M]} \left\{ \begin{array}{l} \text{R} \\ \text{X1} \\ \text{X2} \\ \text{X3} \end{array} \right\} \right],$$

$$\left[ \text{TOPOLOGICAL} \right] \left[ \text{GEOMETRIC} \right] \left[ \text{TOLERANCE } \{ \text{THETA } \} \right],$$

$$\left[ \text{BRANCH } \left[ \left[ \left\{ \begin{array}{l} \text{MESSAGE} \\ \text{NOMESSAGE} \end{array} \right\} \right], \left[ \left\{ \begin{array}{l} \text{BREAK} \\ \text{NOBREAK} \end{array} \right\} \right] \right] \right]$$

Example:

```
SURFACE 10 SET 9 NORMAL X3
```

Descriptor	Meaning
id	Surface identification number (Required).
SET	References a SET command that defines the elements in the surface (required). Either form of the SET command may be used.
sid	Set identification number.
FIBRE	Specifies the fiber location at which stresses will be calculated.
ALL	Requests output at all fiber locations; i.e., z=Z1, Z2, and MID.
Z1	Requests output z = Z1 only.
Z2	Requests output z = Z2 only.

Describer	Meaning
MID	Requests output $z = (Z1+Z2)/2$ only.
SYSTEM	Specifies the coordinate system to be used as the output coordinate system.
ELEMENT	Specifies the element coordinate system for output.
CORD cid	Specifies the coordinate system defined on a CORDij Bulk Data entry for output.
BASIC	Specifies the basic coordinate system for output.
AXIS	Specifies the axis of the coordinate system to be used as the x output axis and the local x-axis when geometric interpolation is used.
X1, X2, X3	Specifies the direction of the axis or the normal. X, Y, and Z may be substituted for X1, X2, and X3, respectively.
NORMAL	Specifies the reference direction for positive fiber and shear stress output, but has no effect when ELEMENT is specified.
M	Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter.
R	Specifies the radius vector from the origin of reference coordinate system to the grid point.
TOPOLOGICAL GEOMETRIC	Specifies the method to calculate the average grid point stress or strain. The default is TOPOLOGICAL.
theta	Specifies the interelement slope difference tolerance (in degrees) to detect stress discontinuity between elements (not used with TOPOLOGICAL) (Real; Default = 0.0).
BRANCH	Selects whether multiple element intersections (BREAK/NOBREAK) are to be treated as discontinuities, and if warning messages (MESSAGE/NOMESSAGE) are to be issued.
BREAK NOBREAK	Multiple element intersections are (or are not) to be treated as stress discontinuities.
MESSAGE NOMESSAGE	A warning message will (or will not) be issued when multiple element intersections are encountered.

#### Remarks:

1. SURFACE commands must be specified after OUTPUT(POST).
2. The surface identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON commands. The sid on the surface entry must reference a SET defined after OUTPUT(POST).

3. The surface normal is also used in the definition of the local reference surface for geometric interpolation. Two options are available. In the first option, the radius vector ( $R$ ) from the origin of the reference coordinate system to the grid point is used. In the second option, one axis ( $X1$ ,  $X2$ , or  $X3$ ) of the coordinate system is used. The direction can be reversed using the modification parameter  $M$ . The positive side of an element is defined as a side from which the **NORMAL** direction emerges, rather than the side determined by the connection specified on the element connection entries.
4. When the parameter **ELEMENT** is present, the element stresses or strains are used unmodified (defaults to output stresses in output system). The **CORD** keyword references a **CORDij** Bulk Data entry with coordinate system identification number *cid*.
5. When  $\theta = 0$ , no testing is made. When  $\theta$  is negative, grid point stresses will be calculated for each element connected to an exception point; otherwise, the best estimation of the grid point stress will be output.
6. **BREAK** is the default if  $\theta$  is nonzero.
7. For all elements defined in SET 9 of the preceding example,
  - All fiber locations are output;
  - The basic output system is used;
  - The x-axis is x-axis of the basic system;
  - The surface normal direction point is z-axis of the basic system;
  - The topological interpolation method is used;
  - No tolerance test is made; and
  - No branch test is made.

The example illustrates a good choice for regular two-dimensional problems in the x-y plane.

## VOLUME Volume Definition

Defines a volume for the calculation of grid point stresses, strains, or mesh stress discontinuities.

Format:

VOLUME id SET sid, [PRINCIPAL, DIRECT STRESS], [SYSTEM { ELEMENT  
CORD cid  
BASIC } ]

Example:

VOLUME 21 SET 2

Descriptor	Meaning
id	Volume identification number.
sid	Set identification number of a SET command that defines the elements in the volume. Either form of the SET command may be used. The default is all elements.
PRINCIPAL	Requests principal stresses or strains, direction cosines, mean pressure, and von Mises equivalent stresses or strains to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
DIRECT	Requests direct stress or strains, mean pressure stress, and von Mises equivalent stress to be calculated. If neither PRINCIPAL nor DIRECT is specified, then the default is to output both.
SYSTEM	Used to specify the reference coordinate system used to define the output stress orientation coordinate system.
ELEMENT	Specifies the element coordinate system.
CORD cid	Specifies the coordinate system defined on a CORDij entry.
BASIC	Specifies the basic coordinate system.

Remarks:

- VOLUME commands must be specified after OUTPUT(POST).
- The volume identification number must be referenced on a SET command appearing after OUTPUT(POST). The SET identification number may then be referenced on GPSTRESS, GPSTRAIN, STRFIELD, ELSDCON, and GPSDCON Case Control commands.
- If ELEMENT is specified, element stresses or strains are not transformed.
- In the preceding example, for all elements in SET 2:
  - Both PRINCIPAL and DIRECT stress are output.

- The BASIC output system is used.

## OUTPUT(PLOT) Commands

The PLOT command requests the generation of undeformed, deformed, or contour plots. All other commands specify how the model will be plotted, type of projection, view angles, scales, etc. All commands have default actions if not specified by the user. The FIND command may be used to calculate an optimal SCALE, ORIGIN, and/or VANTAGE POINT to allow the construction of a plot in a user-specified region of the paper or film. All the commands used in the generation of the various plots will be printed out as part of the output, whether they are directly specified, defaulted or established using the FIND command. Initialization of commands to default values occurs only once. Subsequently, these values remain until altered by direct command input. The only exceptions are the view angles, scale factors, vantage points, and the origins. Whenever the plotter or the method of projection is changed, the view angles are reset to the default values, unless they are respecified by the user. In addition, the scale factors, vantage points, and the origin must be respecified by the user.

The commands are listed here in a logical sequence; however, they need not be so specified. Any order may be used, but if a command is specified more than once, the value or choice stated last will be used.

PLOTTER	Selects format of plot file for interpretation by plotter postprocessor.
ORTHOGRAPHIC, etc.	Selects orthographic projection.
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.
AXES	Assigns axes of the basic coordinate system to the observer's coordinate system.
VIEW	Defines the angular relationship between the observer's coordinate system (r, s, and t axes specified on the AXES command) and the basic coordinate system.
MAXIMUM DEFORM	Defines the magnification of the maximum displacement.
SCALE	Defines reduction, as a scale factor, of the model's dimensions so that the model fits on a plot frame.
DISTORTION	Specifies the distortion scale factors of axes in the basic coordinate system.
CSCALE	Defines scale factor for characters in the plot frame.
ORIGIN	Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.
VANTAGE POINT	Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.
PROJECTION	Defines the separation, along the r-axis, between the observer and the projection plane if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.
OCULAR SEPARATION	Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.
CAMERA	Specifies microfilm plotter options.

PAPER SIZE	Defines the size and type of the paper.
PEN	Generates a message on the printed output to inform the plotter operator as to what size and color pen point to mount in the various pen holders.
PTITLE	Defines a character string that will appear at the top of the plot frame on the line below the sequence number.
SET	Defines a set of elements and/or grid points to be plotted.
FIND	Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.
CONTOUR	Specifies contour plot options for stress, displacement, or temperature.
PLOT	Generates an undeformed plot or a deformed plot per subcase, mode number, frequency, or time step. A contour plot may also be requested with an undeformed or deformed plot.

## AXES

### Orientation of Observer's Coordinate System

Assigns axes of the basic coordinate system to the observer's coordinate system.

Format:

AXES r s t  $\begin{bmatrix} \text{SYMMETRIC} \\ \text{ANTISYMMETRIC} \end{bmatrix}$

Example 1: View toward negative x-axis of model.

AXES MX, Y, MZ

Example 2: Mirror image of model.

AXES Y X Z

Describers	Meaning
r,s,t	Assigns axes of basic coordinate system to axes of observer's coordinate system (Default=X, Y, Z). X X-direction of basic coordinate system (Default for r). Y Y-direction of basic coordinate system (Default for s). Z Z-direction of basic coordinate system (Default for t). MX Negative X-direction of basic coordinate system. MY Negative Y-direction of basic coordinate system. MZ Negative Z-direction of basic coordinate system.
SYMMETRIC	Specifies a symmetric orientation of the view. See Remark 5. (Default).
ANTISYMMETRIC	Specifies an antisymmetric orientation of the view. See Remark 5.

Remarks:

1. If no AXES command is specified, then AXES X, Y, Z is the default.
2. The direction of view is in the negative r-direction; i.e., the projection plane is parallel to the s-t plane.
3. The VIEW command depends on the AXES command specification and defines the angular relationship between observer's coordinate system and the basic coordinate system.
4. The AXES command can be used to preposition the object in 90° increments in such a manner that only rotations less than 90° are required by the VIEW command to obtain the desired orientation. MX, MY, MZ can be used to define left-handed coordinate systems. Note that the default system is right-handed.



5. An undeformed or deformed plot of the symmetric portion of an object can be obtained by reversing the sign of the axis that is normal to the plane of symmetry. In the case of multiple planes of symmetry, the signs of all associated planes should be reversed. The ANTISYMMETRIC option should be specified when a symmetric model is loaded in an unsymmetric manner. This will cause the deformations to be plotted antisymmetrically with respect to the specified plane or planes. Since the AXES command applies to all parts (SETs) of a single frame, symmetric and antisymmetric combinations cannot be made with this command (see the symmetry option on the [PLOT, 766](#) command in this section).
6. To avoid a mirror image, ensure that the r, s, and t axes obey the right-hand rule.

## CAMERA

## Microfilm Plotter Options

Specifies microfilm plotter options.

Format:

```

CAMERA [ PAPER
        FILM BLANK FRAME n
        BOTH ]

```

Example:

```
CAMERA FILM
```

Describers	Meaning
FILM	Requests 35 mm or 16 mm film and positive or negative images.
PAPER	Requests positive prints.
BOTH	Requests positive prints and 35 mm or 16 mm film.

Remarks:

1. If the CAMERA command is not specified, then CAMERA PAPER BLANK FRAMES 0 is assumed.
2. If FILM or BOTH is specified, then these options must be communicated to the plotter operator through normal means of communications at the installation.
3. If FILM or BOTH are specified and if n is greater than 0 then n blank frames will be inserted between plots. The plotter must be operated in the manual mode in order to have blank frames inserted between positive prints. If blank frames are desired only on film, and not on paper, the plotter must be operated in the automatic mode.

## CONTOUR

### Contour Plot Options

Specifies contour plot options for stress, displacement, or temperature.

Format:

$$\text{CONTOUR component} \left\{ \begin{array}{l} \text{EVEN } n \\ \text{LIST } a, b, \dots \end{array} \right\} \left\{ \begin{array}{l} Z1 \\ Z2 \\ \text{MAX} \\ \text{MID} \end{array} \right\} \left\{ \begin{array}{l} \text{COMMON} \\ \text{LOCAL} \end{array} \right\}$$

Example:

CONTOUR MAGNIT LIST 2., 4., 6., 8., 10.

Describers	Meaning
component	Name of stress, displacement or temperature component (Character, Default="MAJPRIN").
MAJPRIN	Major principal stress. Not available for nonlinear elements (Default).
MINPRIN	Minor principal stress. Not available for nonlinear elements.
EQUIVAL	von Mises stress. When STRESS(MAXS) is requested in the Case Control Section, von Mises stress is used for plotting. For nonlinear analysis, Mohr-Coulomb or Drucker-Prager stress may also be plotted in conjunction with the MATS1 command.
XNORMAL	X component of normal stress.
YNORMAL	Y component of normal stress.
ZNORMAL	Z component of normal stress.
XYSHEAR	XY component of shear stress.
XZSHEAR	XZ component of shear stress.
YZSHEAR	YZ component of shear stress.
XDISP	T1 component of displacement in global coordinate system.
YDISP	T2 component of displacement in global coordinate system.
ZDISP	T3 component of displacement in global coordinate system.
MAGNIT	Magnitude of displacement or temperature.
EVEN n	Number of contours ( $50 \geq \text{Integer} > 0$ , Default is EVEN 10).
LIST a, b, ...	List of stresses, displacements or temperatures which define the contours (Real).

Describers	Meaning
Z1	Stresses at Z1 from neutral plane (Default).
Z2	Stresses at Z2 from neutral plane.
MAX	Maximum of stress at Z1 and Z2.
MID	Average of stress (membrane stress) at Z1 and Z2.
COMMON	Plot stress contours in basic coordinate system (Default).
LOCAL	Plot stress contours in element coordinate system. This is the coordinate system used in printed output.

#### Remarks:

1. The CONTOUR command should be specified immediately before its associated PLOT command.
2. A STRESS command must appear in the Case Control Section for all elements included in a CONTOUR request. If printed output is not desired, then STRESS(PLOT)=sid should be specified.
3. In linear analysis, stress contour plots are available for the following elements: CTRIA3, CQUAD4, CSHEAR, and CTRIAX6. In nonlinear analysis, stress contour plots are available for CQUAD4 and CTRIA3 elements. The Bulk Data element connection entries for all elements must list the grid points in either clockwise or counterclockwise order. Mixing the order will result in meaningless or confusing plots.
4. When selecting contour options, note that
  - MAJPRIN, MINPRIN, EQUIVAL are the same in COMMON and LOCAL.
  - ZNORMAL, XZSHEAR, YZSHEAR, if selected in LOCAL, will be changed to COMMON.
  - CSHEAR elements only have the MAXSHEAR value.
5. The CTRIAX6 element stress contour plots are different in that they must be selected as COMMON. Also, the following equivalences apply:

```

XNORMAL is radial
YNORMAL is azimuthal
ZNORMAL is axial
XYSHEAR is shear
XZSHEAR is maximum principal
YZSHEAR is von Mises
EQUIVAL is octahedra

```

**CSCALE**

## Character Scale Factor

Defines scale factor for characters in the plot frame.

Format:

CSCALE cs

Example:

CSCA 2.0

Describer	Meaning
cs	Scale factor applied to characters in the plot frame (Default = .5).

Remarks:

1. CSCALE is used to control the spacing of characters when plots are made with the NASTRAN plotter and postprocessed with the MSC/NASPLOT routine. For example, if the SCALE FACTOR on the NASPLOT data command is 2.0, a value for cs of 0.5 will result in characters of default size (.07 inches) at the regular spacing. A value of 1.8 produces good spacing when using the postprocessing plotter programs NASTPLT, TEKPLT, and NEUPS. On the other hand, to double the size of both the plot and the characters, the SCALE FACTOR and the CSCALE FACTOR on the NASPLOT data command should both be set equal to 2.0.
2. The CSCALE command must immediately precede the PLOTTER command. If a second CSCALE command is specified, then a second PLOTTER command must also be specified.

## DISTORTION

### Distortion Scale Factors

Specifies the distortion scale factors of the axes in the basic coordinate system.

Format:

```
DISTORTION dx dy dz
```

Example:

```
DIST 0.5 1.0 1.0
```

Describers	Meaning
dx	Distortion scale factor of the basic coordinate system's x-axis (Default=1.0).
dy	Distortion scale factor of the basic coordinate system's y-axis (Default=1.0).
dz	Distortion scale factor of the basic coordinate system's z-axis (Default=1.0).

Remarks:

1. If no DISTORTION command is specified, then no distortion is applied.
2. If DISTORTION is specified, then all three values for dx, dy, and dz must be specified even though one or two will use the default.
3. The distortion factors are applied prior to any other scaling commands: SCALE, MAXIMUM DEFORMATION, CSCALE, etc.

**FIND****Automatic Calculation of Scale, Origin, and Vantage Point**

Requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN i, and/or VANTAGE POINT commands.

Format:

```
FIND [ SCALE ORIGIN oid VANTAGE POINT SET setid ,
      REGION { le be re te } ]
```

Example:

```
FIND SCALE ORIGIN 5 SET 2
```

Describers	Meaning
oid	Origin identification number (Integer>0).
setid	Set identification number etc. (Integer>0).
le	Fractional distance of left edge of plot region from the lower left corner of the image area (Real, Default=0.0).
be	Fractional distance of bottom edge of plot region from the lower left corner of the image area (Real, Default=0.0).
re	Fractional distance of right edge of plot region from the lower left corner of the image area (Real, Default=1.0).
te	Fractional distance of top edge of plot region from the lower left corner of the image area (Real, Default=1.0).

Remarks:

1. The FIND command is recommended over the specification of SCALE, ORIGIN, and VANTAGE POINT commands, and should be specified prior to its associated PLOT or CONTOUR command.
2. The FIND command requests the plotter to optimally compute any of the parameters that can be specified on the SCALE, ORIGIN, and/or VANTAGE POINT commands and based on the specification of the
  - PLOTTER command;
  - PROJECTION PLANE command;
  - SET and REGION specifications on the FIND command;
  - VIEW and/or AXES commands;
  - MAXIMUM DEFORMATION command; and
  - PAPER SIZE command. All of these commands must precede the associated FIND command.
3. The FIND command can be used to compute any or all of SCALE, ORIGIN, or VANTAGE POINT as long as they have not been previously specified.
4. If SET is not specified, then the first defined SET will be used.

5. If no options are specified on the FIND command, a SCALE and VANTAGE POINT are selected and an ORIGIN is located, using the first defined SET, so that the plotter object is located within the image area.
6. The plot region is defined as some fraction of the image area (image area = 0.0, 0.0, 1.0, 1.0 and first quadrant = 0.5, 0.5, 1.0, 1.0). The image area is located inside the margins on the paper.



**MAXIMUM DEFORM****Length of Maximum Displacement**

Defines the magnification of the maximum displacement. All other displacements are scaled accordingly.

Format:

MAXIMUM DEFORMATION d

Example:

Magnify the displacements such that the maximum displacement is equal to two units of length of the model.

MAXI DEFO 2.

Describer	Meaning
d	Specifies the length, in units of the model and not of the plot frame, to which the maximum displacement is scaled. (Default=5% of the largest dimension of the model represented by the elements in the SET specification on the PLOT command.)

Remarks:

1. If no MAXIMUM DEFORMATION command is specified, then the previously described default is assumed.
2. If you wish the FIND command to use the d value, a MAXIMUM DEFORMATION command should precede the FIND command.
3. If you wish the plot deformation scaling to be different then the FIND scaling, a different MAXIMUM DEFORMATION command can appear first before the PLOT command.
4. For nonlinear plotting, MAXIMUM DEFORMATION d and the MAXIMUM DEFORMATION field on the PLOT command should have the same value.

## OCULAR SEPARATION

## Separation of the Vantage Points

Defines the separation of the left and right eye vantage points along the s-axis for stereoscopic projections.

Format:

OCULAR SEPARATION os

Example:

OCULAR SEPARATION 2.0

Describer	Meaning
os	Separation, in inches, of the two vantage points along the s-axis (Real, Default=2.756 inches).

Remark:

1. The default value is the separation used in the standard stereoscopic cameras and viewers (70 mm). The default value is recommended.

**ORIGIN****Origin of Plot Frame**

Defines the origin of the plot frame with respect to the origin of the (r, s, t) coordinate system defined on the AXES command.

Format:

ORIGIN oid u v

Example:

ORIG 3 -1. -2.

Describers	Meaning
oid	Origin identification number which may be specified after the ORIGIN describer on the PLOT command (Integer > 0).
u	Horizontal displacement of plot frame origin from the rst origin (Real, Default = 0.0).
v	Vertical displacement of paper origin from the rst origin (Real, Default = 0.0).

Remarks:

1. In the transformation performed for any of the three projections, the origins of both the basic coordinate system and the observer's coordinate system are coincident. The ORIGIN command may be used to locate the plot frame origin (lower left hand corner) from the rst origin. The units are inches, and are not subject to the scaling of the plotted object.
2. The ORIGIN command is not recommended for general use. See the [FIND](#) command to have the origin optimally located so as to place the plotted object in the center of the plot frame.
3. Ten origins may be specified at one time. However, any one can be redefined at any time. An eleventh origin is also provided if more than ten origins are erroneously defined; i.e., only the last of these surplus origins will be retained.
4. If a projection; e.g., ORTHOGRAPHIC, STEREOSCOPIIC, or PERSPECTIVE, is changed in the plot packet, or the PLOTTER command is changed, then all previously defined origins are deleted.

## ORTHOGRAPHIC, etc.

## Type of Projection

Selects type of projection.

Format:

```
[
  ORTHOGRAPHIC
  PERSPECTIVE
  STEROSCOPIC ]
```

Describers	Meaning
ORTHOGRAPHIC	Selects orthographic projection (Default).
PERSPECTIVE	Selects perspective projection.
STEREOSCOPIC	Selects stereoscopic projection.

Remark:

1. If none of the preceding projections are specified, then ORTHOGRAPHIC is used.

## PAPER SIZE

## Paper Dimensions

Defines the size and type of the paper.

Format:

```
PAPER SIZE h X or BY v [TYPE ptype]
```

Example:

```
PAPER SIZE 10. BY 10.
```

Describers	Meaning
h	Horizontal size of paper in inches (Real, Default = 20.0).
v	Vertical size of paper in inches (Real, Default= 20.0).
ptype	Paper type (Character, Default = "VELLUM").

Remarks:

1. The default paper size for the PLOTTER NAST is 20 by 20 inches which is converted to a 7 by 7 inch plot frame by the NASPLOT postprocessor.
2. PAPER SIZE can be specified along with the NASPLOT postprocessor to create rectangular plots. For example, the command will result in a 14 by 7 inch plot frame if the default value of 1.0 is used for the SCALE FACTOR on the NASPLOT command. The SCALE FACTOR on the NASPLOT data command can be used to make larger plots having the shape defined with PAPER SIZE.
3. PAPER SIZE also affects the raster count for the NASTRAN plotter. The default raster count is 1000 for a paper size of 20 by 20. Doubling the paper size to 40 by 40 will double the raster count to 2000.

## PEN Pen Color and Size Assignments

Generates a message on the printed output which may be used to inform the plotter operator as to what size and color pen point to mount in the various pen holders.

Format:

```
PEN pn [ COLOR cname ]
```

Example:

```
PEN 2 COLOR RED
```

Describers	Meaning
pn	Pen identification number (Integer > 0).
COLOR	Flag indicating the next word is a color name.
cname	Pen color (Character).

Remarks:

1. The actual number of pens available will depend on the plotter hardware configuration at each installation.
2. The PEN command does not control the pen used in generating the plot. See the PEN describer on the [PLOT](#) command.
3. The PEN command is optional and is not appropriate for microfilm plotters.

## PERSPECTIVE

Selects Perspective Projection

See the description of the [ORTHOGRAPHIC](#), etc..

## PLOT Undeformed or Deformed Plot Request

Generates an undeformed plot of the model or a deformed plot for a subcase, mode number or time step.

Format:

PLOT [analysis][dtype][CONTOUR][i1, i2 THRU i3, i4, etc.]  $\left\{ \begin{array}{l} \text{RANGE f1, f2} \\ \text{TIME t1, t2} \end{array} \right\}$ ,

$\left\{ \begin{array}{l} \text{PHASE LAG } \phi \\ \text{MAGNITUDE} \end{array} \right\}$  [MAXIMUM DEFORMATION d] ,

[SET sid1] [ORIGIN oid1]  $\left\{ \begin{array}{l} \text{SYMMETRY} \\ \text{ANTISYMMETRY} \end{array} \right\}_w \left[ \left\{ \begin{array}{l} \text{PEN} \\ \text{DENSITY} \end{array} \right\}_p \right]$  [SYMBOLS m[,n]] ,

[LABEL label]  $\left[ \left[ \begin{array}{l} \text{SHAPE} \\ \text{OUTLINE} \end{array} \right] \right]$  [VECTOR v] , [PRINT] ,

[SHRINK t, o] [NORMALS] ,  
[SET sid2] [ORIGIN oid2] etc.

Examples:

See after Remarks Section.

Describers	Meaning
analysis	Type of analysis results (Character, default results in an undeformed plot or undeformed underlay for contour plots). STATIC Plot static deformations. MODAL Plot mode shapes or eigenvectors. CMODAL Plot complex mode shapes or eigenvectors. TRANSIENT Plot transient solutions. FREQUENCY Plot frequency solutions. SENOMOVE Plot undeformed superelements in their original position; i.e., ignore SELOC Bulk Data entry.
dtype	Specifies plot quantity (Character, Default = "DEFORMATION").



Describers	Meaning
DEFORMATI ON	Plot displacements or temperatures in the Z direction (Default).
VELOCITY	Plot velocities.
ACCELERATI ON	Plot accelerations.
CONTOUR	Request for contour plot.
i1, i2,...	Subcase identification numbers. See SHAPE and VECTOR for use of "0" (underlay) command. See Remark 3. (Integer $\geq 0$ , Default is all subcases).
RANGE f1 f2	Specifies range of natural frequencies, eigenvalues, excitation
TIME t1,t2	frequencies, time steps, or load factors. Used to minimize the amount of plotted data. See Remark 4. (Real).
PHASE LAG $\phi$	Specified phase lag, in degrees, for plotting complex quantities. See Remark 5. (Real, Default = 0.0).
MAGNITUDE	Requests magnitude of complex quantities.
MAXIMUM DEFORMATION d	Specifies the magnification of the maximum displacement. See Remark 6. (Real).
SET sid	Set identification number which defines the set of elements or grid points to be plotted (Default is first SET command).
ORIGIN oid	Origin identification number (Default is first origin defined by the ORIGIN or FIND command).
SYMMETRY w ANTISYMMETRY w	Request plot of the symmetric portion of the symmetrically or antisymmetrically loaded model. This symmetric portion will be located in the space adjacent to the region originally defined by ORIGIN oid, and will appear as a reflection of the antisymmetrically deformed model about the plane whose normal is oriented parallel to the coordinate direction w. See Remark 7. (Default is no action).
PEN p	Specifies pen number that is used to generate the plot (Integer $> 0$ ).
DENSITY d	Specifies line density scale factor for film plotters which is d times heavier than a line density of 1 (Integer $> 0$ ).

Describers	Meaning
------------	---------

SYMBOLS m[,n] All the grid points associated with the specified set will have symbol m overprinted with symbol n printed at its location. If n is not specified, only symbol m will be printed. See Remark 8.

m or n	Symbol
0	none
1	X
2	*
3	+
4	-
5	.
6	×
7	[ ]
8	< >
9	/ \

LABEL label

Specifies labeling options at elements and grid points:

- GRID All the grid points included in the specified set have their identification number printed to the right of the undeformed or deformed location (undeformed location in the case of superimposed plots) (Default).
- ELEMENTS All the elements included in the specified set are identified by the element identification number and type at the center of each element (undeformed location in the case of superimposed plots). See Remarks 11. and 12.
- BOTH Both GRID and ELEMENT options.
- GSPC Label those degrees-of-freedom that are constrained to zero through permanent single point constraints on GRID and GRDSET Bulk Data entries, or are constrained through SPC and SPC1 Bulk Data entries. The label consists of the grid point ID number and the constrained degrees-of-freedom.
- EPID Label elements with their respective property identification (PID) numbers. The label consists of the standard element labels and element PID.

Describers	Meaning												
SHAPE	All the elements included in the specified set are shown by connecting the associated grid points in a predetermined manner. See Remark 9.												
OUTLINE	Only the outline of all the elements in the specified set is shown. Elements not supported by contour plots are ignored. Outlines are always drawn with PEN 1. See Remark 9.												
VECTOR $v$	A line will be plotted at the grid points of the set representing by length and direction the deformation of the point. See Remark 10. Possible values of $v$ are: <table border="0" style="margin-left: 20px;"> <tr> <td>X, Y, or Z</td> <td>Requesting individual components.</td> </tr> <tr> <td>XY, XZ, or YZ</td> <td>Requesting two specified components.</td> </tr> <tr> <td>XYZ</td> <td>Requesting all three components.</td> </tr> <tr> <td>RXY, RXZ, or RYZ</td> <td>Requesting vector sum of two components.</td> </tr> <tr> <td>R</td> <td>Requesting total vector deformation.</td> </tr> <tr> <td>N</td> <td>Used with any of the preceding combinations to request no underlay shape be drawn.</td> </tr> </table>	X, Y, or Z	Requesting individual components.	XY, XZ, or YZ	Requesting two specified components.	XYZ	Requesting all three components.	RXY, RXZ, or RYZ	Requesting vector sum of two components.	R	Requesting total vector deformation.	N	Used with any of the preceding combinations to request no underlay shape be drawn.
X, Y, or Z	Requesting individual components.												
XY, XZ, or YZ	Requesting two specified components.												
XYZ	Requesting all three components.												
RXY, RXZ, or RYZ	Requesting vector sum of two components.												
R	Requesting total vector deformation.												
N	Used with any of the preceding combinations to request no underlay shape be drawn.												
PRINT	List of the average stresses at the interior grid points in the set will be printed for contour stress plots.												
SHRINK $t, o$	<p><math>t</math> is the ratio of the reduction to the original dimensions of all two-dimensional elements except the CQUAD8 and CTRIA6 (<math>0.0 \leq t \leq 1.0</math>) (Real, Default = 0.1 which results in a 10% reduction).</p> <p><math>o</math> is the ratio of the reduction in length to the original length for one-dimensional elements. There is no default value for <math>o</math>. <math>t</math> must be specified to shrink one-dimensional elements.</p>												
NORMALS	Plot vector normal to CHBDYP and CHBDYG elements.												

#### Remarks:

1. If PLOT is specified with no describers, then a picture of the undeformed model will be prepared using the first defined set and the first defined origin.
2. Describers analysis through PHASE LAG must be specified in the order shown above.
3. The following should be noted when using subcase numbers for plotting eigenvectors.
  - a. If subcase numbers are specified, then the convention for displacement vectors is that the list of subcases must refer to subcase IDs whenever the number of modes recovered is equal to or less than the number of subcases defined. If the number of modes recovered is more than the subcases defined, the plot request for those modes associated with the subcases must refer to subcase IDs. After the mode associated with the last defined subcase, higher modes will be identified by incrementing the last defined subcase ID by one for each such higher mode.

- b. For the display of element quantities in contour plots, the automatic incrementing beyond the last defined subcase does not occur. All subcase numbers to be plotted must be defined. A MODES command in the Case Control Section may be used for this purpose.
  - c. In problems using cyclic symmetry solution sequences, the plot requests for segments of the model must refer to the coded subcase identification numbers (see [Theory](#) in the *MSC Nastran Reference Guide*). All eigenvectors recovered for the segment will be plotted. The RANGE option can be used to select a subset of all eigenvectors for plotting without use of coded subcase IDs.
  - d. RANGE does not require the use of subcase numbers.
4. RANGE specifies the range of values using requested subcases for which plots will be prepared. If only one number is input, it is used as the lower bound and the upper bound is set to the highest value computed. Unless otherwise noted, the default range is all values computed.
- a. In real eigenvalue analysis, the values are natural frequencies, in units of cycles per unit time.
  - b. In buckling analysis, the values are eigenvalues.
  - c. In frequency response, the values are excitation frequencies in units of cycles per unit time.
  - d. In transient response, the values are in units of time.
  - e. In static nonlinear analysis (SOLs 106 and 153), the values are load factors. The default range is the highest load factor of each subcase.
  - f. In transient nonlinear analysis (SOLs 129 and 159), the values are in units of time. The default range is the last time step for each subcase.
5. PHASE LAG  $\phi$  is used in the equation:

$$u_R \cos \phi - u_I \sin \phi$$

where  $u_R$  and  $u_I$  are the real and imaginary parts of the response quantity, respectively. The printed output for magnitude and phase uses the convention of a phase lead.

6. MAX DEFO is not recommended for general use. Each subcase is separately scaled according to its own maximum if this item is absent. If d is omitted, the set will be scaled to the maximum within the set being plotted.
7. w specifies the basic coordinates X, Y, or Z, or any combination thereof. This option allows the plotting of symmetric and/or antisymmetric combinations, provided that an origin is selected for the portion of the model defined in the Bulk Data Section that allows sufficient room for the complete plot. This does not permit the combination of symmetric and antisymmetric subcases, as each plot must represent a single subcase. In the case of a double reflection, the figure will appear as one reflected about the plane whose normal is parallel to the first of the coordinates w, followed by a reflection about the plane whose normal is oriented parallel to the second of the coordinates w. This capability is primarily used in the plotting of models that are loaded in a symmetric or an antisymmetric manner. The plane of symmetry must be one of the basic coordinate planes.
8. Grid points excluded from the set will not have a symbol. Grid points in an undeformed underlay will be identified with symbol 2.

9. In order to get a deformed shape, either SHAPE or OUTLINE must be present in the PLOT command. Both deformed and undeformed shapes or outlines may be specified. All the deformed shapes relating to the subcases listed may be underlaid on each of their plots by including "0" with the subcase string on the PLOT command. The undeformed plot will be drawn using PEN 1 or DENSITY 1 and symbol 2 (if SYMBOLS is specified).
10. All plots requesting the VECTOR option will have an underlay generated of the undeformed shape using the same sets, PEN 1 or DENSITY 1, and symbol 2 (if SYMBOLS is specified). If SHAPE and VECTOR are specified, the underlay will depend on whether "0" is used along with the subcases with DEFORMATION. It will be the deformed shape when not used, and will be both deformed and undeformed shapes when it is used. The part of the vector at the grid point will be the tail when the underlay is undeformed, and the head when it is deformed. If v="N" then no shape will be drawn, but other options such as SYMBOLS will still be valid.
11. Element type labels are: (Plot labels QH and TH indicate hyperelastic elements)

Element Type	Plot Label	Element Type	Plot Label
CAERO1	AE	PLOTEL	PL
CAXIF2	A2	CQUAD	QH
CAXIF3	A3	CQUAD4	Q4 or QH
CAXIF4	A4	CQUAD8	Q8 or QH
CBAR	BR	CQUADR	QR
CBEAM	BM	CQUADX	QH
CBEND	BD	CROD	RD
CBUSH	BU	CSHEAR	SH
CONEAX	CN	CSLOT3	S3
CONROD	CR	CSLOT4	S4
CDUMI	Di	CTETRA	TE
CFLUID2	F2	CTRIAX6	D1
CFLUID3	F3	CTRIA3	T3 or TH
CFLUID4	F4	CTRIA6	T6 or TH
CHBDYG	HB	CTRIAR	TR
CHBDYP	HB	CTRIAX	TH
CHEXA	HA	CTUBE	TU
CPENTA	HA	CVISC	VS

12. The heat transfer boundary condition elements CHBDYG and CHBDYP can be plotted for undeformed plots. There are several types of CHBDYi elements, as follows:

Type	No. of Primary Grid Points	Normals Available
POINT	1	yes
LINE	2	yes
AREA3	3	yes
AREA4	4	yes
REV	2	no
ELCYL	2	no
TUBE	2	yes
FTUBE	2	yes
AREA6	6	yes
AREA6	8	yes

The secondary grid points are used for ambient conditions and are ignored by the plotter. Type POINT must have a nonzero associated area (see AF on the associated PHBDY entry) and a defined normal direction (see V1, V2, V3 on the CHBDYP entry) to be plotted. It is plotted as a hexagon with approximately the correct area. Type LINE must have a nonzero width (see AF on the associated PHBDY entry) and a defined normal in order to plot.

13. To assign PLOT command to superelements it requires an SEUPPLOT or a SEPLOT command.

#### Examples:

- Undeformed SHAPE using first defined SET, first defined ORIGIN and PEN 1 (or DENSITY 1).  
PLOT
- Undeformed SHAPE using SET 3, ORIGIN 4, PEN 2 (or DENSITY 2) with each grid point of the set having a + placed at its location, and its identification number printed adjacent to it.  
PLOT SET 3 ORIGIN 4 PEN 2 SHAPE SYMBOLS 3 LABEL
- Modal deformations as defined in subcase 5 using first defined SET, first defined ORIGIN, and PEN 1 (or DENSITY 1).  
PLOT MODAL DEFORMATION 5 SHAPE
- STATIC deformations as defined in subcases 3, 4, 5, and 8 deformed SHAPE; drawn with PEN 4, using first defined SET and ORIGIN, underlaid with undeformed SHAPE drawn with PEN 1. This command will cause four plots to be generated.  
PLOT STATIC DEFORMATION 0, 3 THRU 5, 8 PEN 4, SHAPE

5. Deformations as defined in subcases 1, 2, 3, 4, and 5 undeformed underlay with PEN 1, consisting of SET 2 at ORIGIN 3, SET 2 at ORIGIN 4 (with a < placed at each grid point location), and SET 35 at ORIGIN 4. Deformed data as follows: SHAPE using SET 2 at ORIGIN 3 (PEN 3) and SET 35 at ORIGIN 4 (PEN 4); 3 VECTORS (X, Y and Z) drawn at each grid point of SET 2 at ORIGIN 4 (PEN 4) (less any excluded grid points), with o placed at the end of each vector.

```
PLOT STATIC DEFORMATION 0 THRU 5,
  SET 2 ORIGIN 3 PEN 3 SHAPE,
  SET 2 ORIGIN 4 PEN 4 VECTORS XYZ SYMBOLS 0,
  SET 35 SHAPE
```

6. Static deformations as defined in subcases 3 and 4, both halves of a problem solved by symmetry using the X-Y principal plane as the plane of symmetry. SET 1 at ORIGIN 2 and SET 2 at ORIGIN 3, with the deformed shape plotted using DENSITY 3 and the undeformed model plotted using DENSITY 1. The deformations of the “opposite” half will be plotted to correspond to symmetric loading. This command will cause two plots to be generated.

```
PLOT STATIC DEFORMATIONS 0, 3, 4,
  SET 1 ORIGIN 2 DENSITY 3 SHAPE,
  SET 1 SYMMETRY Z SHAPE,
  SET 2 ORIGIN 3 SHAPE,
  SET 2 SYMMETRY Z SHAPE
```

7. Transient deformations as defined in subcase 1 for time = 0.1 to time = 0.2, using SET 1 at ORIGIN 1. The undeformed SHAPE using PEN or DENSITY 1 with an \* at each grid point location will be drawn as an underlay for the resultant deformation vectors using PEN or DENSITY 2 with an < typed at the end of each vector drawn. In addition, a plotted value of 2.0 will be used for the single maximum deformation occurring on any of the plots produced. All other deformations on all other plots will be scaled relative to this single maximum deformation. This command will cause a plot to be generated for each output time step which lies between 0.1 and 0.2.

```
PLOT TRANSIENT DEFORMATION, TIME 0.1, 0.2,
  MAXIMUM DEFORMATION 2.0, SET 1, ORIGIN 1, PEN 2, SYMBOLS 2,
  VECTOR R
```

8. Contour plot of x-component of normal stress for elements in SET 2 in basic coordinate system at a distance Z1 from neutral plane with 10 contour lines, an outline of elements in SET 2, and using

```
ORIGIN 4.
CONTOUR XNORMAL
PLOT CONTOUR, SET 2, ORIGIN 4, OUTLINE
```

Contour plot of magnitude of displacements at grid points associated with elements in SET 5 with 5 contours having values of 2., 4., 6., 8., 10., and an outline of the elements in SET 5 using

```
ORIGIN 4.
CONTOUR MAGNIT, LIST 2., 4., 6., 8., 10.
PLOT CONTOUR, SET 5, OUTLINE
```

9. Plot the imaginary part of the complex eigenvector in SET 1.

```
PLOT CMODAL DEFORMATION PHASE LAG 90. SET 1 VECTOR R
```

## PLOTTER

### Plot File Format

Selects format of plot file for interpretation by plotter postprocessor.

Format:

$$\text{PLOTTER} \left\{ \begin{array}{c} \text{NAST} \\ \text{SC} \end{array} \right\}$$

Example:

PLOTTER NAST

Describers	Meaning
NAST	Specifies format suitable for Postscript plotters (Default).
SC	Specifies Stromberg-Carlson microfilm plotter format.

Remark:

1. If no PLOTTER command is specified, then PLOTTER NAST is the default.
2. If “PLOTTER NAST” is specified (or taken by default) and if the format of the PLOT file has been changed to formatted through the use of an ASSIGN statement, plot data will be generated in PostScript mode directly instead of being generated in binary mode. PostScript plot control information may be specified using the SYS= describer on the ASSIGN statement or by using the SYSFIELD= command-line keyword, specifying the control information using the PLOT (keyword=value, . . .) options. The valid keywords are the same as those that can be specified for the PLOTPS or MSCPLOTPS utility programs except that the “begin”, “debug”, “dump”, “end”, “format” and “output” keywords are not allowed.



## PROJECTION

### Separation Between Projection Plane and Observer

Defines the separation along the r-axis, and between the observer and the projection plane, if not already specified on the VANTAGE POINT command. Used by stereoscopic projections only.

Format:

PROJECTION PLANE SEPARATION do

Example:

PROJ PLAN SEPA 1.5

Describer	Meaning
do	Separation of the observer and the projection plane on the r-axis in model units. The VANTAGE POINT command may also specify the separation (Real, Default = 2.0).

Remarks:

1. The PROJECTION PLANE SEPARATION command is not recommended. The FIND command is recommended because it automatically calculates the optimum separation.
2. A theoretical description of projection plane separation is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

**PTITLE** Plot Frame Title

Defines a character string that will appear at the top of the plot frame on the line below the sequence number.

Format:

PTITLE ptitle

Example:

```
PTITLE RIGHT WING -- LOAD CASE 3
```

Describer	Meaning
ptitle	Any character string (Character, Default= blank).

Remarks:

1. PTITLE may not be continued to the next command line.
2. Up to four lines of title information will be printed in the lower left-hand corner of each plot. The text for the top three lines is taken from the TITLE, SUBTITLE, and LABEL commands in the Case Control Section. (See the [Case Control Commands](#) for a description of the TITLE, SUBTITLE, and LABEL commands). The text for the bottom line may be of two forms depending on the type plot requested. One form contains the word UNDEFORMED SHAPE. The other form contains the type of plot (statics, modal, etc.) subcase number, load set or mode number, frequency or eigenvalue or time, and (for complex quantities) the phase lag or magnitude. The sequence number for each plot is printed in the upper corners of each frame. The sequence number is determined by the relative position of each PLOT execution command in the plot package. The information on the PTITLE command will be printed on the line below the sequence number. The date and (for deformed plots) the maximum deformation are also printed at the top of each frame.

**SCALE****Scale Factor**

Defines reduction, as a scale factor, of model's dimensions so that model fits on a plot frame.

Format:

SCALE a [ b ]

Example:

SCALE 0.5

Describers	Meaning
a	Scale factor (Default=1.0).
b	Ratio of model size/real object size for stereoscopic projection only.

Remarks:

1. The SCALE command is not recommended. The FIND command is recommended because it automatically calculates the optimum scale factor.
2. For orthographic or perspective projections, a is the ratio of the plotted object, in inches, to the real object in the units of model; i.e., one inch of paper equals one unit of model.
3. For stereoscopic projections, the stereoscopic effect is enhanced by first reducing the real object to a smaller model according to b, and then applying a. The ratio of plotted/real object is then the product of a and b.
4. If the NASTRAN general purpose plotter is used in combination with the PLOTTPS postprocessing routine, a scale factor can be computed as follows:

$$a = p \cdot \frac{20}{7} \cdot K$$

where:

$p$  = ratio of plot size to object size. For instance, if the model is 100 inches long and the plot size is 7 inches, then

$$p = \frac{7}{100} = .007$$

$\frac{20}{7}$  = ratio of default PAPER SIZE to default PLOTTPS frame size.

$K$  = SCALE value on PLOTTPS command (Default=1.0). See [Using the Utility Programs](#) in the *MSC Nastran Installation and Operations Guide*.

## SEPLOT

### Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to one or more superelements.

Format:

```
SEPLOT seid1 [ seid2 ... ]
```

Examples:

```
SEPLOT 5  
SEPLOT 0 3 7 200
```

Describer	Meaning
seidi	Superelement identification number (Integer $\geq 0$ ).

Remarks:

1. See also related command SEUPPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEPLOT (or SEUPPLOT) commands will apply in all SEPLOT (or SEUPPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEPLOT followed by a single FIND may be placed above all commands.

## SET Set Definition Under OUTPUT(PLOT)

Defines a set of elements or grid point numbers to be plotted.

The SET command specifies sets of elements or grid points, corresponding to portions of the model, which may be referenced by PLOT and FIND commands. The SET command is required. Each set of elements defines, by implication, a set of grid points connected by those elements. The set may be modified by deleting some of its grid points. The elements are used for creating the plot itself and element labeling, while the grid points are used for labeling, symbol printing, and drawing deformation vectors.

Format:

$$\text{SET } n = \left[ \text{ALL} \left[ \begin{array}{c} \text{ELEMENTS} \\ \text{GRID POINTS} \end{array} \right] \left[ \text{EXCEPT} \left\{ \begin{array}{c} \text{type1 type2 ... typej} \\ k1 \quad k2 \quad \dots \quad kj \text{ THRU } kk \text{ BY } \text{incj} \end{array} \right\} \right] \right],$$

$$\left[ \left[ \begin{array}{c} \text{INCLUDE} \\ \text{EXCLUDE} \end{array} \right] \left[ \begin{array}{c} \text{ELEMENTS} \\ \text{GRID POINTS} \end{array} \right] \left\{ \begin{array}{c} \text{type1 type2 ... typej} \\ k1 \quad k2 \quad \dots \quad kj \text{ THRU } kk \text{ BY } \text{incj} \end{array} \right\} \right],$$

$$\left[ \text{EXCEPT} \left\{ \begin{array}{c} \text{type1 typem ... typen} \\ k1 \quad km \quad \dots \quad kn \text{ THRU } ko \text{ BY } \text{incn} \end{array} \right\} \right]$$

Examples:

- SET 1 consists of elements 1, 5, 10, 11, 13, 14, 15, 20, 22, 24, and 26.  
`SET 1=INCLUDE 1, 5, 10 THRU 15 EXCEPT 12, INCLUDE 20 THRU 26 BY 2`
- SET 2 consists of all CTRIA3 and CQUAD4 elements except element 21.  
`SET 2=QUAD4 TRIA3 EXCEPT 21`
- SET 10 includes all CTRIAR elements plus elements 70 through 80.  
`SET 10 TRIAR INCLUDE ELEMENTS 70 THRU 80`
- SET 15 includes all elements from 15 to 20 and 26 to 100.  
`SET 15=15 THRU 100 EXCEPT 21 THRU 25`
- SET 2 includes all elements except CTETRA elements.  
`SET 2=ALL EXCEPT TETRA`

Descriptor	Meaning
n	Sets identification number (0 < Integer < 999999).
ALL	Selects all elements or grid points. See typei.
ELEMENTS	Specifies that all identification numbers refer to elements.
GRID POINTS	Specifies that all identification numbers refer to grid points.

Descriptor	Meaning
INCLUDE	Includes specified element or grid point identification numbers or elements in the set.
EXCLUDE	Excludes specified element or grid point identification numbers or element types in the set.
EXCEPT	Modifies a prior ALL, INCLUDE, or EXCLUDE specification.
typei	Element types. The allowed element types are (Character):

Element Type	typei on SET Command		Element Type	typei on SET Command
CAXIF2	AXIF2		CQUAD	QUAD
CAXIF3	AXIF3		CQUAD4	QUAD4
CAXIF4	AXIF4		CQUAD8	QUAD8
CBAR	BAR		CQUADR	QUADR
CBEAM	BEAM		CQUADX	QUADX
CBEND	BEND		CROD	ROD
CBUSH	BUSH		CSHEAR	SHEAR
CONEAX	CONE		CSLOT3	SLOT3
CONROD	CONROD		CSLOT4	SLOT4
CDUMi	DUMi		CTETRA	TETRA
CFLUID2	FLUID2		CTRIAX6	TRIAX6
CFLUID3	FLUID3		CTRIA3	TRIA3
CFLUID4	FLUID4		CTRIA6	TRIA6
CHBDYG	HBDY		CTRIAR	TRIAR
CHBDYP	HBDY		CTRIAX	TRIAX
CHEXA	HEXA		CTUBE	TUBE
CPENTA	PENTA		CVISC	VISC
PLOTEL	PLOTEL			

THRU	Specifies a range of identification numbers.
BY	Specifies an increment for a THRU specification.
inci	Increment for THRU range (Integer > 0).

**Remarks:**

1. This form of the SET command can only be specified after an OUTPUT(PLOT) delimiter.
2. The INCLUDE, EXCLUDE, and EXCEPT specifications may be specified more than once in the same set. See previous examples.
3. Commas or spaces may be used as separators.
4. Not all of the identification numbers in a THRU range have to correspond to elements or grid points. For example, elements 2, 4, 7, and 9 may be selected with 2 THRU 9, even if elements 3, 5, 6, and 8 do not exist. This is called an open set. Note that large open sets can cause higher computational costs.

## SEUPPLOT

### Superelement Plot Delimiter

Assigns the subsequent PLOT or XYPLOT commands to a superelement and all of its upstream superelements.

Format:

```
SEUPPLOT seid
```

Example:

```
SEUPPLOT 7
```

Describer	Meaning
seid	Superelement identification number (Integer $\geq 0$ ).

Remarks:

1. See also related command SEPLOT.
2. Any PLOT or XYPLOT commands appearing above all SEUPPLOT (or SEPLOT) commands will apply in all SEUPPLOT (or SEPLOT) packets.
3. For multiple PLOT or XYPLOT commands, there should be a SEUPPLOT command with each PLOT. For the special case where the PLOTs or XYPLOTs refer to the same superelements and use the same FIND, a single SEUPPLOT followed by a single FIND may be placed above all the commands.



## STEREOSCOPIC

Selects Stereoscopic Projection

See the description of the [ORTHOGRAPHIC, etc.](#)

**VANTAGE POINT****Location of the Observer**

Defines the location of the observer with respect to the model in the (r, s, t) coordinate system defined on the AXES command for perspective and stereoscopic projections only.

Format:

VANTAGE POINT ro so to do sor

Example:

VANT 100.

Describers	Meaning
ro	Location of the observer on the r-axis in model units (Real).
so	Location of the observer and left eye of the observer on the s-axis, in model units, for perspective and stereoscopic projections, respectively (Real).
to	Location of the observer on the t-axis in model units (Real).
do	Separation of the observer and the projection plane on the r-axis in model units. The PROJECTION PLANE SEPARATION command may also specify the separation (Real).
sor	Location of the of the observer's right eye for stereoscopic projections in model units (Real).

Remarks:

1. VANTAGE POINT or the FIND command must be specified if the PERSPECTIVE or STEREOSCOPIC command is also specified.
2. The VANTAGE POINT command is not recommended. The FIND command is recommended because it automatically calculates the optimum vantage point.
3. A theoretical description of the vantage point is contained in [Plotting](#) in the *MSC Nastran Reference Guide*.

## VIEW Angular Relationship of Observer's Coordinate System

Defines the angular relationship between observer's coordinate system (r, s, and t axes specified on the AXES command) and basic coordinate system.

Format:

VIEW gamma beta alpha

Example 1:

View the model from the r-axis.

VIEW 0. 0. 0.

Example 2:

View the model midway between the r- and s-axes.

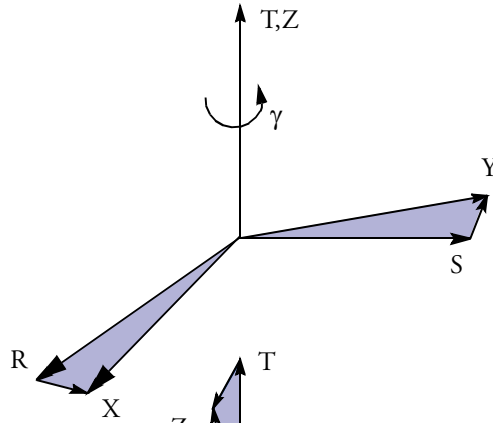
VIEW 45. 0. 0.

Describers	Meaning
gamma	Angle of rotation, in degrees, of t axis specified on AXES command (Default = 34.27).
beta	Angle of rotation, in degrees, of s axis specified on AXES command (Default = 23.17 if ORTHOGRAPHIC or STEREOSCOPIC command is specified and 0.0 if PERSPECTIVE command is specified).
alpha	Angle of rotation, in degrees, of r axis specified on AXES command (Default = 0.0).

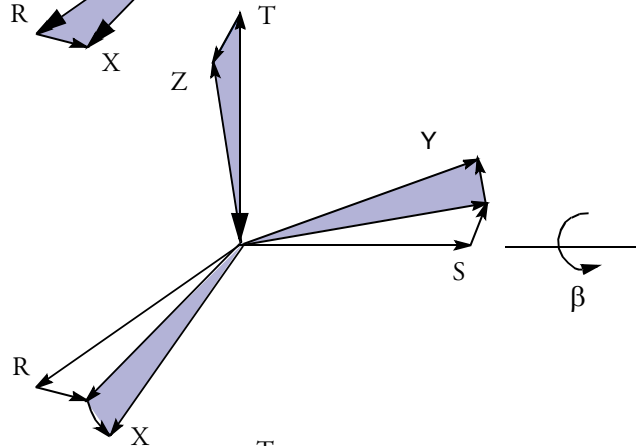
Remarks:

1. If no VIEW command is specified, then VIEW 34.27 23.17 0.0 is assumed for orthographic and stereoscopic projections; and VIEW 34.27 0.0 0.0 is assumed for perspective projections. The default values produce a plot in which unit vectors on the axes of the basic coordinate system have equal lengths.
2. The angles are rotated in sequence: gamma rotates the t-axes, followed by beta which rotates the s-axes, followed by alpha which rotates the r-axes.

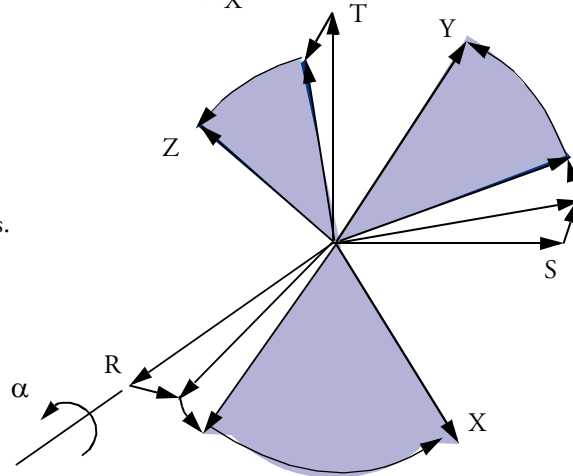
(a) g-rotation about T-axis.



(b) b-rotation about S-axis.



(c) a-rotation about R-axis.



- The VIEW command specifies the position of the model with respect to the s-t plane. Gamma and beta represent the angles of turn and tilt. Alpha is normally not used since it does not affect the orientation of the s-t plane, but only its orientation on the plot frame.

# 6

## Parameters

- Parameter Descriptions
- Parameter Applicability Tables

## Parameter Descriptions

Parameters are used extensively in the solution sequences for input of scalar values and for requesting special features. Parameters values are specified on PARAM Bulk Data entries or PARAM Case Control commands. The PARAM Bulk Data entry is described in the [Bulk Data Entries](#). The PARAM Case Control command is described in the [PARAM \(Case\)](#). PARAMs may also be used in `.rc` and `.ini` files as described in [Specifying Parameters](#) in the *MSC Nastran Installation and Operations Guide*. A complete alphabetical list of PARAMeter names and their functions is given in this section.

If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:

1. PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
2. PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.
3. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.

### ACEXTMTD

ACEXTMTD is used to select a different solver for solving exterior acoustic model. Valid options are ITER for selecting iterative solver and KRYLOV for selecting Krylov solver.

### ACEXTSET

ACEXTSET is used to select ITER bulk data entry for PARAM,ACEXTMTD,ITER.

### ACMPF

Default = -1

ACMPF specifies a Fortran unit number to import the acoustic load modal participation factors provided by Actran in OP4 file to Nastran SOL 111 to perform stress and strain recovery, random and fatigue analyses. This parameter is compatible with multiple acoustic load cases.

The Fortran unit must be a valid one which should not conflict with Nastran reserved Fortran unit numbers.

### ACOUT

Default = PEAK

ACOUT specifies the type of output to be used with the FORCE Case Control command in coupled fluid-structural analysis. ACOUT=RMS requests root-mean-square output.

To obtain sound pressure level in units of dB and dBA given by the FORCE command, a peak reference pressure must be specified with PARAM, PREFDB. The dB level is defined as:

$$\text{dB} = 20 \cdot \log\left(\frac{P}{\text{PREFDB}}\right)$$

Instead of ACCELERATION, suffix INT can be utilized to generate INTENSITY. See also the Case Control command [FLSTCNT \(Case\)](#).

## ACOWEAK

Default = NO

ACOWEAK controls the execution of weakly coupled acoustic formulation logic. PARAM,ACOWEAK,YES will activate weakly coupled acoustic formulation which will solve exterior acoustic model along with CACINFX separately using load generated from structural response and coupling matrix. ACOWEAK,YES is available only in SOL 111 and SOL 200 with ANALYSIS=MFREQ.

In the cases where loading is on exterior acoustic only, PARAM,ACOWEAK,YESR will solve for exterior acoustic model first and generates loading at coupling nodes for structural and interior acoustic model. In addition, PARAM, ACOWEAK, YESR can be used to force NASTRAN to solve exterior acoustic model first if loading is on both structural and exterior acoustic models.

Note that the detailed knowledge on the loading is not required to use ACOWEAK, and with PARAM, ACOWEAK, yes, the Automated logic will determine which part of FSI to solve first.

The ACOWEAK solution technique supports trim components via TRIMGRP or ACTRIM between structure and interior cavity only. If there are trim components situated between the structure and exterior cavity, fully coupled solution techniques should be utilized.

With ACOWEAK solution scheme, the exterior acoustic portion of the model is solved in physical coordinates. Therefore, acoustic structural modal participation factor will not be possible since acoustic field eigenvalues and/or eigenvectors are missing or incomplete.

## ACSYM

Default = YES

By default, the dynamic equations for coupled fluid-structure analysis in frequency response are symmetrized for efficiency. PARAM,ACSYM,NO requests the pre-Version 69 formulation which involves no symmetrization and will require more CPU time. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*. See also the Case Control command [FLSTCNT \(Case\)](#).

If the iterative solver is selected (see the Case Control command, [SMETHOD \(Case\)](#)) then the external work diagnostic will be different between ACSYM=YES and ACSYM=NO.

## ACTDMP

Default=0

This parameter can be used to specify DMP processors for Actran. By default(ACTDMP=0), it will be set to DMP of Nastran job submittal.

## ACTSMP

Default=0

This parameter can be used to specify SMP processors for Actran. By default(ACTSMP=0), it will be set to SMP of Nastran job submittal.

## ACTMEM

Default=0

This parameter can be used to specify TOTAL amount memory for all processors for Actran via DMP or PARAM,ACTDMP in 4 bytes mega-words. By default (ACTMEM=0), Actran get the memory assigned to parent node only. An example, for 'memorymax=64gb mem=max', use PARAM,ACTMEM,16000 which get ACTRAN same amount of TOTAL memory as NASTRAN.

## ADJMETH

Default = 0

This parameter selects the processing method used in a triple matrix product in module DSADJ. The default is usually preferred, but ADJMETH=1 can be used when disk space is critical. ADJMETH=2 only holds the active solution vectors.

## ADMEXTU

Default = 0

ADMEXTU is a file unit number used by the SOL 111/112 restart run along with ADMPOST=1 or 2. This file unit no is assigned to external super element op2 file. See the Case Control command [ADAMSMNF\\* \(Case\)](#), footnote and Remark 20.

## ADMPOST

Default = 0

This parameter allows the user to bring an ADAMS results file into Nastran for modal data recovery in SOL 111 and SOL 112. A value of 1 brings in the ADAMS results without consideration of rigid body motion in the display of the results. A value of 2 brings in the ADAMS results with consideration of rigid body motion in the display of the results. See the Case Control command [ADAMSMNF\\* \(Case\)](#) footnote and Remark 19. for more details on restrictions and required file assignments.

## ADPCON

Default = 1.0

Initial penalty values used in contact analysis are calculated automatically by the program and are given by  $k \cdot \text{SFAC} \cdot |\text{ADPCON}|$  where k is a number selected for each secondary node based on the diagonal stiffness matrix coefficients that are in the contact region, and SFAC is the value specified by the user in the SFAC field of the BCONP Bulk Data entry. The ADPCON value applies to all the contact regions in the model. During the analysis, if convergence problems are encountered, penalty values are automatically reduced. Still there may be some problems where convergence can not be achieved with default values. In such cases, analysis may be restarted with a lower value of ADPCON.

In some cases the default penalty values may be low. In such situations analysis may be restarted with a higher value of ADPCON.

Generally, penalty values are recalculated every time there is a change in stiffness. However, if ADPCON is negative, penalty values are calculated only at the beginning of each subcase, and penalty values are not adjusted during analysis. This is useful if the contact between two elastic bodies is being analyzed.



## ADSTAT

Character; Default = YES

In transient analysis (SOLs 109 and 112) or static analysis SOL101, if there is a preload (see Case Control command [STATSUB \(Case\)](#)) and ADSTAT=YES, the static displacements, SPC forces, and MPC forces will be included in the transient output. There is a limitation that OLOAD is not included. Another limitation is that element stress and force recover will neglect thermal strains and, hence will be wrong if there is a TEMP(LOAD) request. For any other value of ADSTAT the preload effects will not be included in the output, hence the output will be the perturbation displacements, SPC forces and MPC forces.

## AERODOF

Default=35

Digits 1 through 6 indicate which k-set degrees of freedom are to be output. This is only for Double Lattice lifting surfaces that provide output in the plunge (3) and pitch (5) degrees of freedom by default. If additional digits are specified (e.g. AERODOF=1235), then the k-set aerodynamic matrices are padded with null columns for the additional degrees of freedom. Nastran will not create aerodynamic data for these added degrees of freedom. This parameter is only useful to the user that knows how to augment the existing aerodynamic data from an external source. Degrees of freedom 3 and 5 may not be omitted.

## AESDISC

Default = 1.E-8

Tolerance for discarding generalized coordinates in the RITZ method (see Case Control command, [AESMETH](#)) which are not linearly independent.

## AESMAXIT

Default = 15

Maximum number of iterations for the ITER method (see Case Control command, [AESMETH](#)).

## AESMETH

Default = SELECT

Solution method for static aeroelastic analysis.

- |        |   |
|--------|---|
| SELECT | selects the DIRECT method on models with less than 50000 DOF in the solution set; otherwise selects AUTO.             |
| AUTO   | selects the reduced basis method for an approximate solution, which is used as starting vectors for an ITER solution. |
| DIRECT | selects the direct solution.  |
| RITZ   | selects the reduced basis approximate solution.   |
| ITER   | selects the iterative solution.   |

## AESRNDM

Default = 2

Number of random vectors to use as generalized functions in the RITZ method (see Case Control command, [AESMETH](#)).

## AESTOL

Default = 1.E-10

Convergence criteria for the iterative solver.

## AGGROT

Default = YES

PARAM, AGGROT, YES (Default) produces both translation and rotational terms in the structure-fluid interface matrix AGG. PARAM, AGGROT, NO produces only translational terms in the structure-fluid interface matrix AGG. Since fluid elements see no rotational degrees of freedom, AGGROT YES or NO should have no significant effect on the results of the fluid cavity. Results on the structural side may change especially near unconstrained boundaries where there may be a reversal in the signs of some acceleration terms.

## ALPHA1, ALPHA2

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1 and/or ALPHA2 are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1 is the scale factor applied to the mass matrix and ALPHA2 to the structural stiffness matrix. In SOL400, ALPHA1 and ALPHA2 are supported for both the linear and nonlinear elements. In SOL 129, Alpha1 scales the mass matrix, and ALPHA2 is not used.

$$[B'] = [B] + \text{ALPHA1} \cdot [M] + \text{ALPHA2} \cdot [K]$$

If  $\xi_j$  is the damping ratio for the  $i$ -th mode  $\omega_i$  (radians/unit time), then ALPHA1 ( $\alpha_1$ ) and ALPHA2 ( $\alpha_2$ ) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{bmatrix} \omega_j & -\omega_i \\ -1 & 1 \\ \omega_j & \omega_i \end{bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other  $\xi_l$  mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[ \left( \frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left( \frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

**Note:** The use of Rayleigh damping with non-zero values of ALPHA1 may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.

Similarly, the use of Rayleigh damping with non-zero values of ALPHA2 may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.

Since Version 2005r3, the ALPHA1, ALPHA2 parameters apply only to stationary structural components. Prior to this version, the parameters applied to both stationary and rotating structural components. With the introduction of Version 2005r3, new parameters ALPHAR1, ALPHAR2 are defined on the RSPINR and RSPINT bulk data entry for the rotating component, see Remark 8. of the DAMPING bulk data description for a comment on this item.

Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.

### ALPHA1FL, ALPHA2FL

Default = 0.0, 0.0

In frequency and transient response analysis, if PARAM,ALPHA1FL and/or ALPHA2FL are not equal to zero, then Rayleigh damping is added to the viscous damping. ALPHA1FL is the scale factor applied to the mass matrix and ALPHA2FL to the fluid stiffness matrix.

$$[B'] = [B] + \text{ALPHA1FL} \cdot [M] + \text{ALPHA2FL} \cdot [K]$$

If  $\xi_i$  is the damping ratio for the  $i$ -th mode  $\omega_i$  (cycles/unit time), then ALPHA1FL ( $\alpha_1$ ) and ALPHA2FL ( $\alpha_2$ ) may be computed as

$$\begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix} = \frac{2\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \begin{bmatrix} \omega_j & -\omega_i \\ -1 & 1 \\ \omega_j & \omega_i \end{bmatrix} \begin{Bmatrix} \xi_i \\ \xi_j \end{Bmatrix}$$

and the damping ratio for any other  $\xi_l$  mode becomes

$$\xi_l = \frac{\omega_i\omega_j}{\omega_j^2 - \omega_i^2} \left[ \left( \frac{\omega_j}{\omega_l} - \frac{\omega_l}{\omega_j} \right) \xi_i - \left( \frac{\omega_i}{\omega_l} - \frac{\omega_l}{\omega_i} \right) \xi_j \right]$$

**Note:** The use of Rayleigh damping with non-zero values of ALPHA1FL may not be appropriate for enforced motion problems involving large mass since the resulting damping matrix may essentially violate the assumption of large mass in the problem and thus give wrong answers.

Similarly, the use of Rayleigh damping with non-zero values of ALPHA2FL may not be appropriate for enforced motion problems involving large stiffness since the resulting damping matrix may essentially violate the assumption of large stiffness in the problem and thus give wrong answers.

Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.

## ALTRED

Default = NO

ALTRED=YES requests the alternate stiffness and load reduction technique for superelement analysis in SOLs 101 and 114. This technique is described in [Static Solutions in SubDMAP SEKRRS](#), [Static and Dynamic Load Generation](#), and [Data Recovery Operations in SubDMAP SEDISP](#) in the *MSC Nastran Reference Guide*.

## ARBMASP

Default = 2

ARBMASP defines the maximum aspect ratio of CQUAD4 generated for CP/OP options of PBMSECT if ARBMSTYP=timoshen. Since the thickness of a ply is usually the small dimension, ARBMASP affects the size of CQUAD4 lengthwise along a segment.

## ARBMFEM

Default = YES

ARBMFEM controls the generation of .bdf file which contains the Finite Element Model of the arbitrary beam cross section. This parameter is functional for PBRSECT and PBMSECT only. To turn off the capability, set value of ARBMFEM to 'NO'.

## ARBMNOW

Default = 0

ARBMNOW controls the overwrite of segment property for PBMSECT, arbitrary beam cross section. With default value of ARBMNOW, segment property overwrite will occur during the process of assigning the property for each segment. To turn off segment property overwrite, use PARAM,ARBMNOW,1.

## ARBMPS

Default = YES

ARBMPS controls the generation of outline plot for arbitrary beam cross section in PostScript format. This parameter is functional for PBRSECT and PBMSECT only. To turn off capability, set value of ARBMPS to 'NO'.

## ARBMS

Default = NO

ARBMS controls the stress recovery for the whole arbitrary beam cross section and the companion ‘screened’ stresses. The stress recovery for the whole cross section is available in ‘OP2’ format and suitable for post-processing. The ‘screened’ stresses for CBAR and CBEAM elements is available in print file (f06) and can be utilized for design optimization via RTYPE=ABSTRESS on DRESP1. This parameter is functional for PBRSECT and PBMSECT only. To turn on the capability, set value of ARBMS to ‘YES’.

**Note:** The recovery of ‘screened’ stresses will be turned on automatically if RTYPE=ABSTRESS is in use on DRESP1.

## ARBMSTYP

Default = TIMOSHEN

ARBMSTYP controls the solution algorithm for composite arbitrary beam cross-section. Other valid option is ‘TIMOFORC’. This parameter is functional for PBRSECT and PBMSECT only.

## ARF

Default = 0.95

See [FLUIDMP](#).

## ARS

Default = 0.95

See [FLUIDMP](#).

## ASCOUP

Default = YES

In coupled fluid-structure analysis, by default, the coupling between the fluid and structure is computed. This interaction will be ignored if PARAM,ASCOUP,NO is specified. See also the Case Control command [FLSTCNT \(Case\)](#) for alternative selections.

## ASING

Default = 0

ASING specifies the action to take when singularities (null rows and columns) exist in the dynamic matrices (or  $[K_{II}]$  in statics). If ASING=-1, then a User Fatal Message will result.

If ASING=0 (the Default), singularities are removed by appropriate techniques depending on the type of solution being performed. The procedures used are described in [Data Recovery Operations in SubDMAP SEDRCVR](#) and [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide*.

## AUNITS

Default = 1.0

AUNITS is used in SOL 144 to convert accelerations specified in units of gravity on the TRIM Bulk Data entry to units of distance per time squared. Accelerations are divided by this parameter. To convert accelerations input in g's into physical, consistent units, set AUNITS to 1/g.

## AUTOADJ

Default = YES

In SOL 200, a value of yes will automatically choose the direct or adjoint sensitivity analysis based on the performance criteria. Other options for this parameter are:

NO - the adjoint method will not be selected.

NOSTAT - the adjoint method will not be selected for static analysis subcases, but may be selected for frequency response subcases based on the performance criteria.

NOFREQ - the adjoint method will not be selected for frequency responses subcases, but may be selected for static analysis subcases based on the performance criteria.

The default should be preferred in all cases. However, the other options allow investigation of the alternative of using direct sensitivity methods.

## AUTOGOUT

Default = NO

PARAM,AUTOGOUT,YES simplifies grid point data recovery requests when only a set of elements is specified. In other words, given a set of elements the program will automatically determine all connected grid points and honor the data recovery requests for both grid points and elements. See also PARAM,OELMOPT description.

If PARAM,AUTOGOUT,YES is specified then the program assumes that SET 2 defines the set of desired elements and SET 1 defines the additional grids **not** connected to the elements in SET 2. If no additional grids are desired then the user need only specify SET 1=0. For example, in Case Control,

```
Param,autogout,yes
Set 2 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 1 = 0 $ additional grids
Displ=1
Gpfor=1
Stres=2
```

If the user prefers to use different set IDs then user PARAMs OSETELE and OSETGRD may be used to change the set ids used by this feature. For example:

```
Param,autogout,yes
Param,osetele,200
Param,osetgrd,100
Set 200 = 27,35,25,41234,123,thru,134,9701,9901 $ elements
Set 100 = 0 $ additional grids
Displ=100
Gpfor=100
Stres=200
```

And if the user wants to add more grids that are not connected to the element set:

```
Param, autogout, yes
Param, osetele, 200
Param, osetgrd, 100
Set 200 = 27, 35, 25, 41234, 123, thru, 134, 9701, 9901 $ elements
Set 100 = 29 $ additional grids
Displ=100
Gpfor=100
Stres=200
```

## AUTOMSET

Character, Default = NO (Except in SOL 400 with ANALYSIS=STATICS)

The relationship between dependent and independent degrees-of-freedom for rigid elements and MPCs may be altered from the user's input specification via PARAM,AUTOMSET,YES or PARAM,AUTOMSET,LUSOL. For PARAM, AUTOMSET,YES, or PARAM,AUTOMSET, LUSOL, dependent degrees-of-freedom of the model, the M-set are automatically determined by employing a rectangular decomposition of the RMG matrix.

- |       |  |
|-------|--|
| YES   | For solutions other than SOL 600, the rectangular decomposition is performed via UMFPACK, which is used by permission. Copyright (c) 2002 by Timothy A. Davis. All rights reserved. Availability: <a href="http://faculty.cse.tamu.edu/davis/suitesparse.html">http://faculty.cse.tamu.edu/davis/suitesparse.html</a>  |
| LUSOL | The rectangular decomposition is performed via LUSOL, which is capable of Rook or full pivoting during rectangular decomposition. LUSOL option is especially suitable for ill conditioned RMG matrix or problems with redundant multi-point constraints. Availability: <a href="http://web.stanford.edu/group/SOL/software/lusol/">http://web.stanford.edu/group/SOL/software/lusol/</a> |

### Notes:

- When used in SOL 600, AUTOMSET=Yes maps to Marc's AUTOMSET. By default or if PARAM,AUTOMSET,NO is entered, Marc's AUTOMSET will be omitted. See Volume C of Marc for further details.
- PARAM,AUTOMSET,YES is not supported for SOL 200.
- RSSCON is not supported with AUTOMSET.

## AUTOQSET

Default = NO

AUTOQSET=YES requests the automatic calculation of component modes without the need to define a q-set (generalized coordinates).

1. The calculation of component modes is attempted on all superelements including the residual structure.

2. All component modes are treated like SPOINTs which means that all component modes are “brought down” to and assigned to the q-set in the residual structure. In other words, component modes may not be assigned interior to a superelement.
3. Selected component modes may not be removed (constrained).
4. Since the generalized coordinates are automatically defined, the following entries may not be specified: QSETi, SEQSETi, SENQSET, or PARAM,NQSET.
5. This feature is not supported with:
  - a. Design optimization (SOL 200)
  - b. Aerodynamic analyses (SOLs 144, 145, 146)
  - c. Cyclic symmetry analyses (SOLs 114, 115, 116, 118)
  - d. SECSETi and SEBSETi Bulk Data entries that result in a fixed-free or free-free superelement. This restriction also applies to BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries in part (BEGIN SUPER) superelements.

PARAM,SQSETID may be used to specify the starting identification number of the automatically generated q-set degrees-of-freedom except when the EXTSEOUT command is present (see Remark 17. of the EXTSEOUT Case Control command). The default is 99000001.

## AUTOSPC

Default = YES

This parameter is obsolete and replaced by the Case Control command [AUTOSPC \(Case\)](#). It is ignored for the residual structure in all nonlinear analysis solution sequences. SOL 400 only support the AUTOSPC Case Control command. For the residual structure in SOLs 106 and 129 see the description of parameter AUTOSPCR. AUTOSPC and the related parameters EPPRT, EPZERO, PRGPST, and SPCGEN are analogous to and described under the AUTOSPC Case Control command under the EPSING, EPS, PRINT, and PUNCH keywords, respectively. The specification of the AUTOSPC Case Control command overrides the specification of PARAM,AUTOSPC and its related parameters EPPRT, EPZERO, PRGPST, and SPCGEN.

## AUTOSPCR

Default = NO (SOLs 106 and 129 only)

In SOLs 106 and 129 only, AUTOSPCR specifies the action to take when singularities exist in linear stiffness matrix of the residual structure after multipoint constraints are processed. AUTOSPCR=YES means that singularities will be constrained and AUTOSPCR=NO means they will not be constrained. It is recommended that all degrees-of-freedom attached to nonlinear elements be specified on ASETi entries. Parameters EPPRT, EPZERO, PRGPST, and SPCGEN may be used with AUTOSPCR.

## AUTOSPRT

Default = YES

By default, free-free models will be automatically constrained for calculation of residual vectors (RESVEC Case Control command) as long as  $F1 \leq 0.0$  on the EIGR (or EIGRL) Bulk Data entry. The auto-SUPPORT method may be deactivated by specifying a SUPORTi entry, PARAM,AUTOSPRT,NO, or  $F1 > 0.0$ .



FZERO is the maximum frequency assumed for a rigid body mode. FZERO is used by the auto-SUPPORT method to extract the rigid body frequencies. The default is 1.0.

## BAILOUT

Default = 0

See [MAXRATIO](#).

## BEAMBEA

Real, Default = 1000.0

Value           Equivalent radius to be used for beam-beam contact problems. For tubes or round bars, use the outer radius. If the radii are different enter the largest outer radius. For beams, enter an equivalent radius calculated as follows:

$$I=0.5*(I_x+I_y)$$

$$R=\sqrt{A/\pi^2+2*I/A}$$

where A, I<sub>x</sub>, I<sub>y</sub> are the cross-section properties and  $\pi^2 = \Pi^2$ .

## BEIGRED

Default = YES

PARAM, BEIGRED, YES requests a more efficient method to reduce viscous damping matrices. CPU and disk space savings may be significant if there is a small number of viscous damping elements. BEIGRED is automatically disabled if the number of DOF connected to viscous dampers exceeds the value specified by PARAM, MAXDAMP.

MAXDAMP (default=1000) specifies the maximum number of DOF connected to viscous damping elements, such that an alternate reduction method is employed. When there are relatively few viscous dampers, an alternate reduction method is chosen automatically for efficiency. The efficiency is lost however if the number of viscous damping DOF exceeds about 1000. If the number of viscous damper DOF exceeds MAXDAMP, User Warning Message 9166 is printed and the alternate damping reduction logic is disabled. Also note that if MAXDAMP is exceeded, FASTFR is disabled.

## BIGER, BIGER1, BIGER2, BIGER3

Default = 0.0

See S1.

## BUCKLE

Default = -1

BUCKLE=1 requests a nonlinear buckling analysis in a restart run of SOLs 106 or 153. See the *MSC Nastran Handbook for Nonlinear Analysis*. (Not supported for SOL 600.)

BUCKLE=2 requests buckling in a SOL 106 cold start run. (Must be in Bulk Data for SOL 600.)

## BUSHNM

Default = YES

BUSHNM =NO - If the only frequency dependencies are CBUSH/PBUSH/PBUSHT, (CELAS1 or CELAS3)/PELAS/PELAST, or (CDAMP1 or CDAMP3)/PDAMP/PDAMPPT, or CAABSF/PAABSF or any combination of these, then the DMAP will execute the original DMAP path designed for these elements.

The original DMAP path designed for the above elements does not make use of the concept of master frequencies controlled by the Case Control Command MFREQ=n. To obtain the same forcing frequency selection, and hence the same results, with PARAM,BUSHNM,YES that is in effect, with PARAM,BUSHNM,NO, the user should either set MFREQ to the same set as defined on the Case Control Command FREQ=n, or set MFREQ=NOAUTO.

PARAM,BUSHNM,NO in conjunction with a GPFORCE request will cause a Nastran fatal error at the beginning of the Nastran run. The reason is that the stiffness, structural damping, and viscous damping matrices are non-standard in form for the old path and represent incremental changes between nominal and current frequency.

If the user is currently running Porous Elastic Material (PEM), PARAM, BUSHNM is internally set to: PARAM, BUSHNM, NO and overrides any input value.

## CASIEMA

Default = NO, SOL 400 CASI Solver only

When the CASI element-based iterative solver is specified (see the Case Control command [SMETHOD \(Case\)](#)) SOL 400 skips various processes associated with the global structure stiffness matrix. The absence of a complete assembled stiffness matrix and its sub-sets can lead to a termination of the solution process in the NLSOLV module if the CASI solver detects a singularity. If the parameter is set to YES, SOL 400 performs all assembly and displacement set reduction operations involving the complete stiffness matrix allowing the NLSOLV module to select either the matrix-based iterative solver or the sparse direct solver to continue in the event CASI detects a singularity.

Use a value of YES with caution as it adversely affects run-time performance. Substantial additional disk space could be required as well as longer run times. Keep in mind that invocation of the sparse direct solver for large solid element models may degrade performance even further.

This parameter only applies to SOL 400 usage of the CASI solver.

## CASIMEST

Default = Yes

This parameter controls the memory estimation inside CASI iterative solver. (SOL 101, SOL 400 and SOL 200)

= No; Turns off memory estimation.

## CASPIV

Default = 1.0e-10

Input DMAP parameter to the SOLVIT and NLSOLV modules for the CASI iterative solver. It is the pivot threshold for the CASI PCG (Preconditioned Conjugate Gradient) factorization. Pivot value less than CASPIV will exit with an error message.

## CB1, CB2

Default = (1.0, 0.0)

CB1 and CB2 specify factors for the total damping matrix. The total damping matrix is:

$$[B_{jj}] = CB1 \cdot [B_{jj}^x] + CB2 \cdot [B_{jj}^2]$$

where  $[B_{jj}^2]$  is selected via the Case Control command B2GG and  $[B_{jj}^x]$  comes from CDAMPi or CVISC element Bulk Data entries. These parameters are effective only if B2GG is selected in the Case Control Section.

## CDIF

Default = YES for shape optimization with or without property optimization.

Default = NO for property optimization only.

CDIF may be used to override the default finite difference scheme used in the calculation of pseudo loads in SOL 200. PARAM,CDIF,YES forces the selection of the central difference scheme used in the semianalytic approach regardless of the type of optimization requested. PARAM,CDIF,NO forces the selection of the forward difference scheme.

## CDITER

Default = 0

If CDITER > 0, perform constrained displacement iterations in SOL 101. The value is the maximum number of iterations. If CDPRT=YES, print those negative displacements and tension forces which do not satisfy constraints. If CDPCH=YES, punch DMIG CDSHUT entries for final state; by default all gaps are closed. These can be used for initial conditions for restart. Potential contact points must be specified on the SUPORTi entries. The SUPORTi points must be in the residual structure. Optional DMIG entries to define the initial shut vector may be specified. Degrees-of-freedom that are specified on the SUPORT entry and have a value of 1.0 defined on the DMIG,CDSHUT entry will be considered closed initially.

If the DMIG,CDSHUT entry is not supplied, then all degrees-of-freedom specified on the SUPORT entries will be considered shut initially. A fatal message will be issued if this parameter is used and PARAM,INREL is specified. Constrained displacement iteration (PARAM,CDITER,YES) does not support parallel processing (DMP).

## CDPCH

Default = NO

See [CDITER](#), 807.

## CDPRT

Default = YES

See [CDITER](#), 807.

## CFDIAGP

Default = NO

If YES, randomly deleted CFAST elements will be printed. (See [CFRANDEL](#))

## CFRANDEL

Default = 0.

Represents a percent, expressed as a decimal fraction, of the number of CFAST elements to be randomly deleted.

## CHECKOUT

Default = NO

CHECKOUT=YES requests a model checkout in SOLs 101 through 200. See [Geometry Processing in SubDMAP PHASE0](#) in the MSC Nastran Reference Guide. The run will terminate prior to phase 1 of superelement analysis. The PARAM,POST options are also available with PARAM,CHECKOUT,YES. The following options and their user parameters are also available with PARAM,CHECKOUT,YES:

1. PARAM,PRTGPL,YES

Prints a list of external grid and scalar point numbers in internal sort. It also lists external grid and scalar point numbers along with the corresponding sequence numbers in internal sort. The sequence numbers are defined as (1000\*external number) and will reflect any user-requested resequencing.

2. PARAM,PRTEQXIN,YES

Prints a list of external and internal grid and scalar numbers in external sort. It also lists external grid and scalar numbers with the corresponding coded SIL number in external sort. The coded SIL numbers are defined as:

$$10 \cdot \text{SIL} + \begin{cases} 1 & \text{for grid point} \\ 2 & \text{for scalar point} \end{cases} \quad (1)$$

The SIL numbers correspond to degrees-of-freedom, i.e., one SIL number for scalar point and six SIL numbers for a grid point.

3. PARAM,PRTGPDT,YES

Prints, for each grid and scalar point, the following information in internal sort:

- Coordinate system ID in which grid point geometry is defined (ID=-1 for scalar points).
- Spatial location of grid points in the “CP” coordinate system. For scalar points, all entries are zero.
- Coordinate system ID for grid point displacements, forces, and constraints (ID=0 for scalar points).
- Permanent single-point constraints defined on GRID Bulk Data entries. A zero is entered for scalar points.

4. PARAM,PRTCSTM,YES  
Prints for each coordinate system type the transformation matrix from the global to the basic coordinate system, and the origin of the indicated coordinate system in the basic coordinate system. Coordinate system types are: 1 = rectangular; 2 = cylindrical; 3 = spherical.
5. PARAM,PRTBGPDT,YES  
Prints all grid and scalar points listed in internal sort with their x, y, and z coordinates in the basic coordinate system. In addition, the coordinate system ID for grid point displacements, forces, and constraints is indicated for each grid point (ID=-1 for scalar points). The x, y, and z coordinates of scalar points are zero.
6. PARAM,PRTGPTT,YES  
Prints, for each temperature load set, information on element and grid point temperatures.
7. PARAM,PRTMGG,YES  
Prints the g-size mass matrix labeled by grid point/degree-of-freedom.
8. PARAM,PRTPG,YES  
Prints the g-size load vectors labeled by grid point/degree-of-freedom.
9. The summation of forces and moments of applied loads in the basic coordinate system is automatically output for each loading condition requested in the Case Control Section. Related to parameters GPECT and PROUT, and Case Control command ELSUM.

### CK1, CK2

Default = (1.0, 0.0)

CK1 and CK2 specify factors for the total stiffness matrix. The total stiffness matrix (exclusive of GENEL entries) is

$$[K_{jj}^x] = CK1 \cdot [K_{jj}^z] + CK2 \cdot [K_{jj}^2]$$

where  $[K_{jj}^2]$  is selected via the Case Control command K2GG and  $[K_{jj}^z]$  is generated from structural element (e.g., CBAR) entries in the Bulk Data. These are effective only if K2GG is selected in Case Control. A related parameter is CK3.

**Note:** Stresses and element forces are not factored by CK1, and must be adjusted manually.

### CK3

Default = (1.0, 0.0)

CK3 specifies a factor for the stiffness derived from GENEL Bulk Data entries. The total stiffness matrix is

$$[K_{jj}] = [K_{jj}^x] + CK3 \cdot [K_{jj}^y]$$

where  $[K_{jj}^y]$  comes from the GENEL Bulk Data entries and  $[K_{jj}^x]$  is derived using PARAMs CK1 and CK2. CK3 is effective only if GENEL entries are defined. Related parameters include CK1 and CK2.

## CLOSE

Default = (1.0)

A value of 1.05 will set natural frequencies within 5% of each other as close. See SCRSPEC.

## CM1, CM2

Default = (1.0, 0.0)

CM1 and CM2 specify factors for the total mass matrix. The total mass matrix is

$$[M_{jj}] = CM1 \cdot [M_{jj}^x] + CM2 \cdot [M_{jj}^2]$$

where  $[M_{jj}^2]$  is selected via the Case Control command M2GG and  $[M_{jj}^x]$  is derived from the mass element entries in the Bulk Data Section. These are effective only if M2GG is selected in the Case Control Section.

## COMPMA TT

Default = NO, SOL 106 and SOL 400 only

In nonlinear statics (SOLs 106, 400 ANALYSIS=NLSTAT), composite materials compute temperature-dependent properties for the plies only at the reference temperature given on the PCOMP Bulk Data entry. The ply properties are smeared and used for all load steps, regardless of whether the temperature is changing through application of thermal loads.

If the parameter is set to YES, the temperature-dependent properties for the plies are updated and smeared at the current temperature for each load step. For CQUAD4 and CTRIA3 elements with offsets, the large rotation offset method specified by "MDLPRM,OFFDEF,LR OFF" in SOL 400 must be used.

If the parameter is set to NONSMEAR, the temperature-dependent properties for the plies are updated at the current temperature for each load step. This option, only available for the CQUADR and CTRIAR elements, is an alternative to the smeared approach.

This parameter only applies to SOLs 106 and 400 ANALYSIS=NLSTAT, and only applies to composite CQUAD4, CTRIA3, CQUADR and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

Prior to Nastran version 2019, when COMPMA TT was set to YES, material nonlinear was assumed even when no composite plies had temperature-dependent materials or when there was no MATTi type entries associated with temperature dependent plies. This has caused confusion as to when SOL106 or SOL400 is doing linear or nonlinear material solutions. Starting in V2019, PARAM, COMPMA TT, YES will only turn on material nonlinear if there are active MATTi entries as well.

Additionally, prior to Nastran version 2019, when PARAM, COMPMA TT, YES was used in conjunction with TEMPP1 entries, ply stress computation used only the reference plane temperature. Starting in V2019, ply stress computation will use reference plane temperature plus the thermal gradient term applied at the

center of each ply. (To revert to the old method, NASTRAN SDRCMPTP=1 may be added to the Nastran section.) The enhanced nonlinear materials (PSHLN1) do not support the TEMPP1 entry.

## CONFAC

Default = 1.E-5

In superelement analysis, CONFAC specifies the tolerance factor used in checking the congruence of the location and displacement coordinate systems of the boundary points between image superelements and their primaries (see the Bulk Data entry, CSUPER). Specification of this parameter is recommended instead of DIAG 37 (DIAG 37 ignores User Fatal Messages 4277 and 4278).

## COSUBCYC

Default = 1, SOL 700 only

Controls the growth of the subcycling interval in the coupling surface.

### Example:

```
PARAM,COSUBCYC,2
```

### Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, COSUBCYC is set to 1, and the current number of time steps between updates of the coupling geometry is 4. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

## COSUBMAX

Default = 0, SOL 700 only

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated.

### Example:

```
PARAM,COSUBMAX,10
```

### Remarks:

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values of PARAM,COSUBMAX should be used.

## COUPMASS

Default = -1

COUPMASS > 0 Requests the generation of coupled rather than lumped mass matrices for elements with coupled mass capability, as listed in [Table 3-1](#) in the *MSC Nastran Reference Guide*. This option applies to both structural and nonstructural mass for the following elements: CBAR, CBEAM, CONROD, CQUAD4, CHEXA, CPENTA, CPYRAM, CQUAD8, CROD, CTETRA, CTRIA3, CTRIA6, CTRIAX6, CTUBE.

COUPMASS < 0 (Default) causes the generation of lumped mass matrices (which may include torsion inertia for beam elements, and some coupling if there are beam offsets) for all of the above elements. However, if SYSTEM(414) is greater than zero, (Default = 0) then the lumped mass matrices will contain translational components only for the CBAR and CBEAM elements.

## CP1, CP2

Default = (1.0, 0.0)

The load vectors are generated from the equation

$$\{P_j\} = CP1 \cdot \{P_j^x\} + CP2 \cdot \{P_j^2\}$$

where  $\{P_j^2\}$  is selected via the Case Control command P2G, and  $\{P_j^x\}$  comes from Bulk Data static load entries. These parameters are effective only if P2G is selected in the case control section.

## CQC

Default = -1

See [SCRSPEC](#).

## CURV

Default = -1

PARAM,CURV,1 requests that the CTRIA3 and CQUAD4 element stress and/or strain output be computed in a material coordinate system (normal output is in the element or basic coordinate system) and/or to interpolate it to grid points. (CQUAD4 element corner stress output is not supported.)

The integer parameter OG controls the calculation of stress and/or strain data at grid points. If OG is set to -1, the calculation for stresses and/or strain data at grid points is not performed. The default value of zero provides the calculation of these quantities at those grid points to which the selected elements connect.



User parameters S1G, S1M, S1AG, and S1AM, set to 1, request the printout of stresses at grid points, stresses in the material coordinate system, strains at grid points and strains in the material coordinate system, respectively.

The integer parameter OUTOPT may be set in accordance with the below options to select print, punch, and/or plotter output for stress and/or strain data that are computed in user-defined material coordinate systems.

OUTOPT Value	Description
0	Default-standard MSC Nastran device codes are used.
1	Print only
2	Plot only
4	Punch only

The above values may be combined additively to select two or more forms of output. For example, OUTOPT=6 requests both plot and punch output. Related parameters include BIGER, CURVPLOT, DOPT, NUMOUT, NINTPTS, S1G, S1M.

For stress and/or strain/curvature output in a user-defined material coordinate system MCSID must be defined on MAT1 and MAT2 Bulk Data entries. The values of MCSID reference CORDiR, CORDiC, and CORDiS Bulk Data entries. A value of zero for MCSID does not imply the basic coordinate and will eliminate all elements which reference the MATi from the subject calculations.

1. If these data are requested at the element centers, the program will compute the unit vector  $\hat{i}_m$  along the T1 or x-axis of the material coordinate system, and compare

$$|\bar{n} \cdot \hat{i}_m|$$

for each element that references the material coordinate system, where n is the normal to the surface of the element. If

$$|\bar{n} \cdot \hat{i}_m|^2 \geq .4$$

the projection of the y-axis on the surface of element is taken as the reference axis. Otherwise, the projection of the x-axis on the surface of the element is taken as the reference axis. The angle between the x-axis of the element coordinate system and the projection of the selected reference axis of the material coordinate system is used to transform the stress and/or strain data into the material coordinate system at the element centers.

2. If, on the other hand, the user requests these data at the grid points to which the elements connect the program will interpolate the results from (a) to the grid points to which the elements connect. The parameter NINTPTS=N, the stress and/or strain data at the N closest element centers to the grid point in question will be used in the interpolation. The program may include more than N points in the interpolation if the distance of other element centers is not more than 10% greater than the closest N element centers.

The following specifies the output headings for stresses and/or strains in the material coordinate system.

### Element stresses (PARAM,S1M,1)

1. Available in CQUAD4 and CTRIA3 elements
2. Page headings:  
STRESSES IN QUADRILATERAL ELEMENTS (CQUAD4)  
STRESSES IN TRIANGULAR ELEMENTS (CTRIA3)
3. Under the column FIBER DISTANCE:  
Z1 is replaced by MCSID.  
Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

### Grid point stresses (PARAM,S1G,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements
2. Page heading:  
STRESSES AT GRID POINTS
3. Under the column are:

$$\left\{ \begin{array}{l} \text{MAT1 - COORD1 - ID} \\ \text{PROJ-CODE} \end{array} \right\}$$

Z1 is replaced by MCSID.

Z2=A+10\*N where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

### Element strains (PARAM,S1AM,1)

1. Available for CQUAD4 and CTRIA3 elements
2. Page headings:  
STRAINS IN QUADRILATERAL ELEMENTS (CQUAD4)  
STRAINS IN TRIANGULAR ELEMENTS (CTRIA3)
3. Under the column FIBER DISTANCE:  
Z1 is replaced by MCSID.  
Z2 is replaced by 1.0 if the x-axis of the material coordinate system is selected as the reference axis, and by 2.0 if the y-axis of the material coordinate system is selected as the reference axis.

### Grid point strains (PARAM,S1AG,1 and PARAM,OG,1)

1. Available for CQUAD4 and CTRIA3 elements.

2. Page heading:

$Z2=A+10*N$  where A is 1.0, 2.0, or 3.0, depending on whether the x-, y-, or z-axis of the material coordinate system is most nearly normal to the projected plane of the field of elements involved in the calculation.

#### STRAINS AND CURVATURES AT GRID POINTS

3. Under the column are:

$$\left\{ \begin{array}{l} \text{MAT1 - COORD1 - ID} \\ \text{PROJ-CODE} \end{array} \right\}$$

Z1 is replaced by MCSID.

### CURVPLOT

Default = -1

PARAM,CURVPLOT,1 requests that x-y (or curve) plots whose abscissas are a sequence of grid points and whose ordinates may be displacements, loads, SPC forces, or grid point stresses. To obtain stress plots, set the CURV parameter to +1. The default for DOPT is the length between grid points.

Specify the XYOUTPUT Case Control command in the usual manner, replacing the point ID with the SID of SET1 Bulk Data entries.

The SET1 Bulk Data entries must contain unique SIDs for each set of grid points to be plotted.

User requests for xy-plots of output quantities appear in the Case Control Section in the standard form. For example,

```
.
.
.
OUTPUT (XYOUT)
.
.
XYPLOT DISP 1/4 (T3)
.
.
XYPLOT SPCF 2/5 (T1)
.
.
BEGIN BULK
```

The first XYPLOT command will produce an xy-plot from the displacement output of subcase 1. The abscissa of the curve will reflect the grid point IDs listed on the SET1 entry with an SID of 4, and the ordinate will reflect the T3 component of displacement at these grid points. The second XYPLOT command will produce an xy-plot whose ordinates are the T1 components of the forces of constraint in subcase 2 at the grid points listed on the SET1 entry with an SID of 5.

The user has some degree of control over the scaling of the abscissas on these xy-plots. This control is exercised through the parameter DOPT on a PARAM Bulk Data entry. The legal values of this parameter provide the following scaling options for the abscissas.

Value of DOPT	Scaling for Abscissa
0 (Default)	$\ g_j - g_i\ $
1	$ x_j - x_i $
2	$ y_j - y_i $
3	$ z_j - z_i $
4	1.

Thus, the default value of DOPT will place the first grid point listed on the referenced SET1 entry at the origin, and subsequent grid points will be located along the abscissa at intervals proportional to the distance between that grid point and its predecessor. Values of DOPT equal to 1, 2, or 3 will scale the abscissa so that the interval between adjacent grid points is proportional to the difference in the X, the Y, and the Z components of the subject grid points respectively. DOPT=4 will space the grid points equally along the abscissa.

### CWDIAGP

Default = NO

For CWELD element: prints elements randomly deleted if set to yes.

### CWRANDEL

Default = 0.0

For CWELD element: if not zero, then it specifies as a decimal percent for the number of fasteners to randomly delete.

### DBALL

Default = DBALL

By default, all data to be stored on the database for restart purposes will be located on the DBALL database set (DBset). These parameters permit the storage of some data blocks on DBsets other than DBALL, which are defined by the user and specified on the INIT File Management statement. Any or all of these parameters may be set to SCRATCH in order to reduce overall disk space usage; e.g., PARAM,DBUP,SCRATCH or PARAM,DBALL,SCRATCH. However, automatic restarts will be less efficient because data normally assigned to a permanent DBset will have to be recomputed.

A unique value for each superelement may be specified in the Case Control Section for the parameters DBALL, DBDN, DBRCV, and DBUP. Certain DBsets may be taken offline depending on which phase (see [Summary of Solution Sequence Operations](#) in *the MSC Nastran Reference Guide*) of superelement analysis is being performed. PARAM,DBALL specifies the default value for parameters DBDN, DBUP, and DBRCV.

The DBDN DBset contains data blocks necessary for “downstream” processing. For example, the stiffness, mass, damping, and static loads matrices that have been reduced to the boundary of the superelement are stored in this DBset.

The DBRCV DBset contains data blocks that must be online during the first pass through data recovery (Phase 3). These data blocks are used to recover the total displacement vector  $u_g$  of the superelement. This operation is performed by the SSG3 and SDR1 modules. On subsequent data recovery restarts, this DBset may be taken offline. Its default is determined from the value of DBUP.

The DBUP DBset contains data blocks necessary for “upstream” processing. For example, the geometry and property tables along with the stiffness, mass, damping, and static loads matrices related to the interior grid points of the superelement are stored in this DBset. These matrices and tables must be online during the reduction (Phase 1) and data recovery (Phase 3) of the superelement.

The IFP DBset contains data blocks that are required for all phases of the analysis. These data blocks are related to the entire model; examples are Bulk Data, superelement map, IFP module outputs, and resequenced grid points. This DBset must be online for all runs.

PARAM,DBALL also specifies the default value for PARAMs DBEXT and DSO described below.

### **DBC CONV**

Default = XL

See POST=0.

### **DBC DIAG**

Default = 0

See POST=0.

### **DBC OVWRT**

Default = YES

See POST.

### **DBDICT**

Default = -1

Controls the printout of the database directory at the beginning and end of the run. See DBDICT FMS statement description in Section 2. If DBDICT=0, then the database directory will be printed at the start of the run. If DBDICT=1, then the directory will be printed at the end of the run. If DBDICT $\geq$ 2, then it will be printed at the beginning and end of the run. If DBDICT=-1 (the default), the directory is not printed at the beginning or end of the run.

If multiple versions and/or projects exist on the database, then the parameters DBDRPRJ and DBDRVER allow the user to select the desired project and version, respectively. The appropriate values may be found in the Project/Version Table that is printed upon restart. If DBDRVER=0 (or DBDRPRJ=0), then the current version (or project) is selected. If DBDRPRJ=-1 (or DBDRVER=-1), then all projects (or versions) are selected.

### **DBDN**

Default = value of PARAM,DBALL.

See DBALL.

### **DBDRPRJ**

Default = 0

Specifies the desired project-identification number. See DBDICT.

### **DBDRVER**

Default = 0

Specifies the desired version-identification number. See DBDICT.

### **DBEXT**

Default = value of PARAM,DBALL

Specifies the DBset location to store the external superelement information. External superelement information is generated by the EXTSEOUT Case Control command.

### **DBRCV**

Default = value of PARAM,DBUP.

See DBALL.

### **DBUP**

Default = value of PARAM,DBALL.

See DBALL.

### **DDRMM**

Default = 0

DDRMM is only recognized if PARAM,SPARSEDR,NO is specified.

By default, the matrix method of data recovery is used in the modal transient and frequency response solutions. DDRMM=-1 will force calculation of complete g-set solution vectors by the mode displacement method and is required for FATIGUE analysis using SOL 108 or SOL 111 if PARAM,SPARSEDR,no is also specified.

### **DELCLUMP**

Default = 0.5, SOL 700 only

#### **Format:**

PARAM,DELCLUMP,VALUE

#### **Example:**

PARAM,DELCLUMP,0.1

Prevents small clumps in the Euler mesh from determining the time step and prevents the leakage of small masses to isolated regions.

VALUE Value of DELCLUMP. See Remark 1. (Real  $\geq 0.0$ )

**Remarks:**

1. Material in Eulerian elements of a clump with:  

$$fvunc < DELCLUMP \cdot fblend$$
is eliminated
2. See also parameter FBLEND.

**DESPCH**

Default = 0

For sizing and shape topography optimization, DESPCH specifies in SOL 200 when the optimized (updated) bulk data entries are written to the PUNCH file. Currently, all the property entries, material entries, and connectivity entries that can be designed and DESVAR, DRESP1, and GRID entries can be written. Notice that the DRESP1 entries will be written if only when a mode tracking is performed and the DRESP1 responses have type FREQ or EIGN.

For topology/topometry optimization, DESPCH specifies when the topology optimized element density values (or topometry optimized values) are written to the element result file jobname.des. This file can be directly read in PATRAN or third party post-processor to displace and animate the topology/topometry optimization results.

< 0	Never
= 0	at the last design cycle only (Default)
> 0	at every design cycle that is a multiple of DESPCH and the last design cycle. For example, if n=2 and the maximum number of design cycles is 5 (DESMAX=5 on the DOPTPRM entry), then, DESVAR and GRID entries at design cycle 2, 4, and 5 are written in the punch file.

**DESPCH1**

Default = 6

DESPCH1 specifies in SOL 200 the amount of data to be written to the .pch and .des file. A positive DESPCH1 value request large field formats while a negative value requests small field formats. For a shape optimization job, if DESPCH1 <>0, the updated GRID entries of the whole model will be written in the .pch file.

Descriptions of various DESPCH1 values are given below:

- 0 Write no data.
- ±1 Write the property entries that are designed.
- ±2 Write all the property entries of a given type when one or more property of that type is designed.
- ±4 Write DESVAR and DRESP1 entries.



- $\pm n$  Write combine quantities by summing the DESPCH1 values. For example,  $n=1+4=5$  requests writing all the designed property entries, DESVAR and DRESP1 entries to the .pch file for normal modes analysis.
- $> 0$  Write all (topology designed and non-designed) element density values to the topology element density history file jobname, des.
- $< 0$  Write topological designed element density values to the topology element density history file jobname.des.

## DFREQ

Default =  $10^{-5}$

DFREQ specifies the threshold for the elimination of duplicate frequencies on all FREQ*i* Bulk Data entries. Two frequencies,  $f_1$  and  $f_2$ , are considered duplicated if

$$|f_1 - f_2| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQ*i* entries.

## DIROUT

Default = NO

See CQC under SCRSPEC.

## DOPT

Default = 0

See CURVPLOT.

## DPEPS

Default = 1.0E-4

In SOL 200, if any difference between the property value on the property entries and the value calculated from the design variable values on the DESVAR entry (through DVCREL*i*, DVMREL*i*, DVPREL*i* relations) is greater than DPEPS, the design model values override the analysis values. If all the differences are less than DPEPS, analysis results from a previous run are accepted in a subsequent sensitivity/optimization task, thereby avoiding a reanalysis. The PTOL parameter on the DOPTPRM entry is a related parameter that checks the maximum difference.

## DPHFLG

Integer  $\geq 0$ ; Default = 0

Select the method for performing eigenvector sensitivity analysis. Unless repeated eigenvalues are anticipated, the default value is recommended.



DPHFLG	0	Nelson's method (Default)
	1	Subspace iteration method. Recommended when there are repeated roots.
	2	High's method, # of modes for iteration = $\min(2n, n+8, m)$ n: the highest constrained mode m: the number of modes request by EIGR
	3	High's method will all modes requested by EIGR used in the iterator. DPHFLG = 2 and 3 have further control parameters
ITERATE	yes	Do iteration, for improved sensitivity value. (Default)
	no	No iteration, equivalent to Fox's method and provides the most efficient method, but not the most accurate.
ITMAX		Maximum number of iteration. (Default = 10)
TOL		Tolerance for convergence in iteration. (Default = $1.0e-4$ )
LAMBDAS		Shift factor. (Default = 0.0)
KORTHO	no	Use mass for Gram Schmidt orthogonalization. (Default)
	yes	Use stiffness K for Gram Schmidt orthogonalization.
ITRPRNT	no	Do not print sensitivity for each iteration. (Default)
	yes	Print sensitivity for each iteration.
ITFPRNT	no	Do not print final sensitivity, leave print to SOL 200.
	yes	Print final sensitivity inside High's method computation
MDOF	no	Do not reduce DPHI to USET 'U6' DOF. (Default)
	yes	Reduce DPHI to USET 'U6' DOF.

## DSNOKD

Default = 0.0

DSNOKD specifies a scale factor to the differential stiffness matrix in buckling design sensitivity analysis. If  $DSNOKD > 0.0$ , the effect of the differential stiffness matrix is included in buckling the design sensitivity analysis.

If `PARAM,DSNOKD,1.0` is specified in SOL 200, the differential stiffness sensitivity calculation is performed more accurately; i.e., the change in the stiffness matrix due to the changes in the displacements are computed. However, the calculation is more expensive than with `PARAM,DSNOKD,0.0`.

Non-zero values of `PARAM,DSNOKD` cannot be used in SOL 200 with multiple buckling design subcases less each subcase contains the same `STATSUB` command.

## DSO

Default = value of `PARAM,DBALL`

Specifies the DBset location to store datablocks created for design sensitivity and optimization in SOL 200.

## DSZERO

Default = 0.0

DSZERO specifies the minimum absolute value for the printout of design sensitivities.

## DV3PASS

Integer, Default = 1 for SOL 200 with nonlinear property sensitivity evaluation.

DV3PASS controls the number of optimizer internal cycles must pass before the re-evaluation of sensitivity of nonlinear properties, such as I1/I2/J of PBARL/PBEAML/PBRSECT/PBMSECT. With default value of 1, the sensitivity of nonlinear properties is computed for every internal cycle of optimizer. Setting DV3PASS to 10 causes the sensitivity to be evaluated at optimizer internal cycles of 1, 11, 21 and etc. The sensitivity of nonlinear properties is considered as invariants if a sufficiently large number, such as 100, is assigned to DV3PASS.

## DYNSPCF

Default = NEW

PARAM,DYNSPCF,NEW requests that mass and damping coupled to ground be included in the SPCForce calculations for the linear dynamic solutions: SOLs 103, 107 through 112, 115, 118, 145, 146, and 200. OLD neglects these effects and gives the same SPCForce results obtained in versions prior to Version 68.

**Note:**

Some versions of Dytran have trouble with permanent constraints and issue a bogus error message in the d3hsp file. This is the case for MSC Nastran 2005 r2.

## EIGFILT

Default=1.0E-13

Threshold for zero-frequency eigenvalue for ACMS VERSION=NEW (module ACMS1).

## ENFMETH

Default = TOTAL or ABS

This parameter controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static-based solution. The default value of TOTAL/ABS implies that the program solves directly, *in one step*, for the TOTAL solution of the dynamic analysis which includes both the static-based solution and the dynamic solution that is relative to the static-based solution. If the value is specified as REL, then the program obtains the total solution of the dynamic analysis *in two steps*, by first solving for the static-based solution and then solving separately for the dynamic solution RELATIVE to the static-based solution. In general, the TOTAL/ABS and REL solution methods both yield essentially the same results.

The TOTAL/ABS solution method is computationally more efficient. This is also the only method that is meaningful and that should be employed when a problem involves the use of NOLINi or NLRGAP entries. An important point to note regarding this method is that, for modal dynamic analysis, residual vectors are absolutely critical in order for this method to get correct answers.

The REL solution method, though less efficient, may be more accurate for transient solutions and for modal frequency response solutions at very low forcing frequencies. Also, for modal dynamic analysis, this method is not as critically dependent on residual vectors as the TOTAL/ABS solution method. In addition, the current implementation of the REL solution method does not support fluid DOFs with enforced motion. So, if fluid DOF are among the DOFs with enforced motion, the job will be switched to the ABS solution method automatically.

It should be emphasized here that PARAM,ENFMETH and PARAM,ENFMOTN are completely separate, distinct and independent of each other and should not to be confused with each other. The former controls the *solution method* when dynamic enforced motion analysis via SPC/SPCD is used while the latter controls how the results of such an analysis are *output*.

### ENFMOTN

Default = TOTAL or ABS

This parameter controls how the results of the analysis are *output* when dynamic enforced motion analysis via SPC/SPCD is used in SOLs 108, 109, 111, 112, 146 and 200.

The total solution of a dynamic enforced motion analysis using SPC/SPCD can be regarded as a combination of a static enforced motion solution (similar to what is done in SOL 101) and a dynamic enforced motion solution that is relative to this static solution. The default value of TOTAL/ABS implies that the output results of the analysis represent the TOTAL solution of the dynamic analysis which includes both the static solution and the dynamic solution that is relative to this static solution. If the value is specified as REL, then the output results represent the dynamic solution RELATIVE to the static solution.

It should be noted that, in general, the static solution mentioned above may *not* be a stress-free solution. Therefore, in general, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL in an SPC/SPCD enforced motion analysis may give *different results not only for displacements, but also for stresses*.

In general, it is not easy to identify all cases wherein the static solution represents a stress-free solution. However, one can give examples of such cases.

Thus, for instance, if the SPC/SPCD enforced motion is specified at a single grid point and there are no constraints at any other grid point, then the static solution will be a stress-free solution.

Similarly, if the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the enforced motion is the same for all of the enforced motion points and there are no other constraints in the model, then all of the enforced motion points will move in unison and the static solution will be a stress-free solution.

In contrast to the above, consider the situation in which the SPC/SPCD enforced motion is specified at more than one grid point for a specific grid point component, but the magnitude and direction of the specified enforced motion is *different* for the various enforced motion points. In this case, the static solution will *not* be a stress-free solution.

If the static solution represents a stress-free solution, then it can be regarded as the base motion of the model. In such cases, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different* displacements, but the same stresses. Further, in modal dynamic analysis for such cases, the results using PARAM,ENFMOTN,TOTAL (or ABS) will give the *same* results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes included and the results using PARAM,ENFMOTN,REL will give the same results as those obtained by using the large mass approach with all zero (or nearly zero) frequency modes excluded.

If the static solution does *not* represent a stress-free solution, the usage of PARAM,ENFMOTN,TOTAL (or ABS) and PARAM,ENFMOTN,REL will give *different results not only for displacements, but also for stresses*. In this case, the results from the usage of PARAM,ENFMOTN,REL do not have the same significance as in the case of the stress-free static solution except for the fact that these results represent the dynamic solution RELATIVE to the static solution.

**Note:** Currently, the usage of PARAM,ENFMOTN,REL is not supported when PARAM,ENFMETH,ABS is specified (or implied). If this usage is employed, the program terminates the execution with an appropriate fatal message.

## EPVRT

Default = 1.E-8

EPVRT is analogous to and described under the EPSSING keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPVRT.

## EPSILON

Default = SECANT

In nonlinear statics (SOL 106 and SOL 400), thermal loads are computed using the secant method:

$$\varepsilon_T = \alpha_{\text{load}}(T_{\text{load}} - T_{\text{ref}}) - \alpha_{\text{init}}(T_{\text{init}} - T_{\text{ref}})$$

If the parameter is set to INTEGRAL, thermal loads are computed using the integral method:

$$\varepsilon_T = \int_{T_{\text{init}}}^{T_{\text{load}}} \alpha(T) dT$$

This parameter only applies to SOLs 106 and 400 ANALYSIS=NLSTAT, and only applies to CQUAD4, CTRIA3, CQUADR, and CTRIAR elements that are not associated with the enhanced nonlinear materials (PSHLN1).

## EPZERO

Default = 1.E-8

EPZERO is analogous to and described under the EPS keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,EPZERO.

## ERPC

Default = 1.0

This parameter represents the phase speed of the fluid for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERPREFDB

Default = 1.0

This parameter represents a peak reference ERP value used in the computation of ERP in units of dB. The dB level is defined as:

$$\text{ERPdB} = 10\log\left(\text{RHOCP} \cdot \frac{\text{ERP}}{\text{ERPREFDB}}\right)$$

This parameter may alternatively be set using the ERP Case Control command.

## ERPRHO

Default = 1.0

This parameter represents the fluid density for Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERPRLF

Default = 1.0

This parameter represents a Radiation Loss Factor scale factor for the Equivalent Radiated Power (ERP) analysis. This parameter may alternatively be set using the ERP Case Control command.

## ERROR

Default = -1

For SOLs 111-112, when the constraint modes have non-zero generalized force the parameter ERROR set to 0 causes the fatal error exit to be branched over and the dynamic response is computed and output. This option is intended for model checkout work, where bad results are better for diagnostic work than no results at all. This parameter is a traditional feature also used in other, similar circumstances.

## ESLFSAV

Character, Default = NO

ESLFSAV = YES requests that all the intermediate files from an ESLNRO job be saved on disk. The destination of these files can be directed with the 'sdir=' option on a Nastran submittal command line.

## ESLMOVE

Integer, Default = 0

ESLMOVE = 0 selects a move limit scheme that poses restrict lower and upper bounds on design variables during the linear response optimization. The range of the bounds is determined by

$$X_k^L = \max(X_o^L, X_{k-1} - \text{MOVE})$$

$$X_k^U = \min(X_o^U, X_{k-1} + \text{MOVE})$$

$$\text{MOVE} = \max(\text{DXMIN}, \text{abs}(X_{k-1}) \cdot \text{DELXESL})$$

ESLMOVE = 1 selects a move limit scheme that scales back the design move proposed from a linear response optimization. The amount of scaling back is determined by

$$X_k^* = X_{k-1} + (X_{k-1}^1 - X_{k-1}) \cdot \text{DELXESL}$$

where  $X_k^*$  is the scaled design variable for the k-th design cycle,  $X_{k-1}$  is the design variable at (k-1)th design cycle,  $X_{k-1}^1$  is the proposed design from the linear optimization solution at (k-1)th design cycle.

Parameters DXMIN and DELXESL can be specified on the DOPTPRM entry.

## ESLLCOMP

Default = No

ESLLCOMP selects types of compliance response to be included in the design task. The nonlinear compliance response is defined using a DRESP1 entry with RTYPE=COMP for the ESLNRO topology optimization tasks. As the default, it is computed by the product of the applied nonlinear loads and corresponding nonlinear displacement components. Alternatively, ESLLCOMP=YES selects a linear compliance response that computed as the total work done by the equivalent static loads on the linear system.

## ESLMPC1

Default = 0

This parameter applies only to the ESLNRO jobs with 3D contact. Its default has different meanings depending on the type of contact applications. As the default, for a glued contact ESLNRO job, a linear response optimization task will include a set of MPC entries that are created from the nonlinear analysis. For a touching contact ESLNRO job, the linear response optimization task will not include the MPC entries by default. Setting ESLMPC1 to a positive number will turn on the MPC inclusion.

- 1 uses the MPC entries created from the nonlinear analysis at the converged nonlinear analysis.
- 2 uses the MPC entries created at the beginning of the very first nonlinear analysis.

## ESLOPTEX

Default = 0

This parameter allows the user to perform an ESLNRO job at a targeted exit point. The allowable values of ESLOPTX are listed below with their description.

- 0 Do not exit. Proceed with ESLNRO nonlinear response optimization.
- 1 Exit after the initialization of the analysis and design model but before nonlinear FE analysis begins.
- 2 Exit after nonlinear FE analysis ends.
- 3 Exit after design constraint evaluation and screening.

## ESLPRT

Default = 0

ESLPRT specifies how often the ESLNRO results are printed in the .f06 file and saved in the .xdb file. By default, the program will print the results to the .f06 file at the first and the last design cycles and save the results to .xdb (or .op2) at the first and last design cycles on the disk. (See ESLPRT1 for selection of result contents.)

- > 0 then the results are printed at the first design cycle; at every design cycle that is a multiplier of ESLPRT; and the last design cycle.
- < 0 the no results are printed and saved.

## ESLPRT1

Default = 7

ESLPRT1 specifies what type of results to be written to the .f06 and to .xdb (or .op2). It may take any of the following base values or a combination of these base values:

- 0 write no data
- 1 write the nonlinear analysis results to the .f06 file.
- 2 write the optimization data controlled by P1 and P2 to the .f06 file.
- 4 save the nonlinear analysis results to the .xdb (or .op2) file.
- 8 save the linear response optimization results to the .xdb (or .op2) file.

For example, by default, results from the nonlinear analysis, the optimization data will be written to the .f06 file and result data will be written to .xdb or .op2.

## ESLRCF,user\_rc\_file

Character\*8, must be lower case. Default = blank

ESLRCF, user\_rc\_file allows a user to define a custom RC file for the internally spawned jobs.

### Example:

```
PARAM,ESLRCF,myrc
```

The contents of the *myrc* file are:

```
mem=200m
exe=-local_path/MSCNASTRAN
del=-local_path/SSS
```

The example shows the user-defined RC file, myrc, specifies its own memory allocation and its local Nastran executable and local DMAP database.

## ESLTOPCV

Default = 80

ESLTOPCV specifies an alternate convergence tolerance for an ESLNRO's topology optimization job. An ESLNRO's topology optimization job will be terminated when the percentage of design variables whose changes are less than a given value, CONV DV specified on the DOPTPRM Bulk Data entry is greater than ESLTOPCV.

## ESLUNT2

Default = 54

ESLUNT2 is a file unit number used by the nonlinear analysis and the linear response optimization runs in an ESLNRO's topology optimization job for the access of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

## ESLUNT1

Default = 53

ESLUNT1 is a file unit number used by the linear response optimization run in an ESLNRO's topology optimization job for the storage of design variables and designed properties. It only needs to be changed if there is a conflict with the default.

## EST

Replaced by the ELSUM Case Control command.

## EULBND

Default = ELEMENT, SOL 700 only

Defines boundary treatment for Euler boundaries.

### Format:

PARAM,EULBND,option

- |          |   |
|----------|---|
| EXTRAPOL | The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character) |
| ELEMENT  | The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element. (Character)                                  |



**Remarks:**

1. The finite volume representation in general assumes that element values are constant within each element. While this assumption is adequate for the large majority of applications, fluid models involving hydrostatic pressure gradients require that the pressure gradient be also recognized to exist within the element. When element-internal hydrostatic gradients are not accounted for, the calculation will be less accurate and will suffer from numerical symptoms like pair forming of element pressures. By activating the option EXTRAPOL hydrostatic gradients inside the element are taken into account. For meshes without bias, option EXTRAPOL only modifies the numerical schemes along the boundary.
2. When coupling surfaces are used DYPARAM,FASTCOUP has to be activated as well.

**EULSTRES**

Default = VOLUME, SOL 700 only

Defines the update logic for stresses when material is transported in Euler elements.

**Format:**

PARAM,EULSTRES,option

- VOLUME The pressure that a wall or coupling surface exerts on the adjacent Euler element is obtained from extrapolating the element pressure toward this boundary. (Character)
- MASS The pressure that a wall or coupling surface exerts on the adjacent Euler element equals the pressure inside this element.

**Remarks:**

1. Only used for the MMSTREN solver.
2. Stresses are a material property and when material flows in or out an element the stress state in the element is changed. This is analogous to temperature and energy. Not the temperature is transported, but energy. After transporting energy the temperature is re-computed by dividing the energy by element mass and specific heat. In case of stress the “energy” is given by mass times stress. After transporting this “energy” the new stress follows by dividing it by mass. As shown in Chapter 6 of the Theory Manual this gives a correct updating procedure for stresses. There it also proven that stress times mass is conserved during transport.
3. In most simulations variations in density are small and the multiplication by mass can be replaced by a multiplication by volume. This method is activated by option VOLUME which is the default option. Using the MASS option may have some influence on simulations with large density variations. The option MASS will give the most accurate results.
4. The transport logic of the effective plastic strain is identical to that of stresses. When using option MASS the plastic strain is computed more accurately when material is compressed.
5. The (single material) Euler with strength solver makes use of the multiplication by mass. The multiplication by volume is not implemented for this solver.

## EUSUBCYC

Default = 1, SOL 700 only

Controls the maximum growth of the subcycling interval in Euler computations.

### Example:

```
PARAM,EUSUBCYC,2
```

### Remarks:

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If the solver estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

## EXCLUDE

Default = 0.

PARAM, EXCLUDE, Integer. Add PARAM, EXCLUDE, n in the buckling subcase(s) to activate linear buckling with local exclusion of differential stiffness. If n is positive, it selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be ignored. If n is negative, the absolute value selects the SET1 entry ID in the bulk data that defines the list of GRID points for which differential stiffness will be retained.

To select the GRID points care must be exercised by the user which follow natural mesh lines as much as possible, otherwise an element may be connected to some GRID points that have differential stiffness and some GRID points that do not. Sometimes this is impossible to ensure, and so the transition zone between retained-excluded differential stiffness should not be in, or close to, a region of interest. Selecting stiff (unlikely to buckle) zones for the transition zones is also advised.

While, EXCLUDE, can occur in either Case Control or Bulk Data, for multiple buckling subcases, it must occur in Case Control.

SOL105, is a linearization of a nonlinear behavior, therefore it is recommended that parts of the structure, for which buckling is important, are in the residual or SE=0. However, if the user puts the parts of the structure for which buckling is to be considered in a BEGIN BULK SUPER=n; then the sets must be in the Part superelement or Module.

Multiple buckling subcases with different exclude values are allowed. For example:

```
DISPLACEMENT (PLOT, SORT1, REAL) =ALL
SUBCASE 1
  SUBTITLE=Static Load
  SPC = 2
  LOAD = 2
  analysis=statics
subcase 2
  SUBTITLE = Buckling Subcase
```

```

      SPC=2
      statsub=1
      analysis=buck
      METHOD=105
      param,exclude,-1
subcase 3
      SUBTITLE = Buckling Subcase
      SPC=2
      analysis=buck
      statsub=1
      METHOD=105
      param,exclude,-2
BEGIN BULK

```

**EXTDR**

Deprecated. See the EXTDRIN Case Control command description.

**EXTDROUT**

Deprecated. See the EXTDROUT Case Control command description.

**EXTDRUNT**

Deprecated. See the EXTDROUT Case Control command description.

**EXTOUT**

Deprecated. See the EXTSEOUT Case Control command description.

**EXTRCV**

Deprecated. See the EXTDRIN Case Control command description.

**EXTUNIT**

Default = 30.

EXTUNIT specifies the Fortran unit number for an external superelement stored on an op2 or op4 file and is required if the default value of 30 is not desirable and a CSUPER Bulk Data entry is used to define the external superelement in an assembly run. EXTUNIT is also required if it references an op4 file in which case the value for the unit number must be negative.

**FACTOR**

Default = 10000

See OLDSEQ.

**FASTFR**

Default = AUTO

In MSC Nastran 2004, the FASTFR method was introduced for modal frequency response analysis. It can be selected via the Bulk Data entry, PARAM,FASTFR,YES and shows significant performance improvement for certain models in the mid-frequency range. PARAM,FASTFR,NO deselects the FASTFR method causing the program to use the standard method for modal frequency analysis.

By default, the program will decide automatically which solution method will be most efficient for the frequency response part in a SOL 111 analysis. Based on the size of the modal space and some other heuristic criteria, either the FASTFR solution method will be run, or the FRRD1 module with or without the iterative solver will be used.

Limitations for the FASTFR method

1. The FASTFR method works only for SOL 111 and for SOL 200 with ANALYSIS=MFREQ.
2. SESDAMP and FASTFR are not allowed in the same run.
3. To force the iterative solver in FRRD1, FASTFR=yes is required.

## FBATOLR

Default = 1.0E-05

This is the tolerance that is applied to grid point coordinates in order to determine connections between potential connection grid points of various FRF components in the FBA process.

The default tolerance should be satisfactory for most situations. A looser tolerance may be needed in certain situations. An example is the case where the potential connection points of an FRF component are associated with the shell elements of RSSCON solid-to-shell element connectors. In this case, a looser tolerance may need to be specified in order to achieve proper connections between FRF components in the FBA process.

## FBLEND

Default = 0.6667, SOL 700 only

Eulerian elements with uncovered fractions smaller than FBLEND are blended with adjacent elements to form a clump so that they do not control the time step.

### Format:

PARAM,FBLEND,VALUE

### Example:

PARAM,FBLEND,0.5

VALUE            The uncovered fraction below which blending occurs. ( $0.0 \leq \text{Real} < 1.0$ )

### Remarks:

1. The default value is satisfactory for virtually all calculations.
2. Elements are blended only if they would have controlled the time step otherwise.
3. Elements with uncovered fractions greater than FBLEND are not blended and are allowed to control the time step.
4. Large values of FBLEND produce a larger time step but many blends. Small values produce a smaller time step and fewer blends.
5. In a calculation with a coupling surface, STEPFACT is smaller or equal FBLEND to avoid instabilities (see PARAM,STEPFACT).

## FDRLDS

Default = YES

PARAM, FDRLDS, YES turns on the logic to generate additional residual vectors for SOL 111 based on the differences of structural and structural damping matrices. For free-free models, a SUPORT entry may improve the free-free modes and yield additional augmented eigenvalues (PARAM, BAILOUT, -1 may occasionally be needed even for a good static SUPORT set). PARAM, FDRLDS, NO can be used to turn off the generation of frequency dependent residual vectors.

## FIRSTKI

PARAM, FIRSTKI, Integer

This parameter is specified when external superelements are used in assembly models of aircraft. This lists the location of the first k-set point of the first CAEROi point related to each superelement, SEID.

For example, if superelement 10 (SEID10) contains CAERO1 point 1001, and the first DOF related to this CAERO1 point is at internal location 51 in the k-set, then this information can be obtained using "PARAM, USETPRT, 0". This information is then added in the input .dat file using "PARAM, FIRSTKI, 51" after "BEGIN SUPER=10" line.

## FIXEDB

Default = 0

FIXEDB is used to reduce the cost of superelement checkout.

-2	(SOL 101 only) is used on the initial runs when the user suspects that the superelement may contain errors and that only operations necessary for fixed-boundary solutions need be performed. In particular, the generation of the $[G_{oa}]$ matrix is branched over in the SEKR operation and $[P_a]$ is not generated in the SELR operation. These operations typically result in 50% of the reduction cost and are not needed in the fixed-boundary data recovery operations described in the next paragraph. After this operation has been completed, the keyword SELANG will appear in the database dictionary, indicating that the $[P_a]$ stored there is incomplete, and should not be summed into the downstream superelement, because System Fatal Message 4252 will be issued.
$\leq -1$	(SOLs 101 and 103 only) allows uncoupled solutions for any superelement with conventional output requests. This output may be obtained as soon as the superelement is successfully generated and reduced and does not require that the entire model be assembled. In superelement statics, the solution is the component due to the $\{u_o^o\}$ vector, i.e., a fixed-boundary solution. In superelement modes, the solution is the uncoupled eigenvectors of the component. If PARAM, FIXEDB, -1 is specified in the Bulk Data or in the residual structure subcase, the modes of the residual structure will not be computed. For a printout or plotting of the component mode eigenvectors it is recommended that PARAM, FIXEDB, -1 be specified in the Bulk Data Section or above the subcase level in Case Control. If the modes of the residual structure are desired, then PARAM, FIXEDB, 0 should be specified in the residual structure subcase. Exterior degrees-of-freedom listed on SECSETi and SESUP entries are free, and those on SEBSETi degrees-of-freedom are fixed. Data recovery for the residual structure should not be requested for this option.
+1	(SOL 101 only) is used after the superelement has been proven valid. In the SEKR and SELR operations, it provides a branch over all operations already completed in the SEKR and SELR phases and completes the generation of the $[G_{oa}]$ matrix and the boundary stiffness and load matrices. It is also a method to recover the factor of the $[K_{oo}]$ matrix if the run aborted while computing $[G_{oa}]$ .

## FKSYMFAC

Default = 0.024

Follower force stiffness is typically unsymmetric, but in some cases making it symmetric may improve convergence in nonlinear problems, whereas in other cases it may hinder it.

FKSYMFAC controls the symmetrization of the follower force stiffness in SOL 106 and SOL 400. If FKSYMFAC = 1.0 is specified, the follower force stiffness  $K_f$  is symmetrized as:

$$K_{fs} = \frac{1}{2}(K_f + K_f^T)$$

and the symmetric part  $K_{fs}$  is used for efficiency. If FKSYMFAC= 0. is specified, the original follower force stiffness  $K_f$  is used. If a value of  $0. < \text{FKSYMFAC} < 1.$  is specified, the non-symmetric part of the follower force stiffness is calculated as:

$$K_{fn} = K_f - K_{fs}$$

and the ratio of unsymmetry:

$$r = \frac{\|K_{fn}\|}{\|K_f\|}$$

is compared with the user specified value of FKSYMFAC. The norm  $\| \cdot \|$  is the absolute maximum number of the matrix.

If  $r < \text{FKSYMFAC}$ , the symmetric stiffness  $K_{fs}$  is used.

If  $r > \text{FKSYMFAC}$ , the original unsymmetric stiffness  $K_f$  is used.

The default value for FKSYMFAC was determined by a parametric study and for most cases this will make the follower force stiffness symmetric, which will give sufficiently accurate answers. The asymmetry ratio(r) increases as geometric nonlinearity intensifies so when it becomes greater than FKSYMFAC, the unsymmetric follower force stiffness is used.

It is considered important to retain the full unsymmetric follower force stiffness, set FKSYMFAC to zero.

The parameter FKSYMFAC is applicable to SOL 106 and SOL 400 only, all other solution sequences symmetrize the follower force stiffness. See parameter FOLLOWK for a list of solution sequences which calculate the follower force stiffness.

## FLEXINCR

Default = NO

In SOL 144, a value of YES will cause the TRIM subcases to be ignored. Instead, the TRIM Bulk Data will be used to obtain the set of Mach, Dynamic pressure and symmetry values for Unit Solutions (Flexible

Increments). These data can be archived in the aeroelastic database for subsequent reuse. (Flexible Increments are always computed. This param merely avoids the TRIM subcase if these increments are all that is required.)

## FLUIDMP

Replaced by options on the Case Control command [FLSPOUT \(Case\)](#).

## FLUIDNE

Default = 500

The Householder method of eigenvalue extraction is more reliable and is automatically selected for the fluid's system modes if the acoustic cavity is defined in a superelement and there exists fluid boundary points. The switch to Householder occurs if the number of estimated fluid modes is less than or equal to the value specified by PARAMeter FLUIDNE.

## FLUIDSE

Default = 0

PARAM,FLUIDSE,seidf specifies a special superelement reserved for fluid elements. Frequency dependent fluid elements must still be in the residual. Partitioned superelements are not supported.

## FMULTI

Default = .10, SOL 700 only

Defines the dimension of the multimaterial element array.

### Format:

PARAM,FMULTI,VALUE

### Example:

PARAM,FMULTI,,25

VALUE            The relative amount of multimaterial elements. ( $0.0 < \text{Real} < 1.0$ )

### Remark:

The multimaterial Eulerian elements use an overflow array in which to store material data. This array can hold FMULTI times the total number of Eulerian elements. In a problem where more than 10% of the elements have more than one material, the default value of FMULTI must be increased.

## FOLLOWK

Default = YES

In SOLs 101,103, 105, 106, 107, 108, 109, 110, 111, 112, 115, 116, and SOL 400, FOLLOWK=YES (Default) requests the inclusion of follower force stiffness in the differential stiffness. FOLLOWK=NO requests that the follower force stiffness not be included. For FOLLOWK=YES in SOLs 101,103, 105, 107, 108, 109, 110, 111, 112, 115, and 116, a separate static subcase is required and the STATSUB command is also required in the eigenvalue subcase. In nonlinear analysis (SOL 106 and 400), the follower force is

included if PARAM,LGDISP,1 is specified. FOLLOWK is ignored in SOL 106 and SOL 400 if LGDISP is not specified.

## FRQDEPO

Default = NO

By default, frequency-dependent elements cannot be connected to o-set degrees-of-freedom. PARAM,FRQDEPO,YES allows frequency-dependent elements to be connected to o-set degrees-of-freedom. However, results may not be reliable.

## FULLSEDR

Default = NO

In a run with superelements, PARAM,FULLSEDR,YES will merge results (DISPL, STRESS, etc.) from all of superelements into a single result as if the run contained no superelements. This is not supported for BEGIN BULK superelements (parts) unless the element and grid identification numbers are unique across all part superelements and the residual structure. Additionally, if there are part SE present, then FULLSEDR has to be specified in all Bulk Data sections (BEGIN BULK and BEGIN SUPER) or in the Case Control.

## FZERO

Default = 1.0

See AUTOSPRT.

## G, GFL

Default = 0.0

G and GFL specify the uniform structural and fluid-damping coefficient in the formulation of dynamics problems. In coupled fluid-structure analysis, G is applied to the structural portion of the model and GFL to the fluid portion of the model. To obtain the value for the parameter G or GFL, multiply the critical damping ratio,  $C/C_o$ , by 2.0. PARAM,G and GFL are not recommended for use in hydroelastic or heat-transfer problems. If PARAM,G (or GFL) is used in transient analysis, PARAM,W3 (or W3FL) must be greater than zero or PARAM,G (or GFL) will be ignored. See [Formulation of Dynamic Equations in SubDMAP GMA](#) in *the MSC Nastran Reference Guide*.

In frequency response and complex eigenvalue analyses, the use of G or GFL will cause the stiffness matrix corresponding to the structural or fluid portion of the model to become complex. The user should be aware that this will, in general, require more memory and resources for the calculations.

## GEOMU

Default = 40

See POST=0.

## GPECT

Default = -1



GPECT controls the printout of all elements connected to each grid point. GPECT=+1 requests the printout. In superelement analysis, the list is printed if PARAM,CHECKOUT,YES is specified or the SEMG or SEALL Case Control command selects the superelement. GPECT=-1 suppresses the printout.

## GRADMESH

Default = OFF, SOL 700 only

Glues fine meshes to coarse meshes. See the section on Graded meshes in the user manual for further information.

### Format:

PARAM,GRADMESH,OPTION

### Example:

PARAM,GRADMESH,MINVOL

OPTION	OFF	Graded mesh gluing is not used (Character)
	MINVOL	If an element of one mesh is covered by an element of another mesh the element with the largest volume will be inactivated. It will also be removed from the output request for Eulerian archives.
	ELNUM	If an element of one mesh is covered by an element of another mesh the element with the smallest element number will be inactivated. It will also be removed from the output request for Eulerian archives.

### Remarks:

1. This parameter can be used to build block-structured meshes.
2. All Euler elements have to be either orthogonal or axial symmetric.
3. To get meaningful physical results, the change in mesh size going from one element to the next should not be larger than 1.4 or smaller than 0.7.

## GRAVSET

Default = 0

A PARAM, GRAVSET, n where n is a Bulk Data SET1 entry, allows the user to apply gravity loading (GRAV) to just a portion of the structure. GRAVSET like GRAV loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the gravity loading is to be applied. GRAVSET can appear in either the **Bulk Data** or **ABOVE** or **IN the 1st Subcase**. A GRAVSET appearing in any other Subcase will be ignored and may cause incorrect results. A GRAVSET appearing in or above the first subcase takes precedence over a GRAVSET appearing in Bulk Data. See also the companion RFORSET entry. If there is no GRAV loading called out, then PARAM, GRAVSET, n needs to be removed from the run

## GRDPNT

Default = -1

GRDPNT > -1 will cause the grid point weight generator to be executed. The default value (GRDPNT = -1) suppresses the computation and output of this data. GRDPNT specifies the identification number of the grid point to be used as a reference point. If GRDPNT = 0 or is not a defined grid point, the reference point is taken as the origin of the basic coordinate system. All fluid-related masses and masses on scalar points are ignored. The following weight and balance information is automatically printed following the execution of the grid point weight generator.

- Reference point.
- Rigid body mass matrix [MO] relative to the reference point in the basic coordinate system.
- Transformation matrix [S] from the basic coordinate system to principal mass axes.
- Principal masses (mass) and associated centers of gravity (X-C.G., Y-C.G., Z-C.G.).
- Inertia matrix I(S) about the center of gravity relative to the principal mass axes. Note: Change the signs of the off-diagonal terms to produce the “inertia tensor.”
- Principal inertias I(Q) about the center of gravity.
- Transformation matrix [Q] between S-axes and Q-axes. The columns of [Q] are the unit direction vectors for the corresponding principal inertias.

In superelement static or geometric nonlinear analysis, GRDPNT > -1 also specifies the grid point to be used in computing resultants, in the basic coordinate system, of external loads and single point constraint forces applied to each superelement. If GRDPNT is not a grid point (including the default value of -1), then the resultants are computed about the origin of the basic coordinate system. In superelement analysis, weights and resultants are computed for each superelement without the effects of its upstream superelements.

For axisymmetric elements, the GRDPNT output may be misleading as the algorithm is designed for 3D elements. For the TRIAX6 element, the mass is for the entire model and the center of gravity and inertias are for the cross section in the x-z plane. For the hyperelastic TRIAX and QUADX elements, the mass is for one radian and the center of gravity and inertias are for the cross section in the x-y plane. For the harmonic TRIAX and QUADX elements, the mass is for the entire model and the center of gravity and inertias are for the cross section in the element coordinate system x-y plane.

## GUSTAERO

Default = 1

If gust loads are to be computed, for example on restart, set GUSTAERO to -1. The default is recommended if no gust loads are to be computed after the flutter analysis.

## GYROAVG

Default = 0

Used to specify one of two formulations for frequency response analysis using the rotor dynamic capability. The default is to determine any frequency-dependent terms for each frequency. This option activates the frequency-dependent looping option. Setting the value < 0 uses an ‘average’ frequency formulation. This option avoids using the frequency-dependent looping and results in a shorter execution time. For this option, PARAM,WR3 and PARAM,WR4 must be specified to include rotor damping.

## HEATCMD

Character\*16, Default=nastran

Name of a command to run SOL 600 thermal contact runs. Nastran first sets up an Marc run to determine the thermal contact conditions which are output in a file named `jid.nthcnt`. Next, Nastran converts these to standard Nastran thermal elements, and finally spawns a second Nastran job from the primary Nastran job. The command to run the second Nastran job is provided using this parameter. For example, if `nast2005t1` is desired, enter `CMD=nast2005t1`. If the command “Nastran” is desired, either leave the parameter out or enter “nastran”. The MSC Nastran run to be spawned will have the form:

```
CMD jid.nast.dat rcf=RCF
```

Where file RCF depends on PARAM,MARHEATM.

### Remarks:

1. See PARAM,MRSPAWN2 for structural analysis.
2. CMD will be converted to lower case regardless of the case entered.

## HEATSTAT

Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

```
TEMP(LOAD) = Heat Transfer Subcase ID
```

If this default is not acceptable, then in heat transfer subcase add the Case Control word TSTRU=SID and in structures subcase here

```
TEMP(LOAD) = SID
```

See the Case Control command, [TSTRU \(Case\)](#). PARAM,NESET is no longer used.

## HFREQ, HFREQFL

Default = 1.+30

The parameters LFREQ, HFREQ, LFREQFL, and HFREQFL specify the frequency range in cycles per unit time of the modes to be used in the modal formulations. (LFREQ and LFREQFL are the lower limits and HFREQ and HFREQFL are the upper limits.) In coupled fluid-structure analysis, HFREQ and LFREQ are applied to the structural portion of the model and HFREQFL and LFREQFL are applied to fluid portion of the model. The default for HFREQ and HFREQFL will usually include all vectors computed. Related parameters are LMODES and LMODESFL.

**Note:** If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

## HTOCITS

Default = 20 (SOL 106 only)

HTOCITS sets the maximum allowable iterations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

## HTOCPRT

Default = NO (SOL 106 only)

PARAM,HTOCPRT,YES requests the printout of the final cold shape's grid locations in a hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command).

## HTOCTOL

Default = 1.E-2 (SOL 106 only)

HTOCTOL is used to determine convergence of cold shape in hot-to-cold analysis. (See the ANALYSIS=HOT2COLD Case Control command). The parameter is used to compare the geometries as the model deforms from its "hot" to "cold" shape.

## HTSYM

Default = 0

This parameter controls the decomposition method for SOL 400 thermal analysis.

- = 0     Use symmetric decomposition solver
- = 1     Use unsymmetric decomposition solver

## ICOPT

Default = 1, SOL 400 Only

Parameter ICOPT works together with the NLIC Case Control Command. The user input loads may or may not be in equilibrium with the initial condition. If ICOPT=0, MSC Nastran will compute the initial acceleration based on user's inputs. Otherwise, it will be assumed that the initial acceleration is null. In other words, when ICOPT=1 (the default), it is assumed the whole structure is in equilibrium automatically. Theoretically, ICOPT=0 gives better solution. However, due to that the matrix is highly singular, a large amount CPU time may be required and the accuracy of the result may be in doubt for the solution with ICOPT=0.

## IFP

Default = value of PARAM,DBALL.

See DBALL.

## INREL

Default = 0

INREL controls the calculation of inertia relief or enforced acceleration in linear static analysis and buckling. INREL = -1, -2, or -4 requests that inertia relief or enforced acceleration be performed.

For INREL = -1 and -2, enforced accelerations, if desired, are input on the DMIG,UACCEL Bulk Data entry. (See Section 7.2 of the *MSC Nastran Reference Guide* for the theoretical basis.)

Inertia Relief is not currently supported with external superelements and in the contact analysis.

- 1 SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. The total number of degrees-of-freedom specified on SUPORT and SUPORT1 entries must be less than or equal to six.

In SOL 105, SUPORT1, not SUPORT, Bulk Data entries must be used to define the supported degrees-of-freedom and the SUPORT1 Case Control command may only be specified in a separate static subcase.

Loads due to unit rigid body accelerations at the point referenced by PARAM,GRDPNT are computed and then appended to the external loads. If PARAM,GRDPNT is specified in superelement analysis, then the point must be interior to the residual structure and exterior to all superelements.

- 2 The value of PARAM,INREL,-2 will provide an inertia relief analysis without the use of SUPORT<sub>i</sub> entries. To use this capability the structure must contain six and only six rigid body degrees of freedom. SUPORT<sub>i</sub> entries must not be present when using this option. If the structure has either more or less than six rigid body degrees of freedom, the analysis will either fail or give incorrect results.

- 4 SUPORT or SUPORT1 entries are required on one or more grid points in the Bulk Data Section which restrain rigid body motion. Any number of degrees-of-freedom may be specified on SUPORT and SUPORT1 entries. With this method, the mass matrix is reduced which will increase CPU requirements over PARAM,INREL,-1 and -2. The INREL=-4 method is a general approach and is capable of handling mechanisms and unconstrained problems. It is also capable of handling problems that have possible rigid body motion in selected directions. The INREL=-1 and -2 methods are more efficient implementation but are limited to six or less supported degrees-of-freedom because those methods can only handle problems that are fully unconstrained (that is, those with no constraints preventing rigid body motion of any kind and whose model does not contain mechanisms).

## INRLM

Replaced by the INRLOD keyword on the RESVEC Case Control command.

## IRES

Default = -1

IRES=1 requests that the residual load vectors RULV and RUOV be output in all solution sequences. In superelement analysis, the parameters PRPA and PRPJ may also be used to request output of the partial load vectors  $\{P_a\}$  and  $\{P_j\}$ , respectively. In geometric nonlinear analysis, PARAM,IRES,1 will cause the printing of the residual vector

$$\{\Delta P_f\} = [K_{ff}] \left\{ u_f^{n+1} - u_f \right\} + \{F_f\} - \{P_f\}$$

## ITAPE

Default = -1

ITAPE specifies the output status of the DSCMR matrix in SOLs 101, 103, and 105; and the DSCMCOL table and the DSCM2 matrix in SOL 200. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

## IUNIT

Default = 11

IUNIT specifies the FORTRAN unit number on which the DSCMR matrix in Design Sensitivity SOLs 101, 103, and 105 and the DSCMCOL table and the DSCM2 matrix in SOL 200 will be written. (See the OUTPUT2 and OUTPUT4 module descriptions in the [MSC Nastran DMAP Programmer's Guide](#).)

## JWLDET

Default=NOLINK

With the default multiple denotations with EOSJWL are NOT LINKED: the detonation wave of one explosive cannot ignite another explosive. When the option is LINK, multiple denotations with EOSJWL are LINKED: the detonation wave of one explosive can ignite another explosive. For option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

## KDAMP, KDAMPFL

Default = 1

If KDAMP or KDAMPFL is set to -1, viscous modal damping is entered into the complex stiffness matrix as structural damping. In coupled fluid-structure analysis, KDAMP is applied to the structural portion of the model and KDAMPFL to the fluid portion of the model. See [Superelement Analysis](#) in *the MSC Nastran Reference Guide*.

## KDIAG

Default = -1.0 (SOLs 106, 153, and SOL 400 with non-contact analysis), or 0.0 ( SOL 400 with contact analysis)

In SOLs 106 (nonlinear static analysis), 153 (steady state heat transfer), and SOL 400 (nonlinear static and transient analysis), KDIAG may be used to eliminate spurious mechanisms and singularities in the nonlinear stiffness matrix. The absolute value of KDIAG will be added to some or all of the diagonal terms in the nonlinear stiffness matrix as follows:

- < 0.0        Then add the absolute value of KDIAG to the null diagonal terms only (SOLs 106 and 153). For SOL 400, the absolute value of KDIAG is added to the diagonal term of null columns only.
- = 0.0        Then no action is taken.
- > 0.0        Then add the value of KDIAG to all diagonal terms.

## KDMFILT

Default = 0

- 0            Differential stiffness correction will NOT be added to rigid body behavior of pre-stressed models for modal analysis.
- 1            Differential stiffness correction will be added to rigid body behavior of pre-stressed models for modal analysis.
  1. Currently, PARAM, KDMFILT, 1 is active in SOL103 with a preload subcase (STATSUB) and a single modal (METHOD) subcase.
  2. Currently, PARAM, KDMFILT, 1 is active in SOL400 with a series of preload steps and a single ANALYSIS=MODES step. Also, in SOL400, the correction is not available when enhanced nonlinear elements are used via NLMOPTS or PSHLN1, etc., type entries.

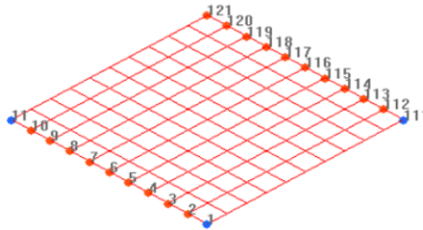
In nonlinear mechanics, in theory a tangent matrix is computed, using a deformation gradient. By polar decomposition, a consistent rotation matrix can be obtained.

In practice, most finite elements such as beam, shell and solid elements, are developed in a heuristic fashion with polynomial type functions representing displacement and volume characteristics. The tangent stiffness is composed of several matrices. A linear K matrix, a  $K_D$  for differential stiffness or geometric stiffness, and often a  $K_F$  for follower load stiffness.

In general, the  $K_D$  matrix which represents a stiffening to the structure due to pre-load of the structure, does not meet rigid body requirements. When a structure is preloaded with a self-equilibrating set of loads and statically supported, for a free-free modal analysis of the pre-loaded structure the static support set is removed. Often because of the deficiency of the  $K_D$  matrix to correctly represent rigid body modes, the resulting free-free modes are poor or missing.

PARAM, KDMFLT, 1 turns on a method to orthogonalize and normalize the computed rigid-body modes.

It cannot be over emphasized the importance of the need to have a self-equilibrating set of loads using a static support system.



The above structure represents a plate structure to be tension pre-stressed in the x-direction. This is accomplished by applying FORCE1 entries on grids 111 thru 121 in the positive x-direction defined by grids 1 thru 11 to grids 111 to 121 and a set of FORCE1 entries along grids 1 thru 11 in the negative x-direction defined by grids 111 thru 121 to grids 1 thru 11.

A static support set is applied at grids 1, 11, and 111. This set is removed for the modal analysis.

If, instead of the FORCE1 entries along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction, a self-equilibrating loading would not be obtained. This is so, because FORCE1 entries are follower loads while SPC1 entries are not. With the SPC1 entries, the structure would not be in true static equilibrium upon removal to the static support set.

If, instead, along grids 1 thru 11, the user chose to apply SPC1 entries for these grids, constraining then in the x-direction and non-follower FORCE entries along the grids 111 thru 121, an almost self-equilibrating loading is achieved and PARAM, KDMFLT, 1 will get six good free-free modes. In this case, however, the Lanczos Method has some stability issues and AHO should be used.

## K4RITZ

Default = 0

Setting PARAM,K4RITZ to a positive value activates logic to generate additional residual vectors for SOL 111 when using ACMS to compute the modal space. Structural damping degrees of freedom (K4) are used to augment the existing set of residual vectors in order to capture damping effects when GE inputs might contribute significantly to overall frequency response. The value of K4RITZ specifically controls the number of iterations performed to compute the basis vectors. Due to extra computation required, this parameter should be used with caution and should rarely be set greater than 1.

## K6ROT

Default = 100.

K6ROT specifies the scaling factor of the penalty stiffness to be added to the normal rotation for CQUAD4 and CTRIA3 elements. The contribution of the penalty term to the strain energy functional is



$$\Pi_p = 10^{-6} K6ROT \frac{1}{2} G \int_A (\Theta_z - \Omega_z)^2 t dA \quad \text{with} \quad \Omega_z = \frac{1}{2} \left( \frac{\partial u_y}{\partial x} - \frac{\partial u_x}{\partial y} \right)$$

where  $A$  is the area of the shell element,  $t$  is the shell thickness,  $G$  is the in plane shear modulus, see the MID1 material identification number on the **PSHELL** Bulk Data entry. The in plane displacements  $u_x$ ,  $u_y$  and the normal rotation  $\Theta_z$  are shown in **Figure 6-1**. The normal rotation has no physical meaning and should be ignored. The penalty stiffness removes the singularity in the normal rotation. A higher value than  $K6ROT=100$ . is not recommended because unwanted stiffening effects may occur. If you use only the MID1field, Nastran automatically sets  $K6ROT=0.0$

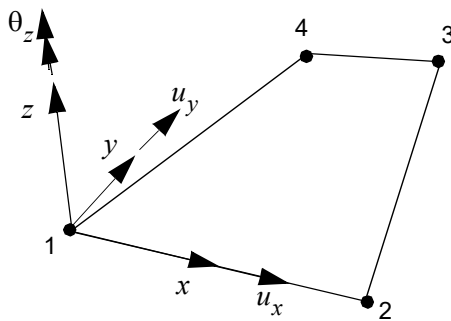


Figure 6-1 In plane displacements  $u_x$ ,  $u_y$  and normal rotation  $\Theta_z$

## LANGLE

Default = 1

LANGLE specifies the method for processing large rotations in nonlinear analysis. By default, large rotations are computed with the gimbal angle method in nonlinear analyses SOLs 106, 129, 153, and 159 with geometric nonlinearity (PARAM,LGDISP,1). If PARAM,LANGLE,2 is specified, then they are computed with the Rotation Vector method. The value of LANGLE cannot be changed in a subsequent restart. For SOL 400, users should not use LANGLE. SOL 400 will use the appropriate method depending on type of element or type of analysis.

## LDSUM

Default = 0

Dictates what trim information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM,XYUNIT, n. LDSUM has the following options:

0	(Default) – Do not create a CSV file for static aeroelasticity
1	Create a CSV file that contains for each static aeroelastic subcase: <ul style="list-style-type: none"> <li>■ Subcase ID</li> </ul>

	<ul style="list-style-type: none"> <li>■ Mach number</li> </ul>
	<ul style="list-style-type: none"> <li>■ Dynamic Pressure</li> </ul>
	<ul style="list-style-type: none"> <li>■ Trim Values</li> </ul>
	<ul style="list-style-type: none"> <li>■ Mass and CG information (mass, xcg,ycg,zcg, IXX,IYY,IZZ,IXY,IXZ and IYZ)</li> </ul>
2	Same as 1 plus net structural monitor point (MONPNT1, MONDSP1, MONPNT2, MONPNT3) results.
3	Same as 2 plus the output of RIGID AIR, ELASTIC RESTRAINED, and INERTIAL, RIGID APPLIED and ELASTIC APPLIED components for the structural MONPNT1 results.
4	Same as 3 plus the output of RIGID AIR and ELASTIC RESTRAINED components for aerodynamic MONPNT1 results.

## LFREQ, LFREQFL

Default = 0.0

See HFREQ, HFREQFL

If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

## LGDISP

LGDISP is a global parameter. For the case with multiple LGDISP definitions, Nastran will use the first definition for all the subcases and steps. Default = -1

- ≥ 0 The differential stiffness for structural elements is computed for the linear elements and added to the differential stiffness of the nonlinear elements.
- 1 All the nonlinear structural element types that have a large displacement capability in SOLs 106, 129, 153, 159, 600 and SOL 400 (see [Table 3-1](#), under “Geometric Nonlinear” in the *MSC Nastran Reference Guide*) will be assumed to have large displacement effects (updated element coordinates and follower forces). For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).
- 1 No large displacement effects will be considered.
- 2 Follower force for structural elements effects will be ignored but large displacement effects will be considered. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be treated the same as LGDISP = -1 (i.e., original geometry for thermal stiffness, element fluxes and external loads).

- 11 Nonlinear structural elements will be treated the same as LGDISP = 1 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on the current updated geometry (updated element coordinates for the thermal stiffness, element fluxes and external loads).
- 12 Nonlinear structural elements will be treated the same as LGDISP = 2 in all supporting solution sequences. For simultaneous multi-physics coupling analysis defined through the SUBSTEP Case Control option in SOL 400, nonlinear heat transfer element types will be based on updated current geometry (updated element coordinates for the thermal stiffness and element fluxes, original geometry for external loads).

## LMFACT

LMFACT and PENFN are the scale factor and penalty function for the Lagrange rigid elements and the contact analysis. For Lagrange rigid elements, please see Case Control command, RIGID. The purpose of LMFACT and PENFN is to make the values of stiffness matrix of the Lagrange rigid elements and/or the contact components about the same relative magnitude as those of the other elements in the model. Too small a value will produce inaccurate results and too large a value will produce numerical difficulties. The same value is usually assigned to both LMFACT and PENFN. Under special requirement, user may assign different values for LMFACT and PENFN. For example, if PENFN=0.0 and LMFACT  $\neq$  0.0, then the solution method for the rigid elements becomes the pure Lagrange multiplier method instead of the augmented Lagrangian method. However, user must exercise caution if different values are assigned to LMFACT and PENFN. MSC Nastran will compute the appropriate default values for LMFACT and PENFN. The default value is 1.0e+5 for all solution sequences except SOL 400 and SOL 101 contact.

For SOL 400 and SOL 101 contact, MSC Nastran will compute the appropriate default values for LMFACT and PENFN. These defaults are currently calculated as 0.05% of the average stiffness of the diagonal terms of the stiffness matrix. This computed default is good in general. But when the material/element stiffnesses (e.g., Young's Modulus, Spring Stiffnesses) used in the model vary in a wide range, users may have to adjust the computed LMFACT and PENFN. Note that the computed LMFACT, PENFN are printed out in a .f06 file. It is generally recommended that for these cases, the LMFACT and PENFN should be reduced by several orders of magnitude to avoid numerical difficulty.

If in a coupled multi-physics analysis distinct values for LMFACT and PENFN are desired for each physics pass, the parameter definitions must be made inside the SUBSTEP Case Control command. If no values are entered each physics pass computes its own defaults.

## LMODES, LMODESFL

Default = 0

LMODES and LMODESFL are the number of lowest modes to use in a modal formulation. In coupled fluid-structure analysis, LMODES specifies the lowest modes of the structural portion of the model and LMODESFL the modes of the fluid portion of the model. If LMODES (or LMODESFL) = 0, the retained modes are determined by the parameters LFREQ and HFREQ (or LFREQFL and HFREQFL).

In SOL 103, LMODES may be used to reduce the number of eigenvectors to be processed in data recovery which may significantly reduce the CPU and storage costs.

**Note:** If the MODESELECT Case Control command is used, it takes precedence over the parameters LMODES, LFREQ and HFREQ (or LMODESFL, LFREQFL and HFREQFL if MODESELECT refers to fluid modes). For the hierarchy of usage when the MODESELECT Case Control command is used in conjunction with these parameters, refer to the Remarks in the description of the MODESELECT Case Control command. See also the FLSFSEL Case Control command for an alternative selection.

## LOADU

Default = -1

See POST=0.

## LOOPID

Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153 the restart proceeds from the last loop ID of the subcase defined by SUBCASID or SUBID. In SOLs 106, and 153 PARAM,SUBID or SUBCASID may also be specified.

## LSTRN

Replaced by the STRAIN Case Control command.

## MACH

Default = 0.0

Mach number. If more than one Mach number was used to compute aerodynamic matrices, the one closest to MACH will be used in dynamic aeroelastic response analysis. The default causes the matrices computed at the lowest MACH number to be used.

## MARALPHA

Default = 10, SOL 600 only.

Determines what type of coefficient of thermal expansion (CTE) is entered in the Nastran input file when CTE varies with temperature. For most Nastran solution sequences, such as SOL 106 and SOL 400, if CTE's vary with temperature, secant values need to be entered. This parameter specifies whether secant or actual values are entered at each temperature on curves using the TABLEM1 entry. In addition, it offers methods to convert the secant CTE's instantaneous values required by Marc. Please read MAT1 Remark 10 and MATHE Remark 8 for further details.

- 1 CTE's (vs temperature) are entered in the Nastran input as instantaneous values – no conversion is required (Warning if this option is used, the CTE's should only be used in SOL 600 and will produce incorrect results if used in SOL 106 or SOL 400).
- 0 CTE's (vs temperature) are secant values (as required by SOL 106, 129 and 400). They will be converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = A_i + \left( \frac{A_i - A_{i-1}}{T_i - T_{i-1}} \right) (T_i - T_{ref})$$

- 1 CTE's (vs temperature) are secant values and are converted to instantaneous values using the formula immediately below for use in the Marc portion of SOL 600.

$$\alpha_i = \frac{A_i(T_i - T_{ref}) - A_{i-1}(T_{i-1} - T_{ref})}{T_i - T_{i-1}}$$

- 2 CTE's (vs temperature) are secant values and are converted to instantaneous values using the average of the values computed by options 0 and 1.
- N Same as option 1 except N intermediate points are placed between each original CTE vs temperature value to increase accuracy. N must be 3 or greater. Testing has shown that for CTE's that change appreciably over the temperature range, N should be 10 or larger. The total number of points must not exceed 8000.

### Remarks:

1.  $\alpha$  is the actual (instantaneous) CTE
2.  $T$  is temperature  $T_{ref}$  is specified on MAT1 and other MAT entries.
3. At  $T(1)\alpha_1 = A_1$
4. The above equations are applied at mid temperatures. The first temperature is retained. Starting with the second temperature  $T_{iN} = 0.5 \cdot (T_i + T_{i-1})$  original. Since the new final temperature will be smaller than the original the user should ensure that the final average temperature is large enough to cover the applied temperature range of the analysis.
5. If the CTE's entered in the Nastran input file are secant values, it is recommended that MARALPHA=N with N set to a value of 20 or larger but less than 8000 divided by the number of points in the largest CTE vs temperature TABLEM1 entry.
6. Only option -1 was available prior to MD Nastran 2010 and MSC Nastran 2008.

### MARAUTOC

Default = 0, SOL 600 only.

Determines whether NLAUTO entries for SOL 600,129 will override the default or not.

- 0 Do not override the default. (Default)
- 1 NLAUTO initial time step and final time will override the defaults.

**Remark:**

The default for the initial time step is to use the DT value from the entry or the first non-zero time value on the TABLED1 entry divided by 100.0 whichever is smaller. The default for the final time is NDT\*DT from the entry or the last time point from the TABLED1 entry whichever is smaller.

**MARBATCH**

Default = 0, SOL 600 only.

Specifies whether Marc will be spawned from Nastran in the “batch” mode or not.

- 0            Marc will be spawned using batch=no. (Default)
- 1            Marc will be spawned using batch=yes.

**Note:** PARAM,MARBATCH,0 requires PARAM,MARCTEMP,1 (which is the default). This combination of parameters will place the Marc log file in the Nastran log file.

**MARBK105**

Default = 1, SOL 600 only.

This parameter controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,105.

- 1            Nonlinear eigenvalues are found. In other words, all loads are placed after Marc’s END OPTION and default values are used for CONTROL and AUTOSTEP. This option simulates what happens with SOL 106 or SOL 400.
- 1            Linear eigenvalues are found. In other words, all loads are placed before Marc’s END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 105. (Default)

**MARBK106**

Default = 1, SOL 600 only.

Controls whether linear buckling or nonlinear buckling eigenvalues are calculated for SOL 600,106.

- 1            Nonlinear eigenvalues are found. In other words, all loads are placed after Marc’s END OPTION and default values are used for CONTROL and AUTO STEP. This option simulates what happens with SOL 106 or SOL 400.
- 1            Linear eigenvalues are found. In other words, all loads are placed before Marc’s END OPTION, a linear analysis is used to obtain the differential stiffness and eigenvalues are then calculated. This option simulates what happens with SOL 106.

## MARC4401

Default = 0, SOL 600 only.

Determines whether recycling due to body to body contact occurs.

- 0 Recycling due to body to body contact can occur
- 1 Recycling due to body to body contact is prevented - will add feature,4401 to the Marc input

## MARC7601

Default = 0, SOL 600 only.

Determines whether large deformation RBE3 element will be used.

- 0 Use small deformation RBE3
- 1 Use large deformation RBE3

### Remarks:

1. MARC7601=0 sets FEATURE,7601 in Marc.
2. MARC7601=1 will not set FEATURE,7601.

## MARCASUM

Default is -1 for nonlinear analysis and 1 for linear analysis. SOL 600 only.

Marc's assumed strain formulation is used for plane stress, plane strain and solid elements (Marc types 3, 11 and 7). The assumed strain formulation improves the bending behavior of these elements and uses an enriched set of interpolation functions. Assumed strain should be off for analyses with a significant amount of plasticity. In determining the type of analysis (linear or nonlinear) for defaults of this parameter, the SOL 600, ID Executive statement is used. If ID is 106 or 129, the analysis is considered to be nonlinear and the default is -1. If ID is any other value, the analysis is considered to be linear and the default is 1. For nonlinear analyses without plasticity, this parameter should be turned on for models with solid elements.

- 1 Assumed strain is not used.
- 1 Assumed strain is used.

## MARCAUTO

Default = leave out parameter, SOL 600 only.

Determines which Marc's increment option is used.

- 1 NLPARM entries will be translated to Marc's AUTO STEP option. If contact is present, the number of steps (NINC) is less than 100, it will be reset to 100. Marc will adaptively reduce the number of steps if possible, however, this option forces the first step to be 1% of the total time. If the first step is too large, experience has shown that convergence problems may result. To start with a different initial time step, see options 999 or -999.
- 1 NLPARM entries will be translated to Marc's AUTO INCREMENT option. If contact is present, the number of steps is automatically set to 100. It has been found that certain difficult contact problems which fail using the AUTO STEP option run successfully using AUTO INCREMENT. This option is not available if the only "loading" is rigid contact or velocity control.
- 2 NLPARM entries will be translated to Marc's AUTO LOAD option with no adjustment in the number of steps. Use of the option is not recommended. This option is not available if the only "loading" is rigid contact or velocity control.
- 999 Marc's AUTO STEP option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.
- 999 Marc's AUTO INCREMENT option will be used with no adjustment in the number of steps whether or not contact is present. This option is not available if the only "loading" is rigid contact or velocity control.

See PARAM,MARCITER for a similar option. Do not use both MARCAUTO and MARCITER parameters.

## MARCAXEL

Default = 0, SOL 600 only.

Allows a combination of axisymmetric and plane stress elements for 2D analyses. This analysis technique is sometimes used for approximate turbine disk/blade analysis.

- 0 The combination, if present in the input data will cause a Severe Warning and Marc will not be spawned.
- 1 Combination is allowed and all CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR elements will be considered to be plane stress and mate with CTRIAX6 elements.

## MARCBEAM

Default = -1, SOL 600 only.



- 1 All CBEAM elements which reference PBEAML will be mapped to Marc element type 14 for all cross if any CBEAM elements in the model reference MATS1 or MATEP entries. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM (where only A, I, J are available) will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 0 CBEAM will be mapped to Marc element type 14 for all cross sections specified with PBEAML. Full plasticity is available for all such cross section shapes with this option. CBEAM cross sections specified using PBEAM will be mapped to Marc element 98 and remain elastic even though they might reference MATS1 or MATEP.
- 1 All CBEAM elements will be mapped to Marc element type 98 and remain elastic regardless of whether the cross section is specified using PBEAM or PBEAML or whether they reference MATS1 or MATEP.

**Remark:**

If PBMARB6 or PBMNUM6 beam properties are used, MACRBEAM must be set to zero which automatically activates the new forms of Marc element 98 that can include plasticity.

**MARCBODY**

Default=0, SOL 600 only.

Control the logic when the specified value of NBODIES entry on BCPARA bulk data card does not match the actual number of contact bodies in the model.

- 0 Analysis continuing with number of bodies actually in the model
- 1 Analysis continuing with number of bodies specified on NBODIES entry on BCPARA bulk data card. (This may produce exit 13 in Marc)

**MARCBUSH**

Default = -1.0 if parameter is not entered, no “small” stiffness will replace zero stiffness terms, SOL 600 only.

Determines whether “small” stiffness values will be used instead of zero for the stiffness values in various directions of CBUSHi elements.

- 1.0 No “small” stiffness terms will replace zero stiffness values in any direction.
- 0.0 Stiffness values of 0.01\*Kmax will be added for any direction that is zero Marc input (applies to both translational and rotational directions)
- Value The value entered will be used to calculate  $\text{stiffness} = \text{Value} * K_{\text{max}}$  to replace any zero stiffness values (applies to both translational and rotational directions)

**MARCCBAR**

Default = 0, SOL 600 only.

Specifies whether CBAR will be replaced by CBEAM for SOL 600.



- 0      CBAR is not replaced by CBEAM.
- 1      CBAR is replaced by CBEAM (PBAR is replaced by PBEAM, PBARL is replaced by PBEAML).

**Note:** Use of this parameter is not usually required but might be beneficial in combination with PARAM,MSPEEDSE,1 to speed up translation of models with a large number of CBAR elements particularly when there are a large number of PBAR entries or PBARL entries.

## MARCCENT

Default = 0, SOL 600 only.

Controls where the element output is generated.

- 0      Element output from Marc will be generated for each integration point.
- 1      Element output from Marc will be generated at the center of each element only. This option saves disk space and computer time, but may not catch the maximum stresses or strains. Because the residual load calculation is not accurate, this should not be used in a nonlinear analysis.

## MARCCON2

Default = Program determines value, SOL 600 only.

- Value      If entered, the integer value entered is the second value on Marc's CONTACT second entry representing the maximum number of entities to be created for any contact surface.

## MARCCON3

Default = Program determines value, SOL 600 only.

- Value      If entered, the integer value entered is the third value on Marc's CONTACT second entry representing the maximum number of nodes that lie on the periphery of any deformable contact surface.

## MARCCPY

If MARCCPY is specified, Marc files will be copied to Nastran output files and/or deleted according to the option (0, 1, or 2) shown below.

MARCCPY Option	Copy Marc Output Files to Nastran Output Files	Delete Marc Input & Output Files
0 (Default)	No	No
1	Yes	Yes
2	Yes	No
3	No	Yes

If MARCCPY is 1 or 2, the out and log files will be copied as produced by Marc. If MARCCPY is -1 or -2 the actions as shown above for +1 or +2 will occur, and Marc-type test will be converted to Nastran-type text using and ASCII file named marcfilt.txt which must be located in the same directory where the Nastran input resides or in the same directory where the Nastran executable resides.

The following Marc files are potentially affected by the MARCCPY option:

Marc Output File	Nastran Output Copied to	MARCCPY
name.marc.out	name.f06	1, 2, -1, -2
name.marc.log	name.log	1, 2, -1, -2
name.marc.t16	not copied, will remain if produced	
name.op2, fort.11, or ftn11	not copied, will remain if produced	

## MARCDEF

Default = 2, SOL 600 only.

- 0 SOL 600 default options for Marc will be set to values determined to be best for Nastran-type problems (for MARCDEF=0, Marc's SHELL SECT parameter will be set to 11 if the value of MARCDEF is zero.
- 1 Default values will be set to current Marc standard (Mentat) values.
- 2 Default values will be set to "improved" Marc default values agreed on by the Marc and Nastran development groups.

Default values affect the following Marc input data entries and fields:

MARCDEF Value	MARC Implicit Entry Type	Field	Value
0	control	2	10
0	Auto Step	5	0.01*max time
0	Auto Step	8	10
0	Auto Step	10	1

MARCDEF Value	MARC Implicit Entry Type	Field	Value
1	control	2	3
1	Auto Step	5	1.0E-5*max time
1	Auto Step	8	5
1	Auto Step	10	0
2	control	2	10
2	Auto Step	5	1.0E-3*max time
2	Auto Step	8	5
2	Auto Step	10	1

**Note:** For MARCDEF=0, the first three values were found to provide better convergence and the last (auto step 10) allows snap-through solution to converge correctly without having to use arc-length methods. This parameter can be set in the system-wide rc file as well as the user's rc file or the local rc file (same directory as the Nastran input data to provide the selected set of defaults for all runs if so desired. If the parameter is entered in the Nastran input data file, it will override any parameters set in any of the rc files.

## MARCDILT

Default = 0, SOL 600 only.

If omitted, SOL 600 determines the value.

- 0 Constant dilatation is not used.
- 1 Constant dilatation formulation is used for solids, axisymmetric, and plane strain elements (advance nonlinear element types 7, 10, 11, 19 and 20) if the model includes any of these element types. For elastic-plastic and creep analysis this formulation is usually too stiff when constant dilatation is not used. MARCDILT=1 and MARCASUM=1 should not both be used.

## MARCDIS2

Default = Program determines value, SOL 600 only.

- Value If entered, this integer value entered here is the second value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of different lists of distributed loads.

## MARCDIS3

Default = Program determines value, SOL 600 only.

**Value**        If entered, the integer value entered here is the third value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of elements in any particular distributed loads list.

## MARCDIS4

Default = Program determines value, SOL 600 only.

**Value**        If entered, the integer value entered here is the fourth value on Marc's DIST LOADS ("parameter" Section 2 of Marc's Volume C Program Input) entry representing the maximum number of nodes with point loads applied.

## MARCDMIG,N

Default = 0, SOL 600 only.

If matrices or loads are entered using K2GG, M2GG, B2GG, K2PP, M2PP, B2PP, P2G in the Nastran Case Control Section, they will be translated to Marc as follows depending on the value of N:

**N=0**        All DMIG's in the Nastran file (and include files) will be placed in the Marc input file whether used or not.

**N>0**        All DMIG's in the Nastran file (and include files) will be placed on a new file named dmigxxx.dmi where xxx is the value of N. This new file will be "included" in Marc using a Marc include statement. For example, if N=100 the file name will be dmig100.dmi if N=25765 the file is dmig25765.dmi. N must not exceed a value of 999999.

**Note:** This parameter is ignored for External Superelements (if the MESUPER Bulk Data entry is present).

## MARCDUPE

SOL 600 only.

Controls whether SOL 600 will check for duplicate entries for most every type of bulk data entry. SOL 600 does not allow duplicate entries, but the portion of IFP that runs prior to spawning Marc does not usually check for duplicate entries.

**1**        Duplicate entries will be checked and exact duplicates are found, the job will fatal out.

**2**        In addition to option 1, entries will be checked and if duplicate ID's (field 2 for most entries or field 3 for loads) are found, the job will fatal out.

**-1**       Duplicate entries and ID's will not be checked (this is desirable for certain models)

## MARCDYND

Integer, Default = 0

Determines whether dynamic loads with “advanced table input” will be used in SOL 600.

- 0 Do not use advanced table input for dynamic loads
- 1 Use advanced table input for dynamic loads

If MARCDYND=1, PARAM,MARCTOTD,1 and PARAM,MARCTOTT,1 must also be set.

## MARCEKND

Default = 0, SOL 600 only.

- ID Selects the type of strain results to be placed in a Nastran op2 file (if a request for an op2 file is made). ID can take the following values:

MARCEKND=0, Total strains will be processed

MARCEKND=1, Plastic strains will be processed

MARCEKND=2, Elastic strains will be processed

For creep analyses, creep strain is output if a request for strain output is made.

## MARCEXIT

Default = 0, SOL 600 only.

- 0 If one of the COPYR options on the SOL 600 statement is specified, Nastran will process these options and then a DMAP exit will occur.
- 1 The COPYR options will be processed and Nastran will not exit.

## MARCFEAT,N

SOL 600 only.

If entered will add FEATURE,N to the Marc input file in the Parameters Section.

- N Feature to be added, for example PARAM,MARCFEAT,5102 will generate a heat transfer thermal contact file jid.marc.nthcnt

Only one PARAM,MARCFEAT may be entered.

## MARCFILI

No Default, SOL 600 only.

Name Name a file name limited to 8 characters (16 characters if param\* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. For example, if CONTINUE=1 on the SOL 600 statement and PARAM,MARCFIL1,DMIG44 is entered, friction stiffness and possibly damping) matrices are created in DMIG format by Marc and placed on file DMIG44. The various CONTINUE options use the following MARCFILi entries:

Continue Option	MARCFILi	SOL Executed	K2GG/K2PP
1.	MARCFIL1	107	K2GG
2.	MARCFIL2	107	K2GG
3.	MARCFIL3	111	K2PP
4.	MARCFIL1	112	K2PP
5.	MARCFIL1	*	*
6.	MARCFIL1	110	K2GG
7.	MARCFIL1	103	K2GG

**Remarks:**

- For most continue options other than brake squeal models, the Marc portion of SOL 600 will produce files with very long names such as jid.marc.conmpc\_0007. These file names are too long for a Nastran parameter field even if PARAM\* is used. The solution is to specify a short name such as PARAM,MARCFIL1,ABCD.TXT  
 Inside ABCD.TXT place an include line for the actual file desired, such as  
 INCLUDE 'myjob.marc.conmpc\_0007'  
 Where myjob would be replaced by the actual JID of the primary Nastran run without the bdf or dat extension.
- If INITCON=4 on the BCPARA entry is specified contact MPC's for each increment will be out on a file named jid.marc.conmpc\_incr (if the increment is 7 it would be jid.marc.conmpc\_0007).
- If the DMIGOUT Bulk Data entry is used, DMIG's will be produced in the form jid.marc\_cglsti\_incr. See the description of DMIGOUT for further details.
- For most cases, the difference between jid.marc.conmpc\_last and jid.marc.conmpc.0001 is not large enough to affect the subsequent analysis significantly. To be sure that the last jid.marc.conmpc\_XXXX is used specify the following  
 param,marcfil1,lastt  
 Nastran will search for all jid.marc.conmpc\_\* files and choose the one with the largest \_XXXX. If no jid.marc.conmpc\_\* files exist, the job will terminate with an fatal error message.

## MARCFRIC

Default = 0.0, SOL 600 only.

When the Case Control command, BCONTACT = ALL is specified, no other 3D contact data is required in the input file, except that the Coulomb coefficient of friction may be entered using the value of this parameter. Do not enter this entry if contact surfaces are specified in the Bulk Data.

## MARCGAPD, D

Default is U0 of the PGAP entry, SOL 600 only.

- D        Depending on the value of PARAM,MARCGAPP, enter the gap closure distance for fixed direction gaps or the minimum distance between end points for the true distance gap. If  $d > 0$ , the two end points are never closer than a distance  $|d|$  apart. If  $d < 0$ , the two end points are never farther apart than  $|d|$ .

## MARCGAPN, ID

No Default, SOL 600 only.

- ID        ID of gap element for which the immediately following PARAM,MARCGAPP and PARAM,MARCGAPD apply. Unlike most other parameters, several sequences of parameters MARCGAPN, MARCGAPP and MARCGAPD may be entered to specify values for all gap elements. If no MARCGAPN is entered, the values entered for MARCGAPP and MARCGAPD will be used for all gaps in the model.

## MARCGAPP

Default = 0, SOL 600 only.

- 0        Nastran gap elements will be translated to Marc fixed gap elements.  
1        Nastran gap elements will be translated to Marc True Distance gaps.

## MARCGAUS

Default if parameter is not entered = 1, SOL 600 only.

- 1        SOL 600 output stresses and strains will be at the Gauss points for solid elements and extrapolated to the corner points for plates/shells elements. Strains are handled the same way as stresses.



- 2 SOL 600 output stresses and strains will be at the center and at the grid points for solid elements (the maximum stress from any Gauss point is determined and compatible stresses for that Gauss point are placed at the center and at each of the grid points). Shell/plate stresses are at the center (top and bottom surfaces). The maximum Gauss point stress at each surface is found and a compatible set of stresses at that Gauss point are placed in the center of the surface. Strains are handled the same way as stresses.
- 3 Solid stresses/and strains are the same as option 1 and shell/plate stresses and strains are the same as option 2.

If this parameter is entered with values of zero or less or values greater than 3, it will be reset to 1.

## MARCGLUE

Default = 0 if parameter is not entered, SOL 600 only.

If MARCGLUE is set to 1, all contact surfaces will be glued whether or not IGLUE=1 is specified on the BCTABLE entry or not.

- 0 IGLUE on BCTABLE entries specifies whether or not glued contact is used
- 1 IGLUE on BCTABLE entries will be ignored and glued contact will be used for all contact surfaces.

### Remark:

For this parameter to function BCTABLE entries must be entered (do not set BCONTACT=ALL).

## MARCRAV

Integer, Default = 0

Determines whether PARAM,MNASTLDS,777 will automatically be set if multiple GRAV entries are present in any subcase. This is determined in a simplified fashion by comparing the number of GRAV entries to the number of subcases to save computer time.

- 0 Set PARAM,MNASTLDS,777 for multiple GRAV entries.
- 1 Do not set PARAM,MNASTLDS,777 regardless of the number of GRAV and SUBCASE entries.

## MARCHOST

No Default, SOL 600 only.

Determines the name of a hostfile to be used with SOL 600 parallel runs. If this parameter is missing, no host file is used and the parallel run will run on one machine. That machine may have several processors and as many processors as specified on the PARAMARC Bulk Data entry will be used. If

PARAM,MARCHOST,Name is specified, the hostfile must be generated by the user in a format acceptable to Marc (see the *Marc and Marc Mentat Installation and Operations Guide*). Each line of the hostfile normally

lists how many processors are used on each machine. If PARAM\*,MARHOST is entered, the name is limited to 16 characters (all lower case).

## MARCIAMN

Default = 1, SOL 600 only.

- 0 Nastran is directed to spawn Marc (as specified by the SOL 600 Executive Control statement or PARAM,MARCRUN), using a full version of Marc. Standard Marc licensing is required.
- 1 A special version of Marc is spawned by Nastran. This version of Marc may have certain features that are not available in the full version. Marc will be spawned from Nastran with the additional command line switch - iam nanl. The licensing for both Nastran and Marc reflect this situation. This option applies only to Marc version 2003 or later. If PARAM,MARCVERS points to a Marc version earlier than 2003, MARCIAMN will be set to zero and a full version of Marc is required.

The parameter may be set in the system-wide rc, the user rc file or as an environmental variable using NASM\_IAMN=0 or 1 (similar to the way values on the SOL 600 statement are set).

## MARCINTC

Default = 2.

Option to ignore or fatal SOL 600 if any CINTC Bulk Data entries are found This option will also ignore or issue a warning for GMBNDC entries.

- 0 Fatal job if any CINTC entries are found (also issue warning messages if any GMBNDC entries are found).
- 1 Ignore all CINTC and GMBNDC entries in SOL 600.
- 2 Generate the MPC's for CINTC/GMBNDC and combine them with standard MCP entries (if they exist), then use them in the SOL 600 analysis (SOL 600 only).

### Remark:

MARCINTC=2 spawns a secondary SOL 100 Nastran job to generate MPC's for the CINTC's. Licenses from the primary job are released, then reclaimed after the secondary job terminates. The secondary job stops prior to decomp and uses the standard SOL 101 licenses.

## MARCINTF

Default = 1.0D-6.

Threshold value below which MPC coefficients generated by CINTC/GMBNDC are not considered. This parameter is ignored unless PARAM,MARCINTC,2 has been entered.

## MARCITER

Default = 0, SOL 600 only.

Used to control fixed time stepping in SOL 600.

- 0 Fixed time steps or auto time steps will be controlled by PARAM,MARCAUTO.
- N Allows fixed time steps to be used without needing to set the maximum and minimum times to nearly the same value (using Marc's AUTO STEP option). This parameter triggers true fixed time stepping with the other advantages AUTO STEP has over methods such as AUTO LOAD. For example, it uses better numerical damping. If this parameter is entered with a positive integer (N), a value of 2 is placed in the AUTO STEP field 9 and N is placed in field 7.
- 1 This option is similar to PARAM,MARCITER,N (fixed time stepping will be used) except that the time comes from the NLPARAM or entry. This option is not available if the only "loading" is rigid contact or velocity control. See PARAM,MARCAUTO for a similar option. Do not use both MARCAUTO and MARCITER parameters.

### MARCL001

Default = -1, SOL 600 only.

Determines whether Marc's POINT LOAD (without tables) 2<sup>nd</sup> datablock, 3<sup>rd</sup> field will be honored or not. If this value is set to 1 multiple loads at the same dof in the same subcase will usually be summed (however, see item d.).

- 1 The 2<sup>nd</sup> datablock 3<sup>rd</sup> field will be set to 1 (see Remark)
- 1 The 2<sup>nd</sup> datablock 3<sup>rd</sup> field will not be set to 1. If it is necessary to sum the loads, this must be accomplished in one of several ways:
- a. Summed in the GUI or other method of data input
  - b. By setting PARAM,MNASTLDS,1
  - b. By setting PARAM,MARCTOTT,1
  - d. For certain simple multiple force/moment cases, SOL 600 can sum them and PARAM,MNASTLDS,1 must be specified to prevent loads from being specified multiple times. However, it may be difficult to determine whether the loads are complex or not, so the use of either option c or b is recommended.

#### Remarks:

1. For releases prior to MD Nastran 2010 and MSC Nastran 2009, this field was set to 1 to automatically sum any forces or moments that might have been entered more than once by the GUI (see PARAM,MNASTLDS for a discussion of ways this can happen). This field, when set to one, allows loads entered more than once at the same grid ID in the same subcase to be summed. However, experience shows that this field does not work in certain circumstances, particularly when "total loads" using PARAM,MARCTOTL are requested. PARAM,NASTLDS has been added beginning with MD Nastran 2010 and MSC Nastran 2009 to handle the cases where the 2<sup>nd</sup> datablock 3<sup>rd</sup> field does not work as expected.

2. To maintain backward compatibility with previous runs, set the following parameters:  
 PARAM,MARCL001,1  
 PARAM,MNASTLDS,0  
 PARAM,MARCTOTL,0  
 PARAM,MARCTOTT,0
3. If PARAM,MARCTOTL,1 is entered, PARAM,MARCL001,1 will be set unless PARAM,MARCL001,-1 is entered.
4. This parameter may be set in RC files.

## MARCLOWE

Default = 0, SOL 600 only.

Used in conjunction with superelement matrices created by Marc.

- |   |  |
|---|--|
| 0 | Standard modulus values for all materials will be used.  |
| 1 | All modulus of elasticity values will be changed to 1.0E-9 for the second Nastran run (when Nastran spawns another Nastran run using the SOL 600 continue option. This option is sometimes necessary for cases where Marc creates a superelement or substructure stiffness matrix but does not create a mass matrix. In this case, the second Nastran run will create the mass matrix using standard elements, density and other concentrated and distributed masses but the stiffness created by Nastran will be very low. Essentially the entire stiffness of the model will come from the stiffness matrices created by Marc. |

## MARCLUMP

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable). (Default) |
| 1 | Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems with rotational masses (if applicable).               |
| 2 | Consistent mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.                        |
| 3 | Lumped mass will be used for SOL 600 transient dynamics or eigenvalue problems without rotational masses.                            |

### Remark:

This parameter is only used with dynamic analysis.

## MARCMAT2

Default = -1.0, SOL 600 only.

Used if  $g_{33} = 0.0$  on MAT2 entries. Marc will diverge if  $g_{33} = 0.0$  for MAT2 entries. If the value entered is positive, the value is a multiplier of  $g_{11}$  and  $g_{22}$  to calculate  $g_{33}$  as follows:

$$g_{33} = \text{marcmat2} * (g_{11} + g_{22})$$

### MARCMAT3

Default = 1, SOL 600 only.

Used if Nastran has generated MAT2 from PCOMP and the MID of MAT2 is 30000001 is greater corresponding to MID3 for PSHELL.

If the value entered is 0, this entry is ignored regardless of the MAT2 MID value.

If the value entered is 1 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all  $C_{ij}$  are zero except the following:

$$C_{55} = g_{11}$$

$$C_{56} = g_{12}$$

$$C_{66} = g_{22}$$

If the value entered is 2 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all  $C_{ij}$  are zero except the following:

$$C_{44} = g_{11}$$

$$C_{45} = g_{12}$$

$$C_{55} = g_{22}$$

$$C_{66} = g_{22}$$

If the value entered is 11 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all  $C_{ij}$  are zero except the following:

$$C_{55} = g_{11}$$

$$C_{56} = g_{12}$$

$$C_{66} = g_{22}$$

If the value entered is 12 the MAT2 entry will be mapped to Marc's ANISOTROPIC entry such that all  $C_{ij}$  are zero except the following:

$$C_{44} = g_{11}$$

$$C_{45} = g_{12}$$

$$C_{55} = g_{22}$$

$$C_{66} = g_{22}$$

**Note:** This entry is ignored unless the MAT2 MID is greater than 30000000.

### MARCMATT

Default = -1 if parameter is not entered, SOL 600 only.

Determines if Marc input file will be created with materials using the table-driven formats or not.

- 1 Table-driven formats for materials will not be used. (Default)
- 1 Table-driven formats for materials will be used

**Remark:**

This parameter can be set in RC files.

**MARCMEM, Value**

Default = Program determines value, SOL 600 only.

**Value**        If entered, the integer value entered here is the second field on Marc's SIZING entry (MAXALL) and is the main memory specification for memory in Marc. This value is entered in MW (the program multiplies it by 1,000,000). For example, if a value of 350 is entered, the number of 350000000 will be placed in the second field of the SIZING entry.

**MARCMID3**

Default = 0, SOL 600 only.

Controls whether MID3 will be set to the same value as MID2 when the Marc PSHELL option is used (designated by PARAM,MRPSHELL,1 or when the SMEAR option is used on the SOL 600 Executive Control statement.)

- 0        MID3 will not be changed (if zero or blank, it will remain zero or blank). (Default)
- 1        MID3 will be set to MID2. This improves the singularity ratio in some problems without appreciably changing the results particularly when orthotropic material properties are used. It is not necessary if PARAM,MMAT2ANI,11 or a similar option is used to specify anisotropic material properties.

**MARCMNF**

Default = 0, SOL 600 only.

Controls creation of an Adams MNF file by Marc for eigenvalue analysis when Marc is spawned from SOL 600.

- 0        MNF file will not be created. (Default)
- 1        MNF file will be created (for Marc 2003, the MNF file is located in the .t19 file, so PARAM,MARCT19,1 must also be specified).

**Note:** It is not necessary to use this parameter if the MDMIOUT Bulk Data entry is used to request an MSC Adams MNF file.

**MARCMPC**

Default = 0, SOL 600 only.

Determines whether SOL 600 converts MPCs to stiff beams or not.

- 0 MPC's are not converted to stiff beams. (Default)
- 1 MPC's are converted to stiff beams.

**Note:** PARAM,MARCMPC,1 should not be used if RBE's are converted to MPC's using PARAM,MARCRBE2 or PARAM,MARCRBE3.

## MARCMPII

Integer, Default = 2 for DDM parallel executions, SOL 600 only.

Determines whether the MPI service on PC systems will remain running after the job finishes.

- 1 The MPI service will remain running after SOL 600 finishes
- 1 The Intel MPI service, ismpd.exe will be uninstalled before SOL 600 terminates.
- 2 The Intel MPI service, ismpd.exe will be started before each Marc execution and terminated at the end of each Marc execution from within the Nastran portion of SOL 600.
- 771 MSMPI rather than Intel MPI is requested (starting and/or stopping of the ismpd.exe Intel MPI service is ignored)

### Remarks:

1. If SOL 600 terminates abnormally the service may remain running regardless of the value of this parameter.
2. Leaving the MPI service running will speed up the execution of multiple jobs.
3. This parameter may be entered in RC files.
4. Although ismpd.exe is started by the Marc script, it sometimes fails to start properly unless started outside the script. The user can either do this in his own script or set MARCMPII=2 to have Nastran do it. Either of these options improves reliability of the Intel MPI starting.

## MARCN99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all nodal quantities will be generated or not. If MARCN99=1, all Case Control nodal output requests must have the PLOT qualifier or the job may fail.

- 1 A set named ND999999 will be generated to output all nodes for at least one type of nodal output. This is the default of all Case Control nodal requests do not have (plot).
- 1 The set will not be generated. If all nodal Case Control requests have (plot) such as DISP(PLOT)=ALL, ACCEL(PLOT)=ALL, etc. the default is 1 even if the parameter is not entered.

## MARCNOER

Default = 0, SOL 600 only.

Determines action to take when unsupported features are encountered.

- 0 The internal Marc translator will stop and generate FATAL ERRORS if unsupported features in Marc or in the internal translator are encountered.
- 1 If unsupported features are encountered, they are ignored, no FATAL ERROR messages are issued and if requested, Marc will be executed.

## MARCOFFT

Default = -1, SOL 600 only.

Controls whether Nodal Temperatures are specified at the original or offset grid point or both grid points for beams and shells with offsets or connected by RBE2's (see note 2) when PARAM,MAROFSET is set to 0,2,3. For the default MAROFSET=1 where Marc handles the offsets and RBE2's are not added, this parameter is not applicable and will be reset to -1 internally. Applies only if MAROFSET=0.

- 0 Temperatures are applied both at the original grid point and at the offset grid point.
- 1 Temperatures are applied at the original grid point only.
- 2 Temperatures are applied at the offset grid point only.
- 1 Temperature loading is not altered in any way from the Nastran input.

**Note:**

1. Processing time can increase significantly if beam or shell offsets are present and param,marcofft is zero or greater unless PARAM,MOFFCORE,1 is used.
2. If nodes are connected by RBE2's and a temperature is applied at an independent node, the temperature will be applied at the independent node and the first dependent node of the rbe2 if MARCOFFT=0.
3. If MAROFSET is 0, 2 or 3 and a node has applied temperature and is also a part of a standard RBE2 with many grids, the job may abort or the results can be wrong.

## MARCONTF

No Default, SOL 600 only.

- Name Name of a file name limited to 8 characters (16 characters if param\* is used) used in conjunction with one of the CONTINUE options on the SOL 600 statement. If entered this file will be used as the input file for the second Nastran execution (after Marc has finished). If specified, this file will be used instead of automatically creating a file named jid.nast.dat from the original jid.dat input. This option allows more versatility in achieving exactly what is desired in the Nastran continuation run input at the expense of additional input data preparation.



## MARCOOCC

Default = 0, SOL 600 only.

- 0 Standard Marc memory management will be used. If the memory is sufficient the decomposition will be in core. If the memory requirements are too large, an out of core solution will take place.
- 1 An out of core solution will be forced if solvers 0, 2, 4 or 8 are used. This option triggers Marc parameter, OOC (without any other characters).
- 2 An out of core solution will be forced if solver 0, 2, 4 or 8 are used - available starting with Marc 2005 r2. This option generates Marc parameter OOC,0,1.
- 3 Same as option2 except Marc parameter version,4902 will be added. This option triggers Marc parameter OOC,0,1 plus version,4902.

## MARCOPT

Default = 9, SOL 600 only.

Determines which bandwidth optimizer is to be used.

- 0 Marc does not optimize the bandwidth.
- 2 Cuthill-McKee bandwidth optimization is used
- 5 External user-supplied bandwidth optimization is used
- 9 Sloan bandwidth optimization is used. (Default)
- 10 Minimum degree bandwidth optimization is used (only available with the sparse solver)
- 11 Metis nested dissection algorithm (only available with multifrontal direct sparse solver)
- 9999 Set MARCOPT to -9999 if the OPTIMIZE entry is not wanted in the Marc file for example with use by the iterative solver.

## MARCOSET

Default = 0, SOL 600 only.

Determines whether SOL 600 set names will be standard sets or “open sets” for nodes and elements. The standard Marc sets are:

DEFINE, ELEMENT, SET, PR00001  
DEFINE, NODE, SET, ND001

The Marc open sets (OSET) are:

DEFINE, ELEMENT, OSET, PR00001  
DEFINE, NODE, OSET, ND001

- 1 Standard sets are defined.

- 0 Standard sets are defined for small models and open sets are defined for large models.
- 1 Open sets are defined.

**Note:** For Parallel (DDM) analyses, it is sometimes necessary to set MARCOSET=1.

## MARCOTIM

Default = 0, (SOL 600 only and is mapped to Marc's POST 2nd line 9th field.

Determines if Marc is to be processed at selective or at all output times.

- 0 or 1 Marc output data will be processed by Nastran at all converged output times. This option is similar to INTOUT=YES on the NLPARM entry.
- 2 Marc output data will be processed by Marc only at times near 1.0, 2.0, 3.0, etc. For this option, no additional output times will be available in the Marc .t16 and/or .t19 output files. This option is similar to INTOUT=NO on the NLPARM entry.
- N Marc output data will be processed by Marc every Nth increment for the .t16 and/or .t19 output files. (N > 2)

## MARCOUTR

Default = 1 if Marc single file parallel input is used, Default = 0 if Marc multiple file inputs are used, see Note, SOL 600 only.

Determines how Marc t16 file results will be handled for SOL 600 parallel processing.

- 0 Multiple t16 files, one for each domain will be produced.
- 1 A single t16 file will be produced by Marc. This option requires Marc 2005 or later versions and the parallel run made using the "single file" input (PARAMARC KIND=0). (Default)

**Note:** Whether single file or multiple Marc inputs are used for parallel processing is determined by the PARAMARC Bulk Data entry.

## MARCPARR

Default = 0, SOL 600 only.

Controls options for splitting an Marc file into parts for DDM.

- 0 All Marc files will be created during this run. (Default)
- 1 Nastran will be stopped after the single-processor file has been created and before DDM files are created. If desired, all files may be copied to a backup directory for use with MARCPARR=2.

- 2 The parallel files will be created starting with the single processor file created using the MARCPARR=1 option.
- 3 Same as MARCPARR=2 except the debug option MARCBUG=1 is turned on.

## MARCPENT

Default = 0, SOL 600 only.

Option to specify if CPENTA is mapped to Marc brick element with degenerate nodes or actual penta elements (see Remarks).

- 0 Map CPENTA to Marc brick elements with degenerate nodes. (Default)
- 1 Marc CPENTA to actual Marc penta elements.

### Remarks:

1. Option zero is the default to maintain backwards compatibility.
2. The t16 to op2 version is not available for option 1. That means no op2, xdb, f06 or punch file is available for option 1. Postprocessing must occur using the t16 or t19 file if option 1 is used.
3. Speed options such as invoked using param,mspeedse are available using option 1 only.

## MARCPINN

Default = 0, SOL 600 only.

- 0 Pin flags will be included by created new nodes and appropriate MPC's by the translator in Nastran. (Default)
- 1 Pin flags will be ignored and the translator will continue.
- 2 A severe warning will be issued and Marc will not run.
- 3 The new Marc PIN CODE option will be used. feature,6901 will also be used so that extra nodes are not introduced.

### Remark:

This parameter can be set in RC files.

## MARCPLAS, n

Default = 3 if there is plasticity in the model. SOL 600 only.

This parameter effects the value of Marc's PLASTICITY parameter. The value of n can range from 0 to 6 (0 is the same as 3).

- 1 Additive decomposition using the mean normal method; small strain formulation
- 2 Additive decomposition using the radial return method; small strain formulation.

- 3 Additive decomposition using the mean normal method; large strain formulation using the updated Lagrange procedure.
- 4 Additive decomposition using the radial return method; large strain formulation during the updated Lagrange procedure.
- 5 Multiplicative decomposition (FeFp) using the radial return method and the three field variational principle; large strain formulation using the updated Lagrange procedure.
- 6 Advance nonlinear elements type 3 and 26 (plane stress), 18 and 30 (membrane) using multiplicative decomposition with the radial return method; large strain formulation using the updated Lagrange procedure.
- 1 Ensures that the Marc PLASTICITY parameter will not be used.

## MARCPOS

SOL 600 only.

Determines whether to terminate Marc if a non-positive definite matrix is attempted.

- 0 The run will terminate if a non-positive definite matrix decomposition is encountered.
- 1 Non-positive definite matrices will be solved.

## MARCPOST

Default, SOL 600 only.

Determines the format for the .t16 and .t19 files.

- 1 The .t16 and .t19 files will be created using the Marc default for the version of Marc that is executed.
- 0 Same as MARCPOST=9.
- 1 The .t16 and .t19 files will be created using Marc K2 formats.
- 3 The .t16 and .t19 files will be created using Marc K3 formats.
- 4 The .t16 and .t19 files will be created using Marc K4 formats.
- 5 The .t16 and .t19 files will be created using Marc K5 formats.
- 6 The .t16 and .t19 files will be created using Marc K6 formats.
- 7 The .t16 and .t19 files will be created using Marc K7 formats.
- 8 The .t16 and .t19 files will be created using Marc K8 formats.
- 9 The .t16 and .t19 files will be created using Marc 2000 formats.
- 10 The .t16 and .t19 files will be created using Marc 2001 formats.
- 11 The .t16 and .t19 files will be created using Marc 2003 formats.

- 12 The .t16 and .t19 files will be created using Marc 2005 formats.
- 13 The .t16 and .t19 files will be created using Marc 2005 r3 formats.

**Note:** It is suggested that a small test case be executed and tested with your postprocessor to determine what version is necessary for your postprocessor.

### MARCPR99

Default-see below. SOL 600 only.

Determines whether a set in the Marc input file to output all elemental quantities will be generated or not. If MARCPR99=1, all Case Control element output requests must have the PLOT qualifier or the job may fail.

- 1 A set named PR999999 will be generated to output all elements for at least one type of element output. This is the default if all elemental Case Control requests do not have (plot).
- 1 The set will not be generated. If all elemental Case Control requests have (plot) such as STRESS(PLOT)=ALL, STRAIN(PLOT)=ALL, etc. The default is 1 even if the parameter is not entered.

### MARCPRN

SOL 600 only.

Controls the amount of contact information printed.

- 0 Detailed contact information is not printed.
- 1 Detailed contact information is printed (this is equivalent to Marc parameter PRINT,2,8).
- 2 Somewhat less detailed contact information is printed (this is equivalent to Marc parameter PRINT,2). Print constraint matrices associated with MPC's, RBAR, RBE2, RBE3 and the formable to deformable contact.
- 5 Marc print option. PRINT,5 will be used. Prints messages when changes in contact status occur.
- 25 Marc print options. PRINT,2,5 will be used.
- 258 Marc print options. PRINT,2,5,8 will be used. In addition to 2 and 5, also prints the displacement and reaction forces in the local coordinate system associated with formable to rigid contact.

For other print options, use the MARCIN Bulk Data entry.

### MARCPRNG

Default = 0, SOL 600 only.

Determines whether geometry is printed in the Marc .out file.



- 0 Most geometry printing is suppressed.
- 1 All geometry is printed.

## MARCPNR

Default = 0, SOL 600 only.

Determines whether nodal stress and strain output is printed in the Marc .out file.

- 0 Nodal, stress and strain printing is suppressed.
- 1 Nodal, stress and strain printing will occur if Case Control options request it specifically or by default.

## MARCPROG

No Default, SOL 600 only.

- prg “prg” is the name of a program to be executed instead of Marc. The program may be any program desired and must already be fully compiled on the computer system being run. prg is limited to 8 characters, however 16 characters can be used if PARAM\* is entered. “prg” must be a compiled program, not a script or batch file. The name of the program must be in lower case (if not, it will be converted to lower case).

## MARCRACC

Default = 0, SOL 600 only.

SOL 600 does not normally support RACC on the RFORCE entry. The default is to fatal a job when RACC is zero or blank. This parameter may be used to set it to zero or to obtain approximate results.

- 0 If MARCRACC is zero and RACC is nonzero, the job will terminate with an appropriate message (Default).
- 1 If MARCRACC is one and RACC is nonzero, RACC will be set to zero internally and the job will continue.
- 2 If MARCRACC is two and RACC is nonzero, the following will occur:  
for brake squeal, coriolis loading and values (C1, C2, C3) will be placed in Marc ROTATION A 3rd datablock fields 4-6 as follows:  

$$RR = \sqrt{R1^2 + R2^2 + R3^2}$$

$$C1 = RACC * R1 / RR$$

$$C2 = RACC * R3 / RR$$

$$C3 = RACC * R3 / RR$$

Where RACC, R1, R2, R3 are described on the RFORCE entry.

For standard structural analysis if MARCRACC=2, the value of A in RFORCE field 5 will be altered using the following equation:

$$A_{new} = \sqrt{A^2 + RACC}$$

Marc's ROTATION A entry will be the same as if RACC=0.0 was entered unless A is zero, in which case the third line will have all zero entries. Results should be considered as an approximation.

### Remarks:

1. See PARAM,MARCRCID for related SOL 600 RFORCE uses.
2. MARCRACC=2 only applies for brake squeal and only if the BRKSQL entry is used for releases prior to MD Nastran R2.1 and MSC Nastran 2007.

### MARCRBAL

Default = 0, SOL 600 only.

This parameter is used only for eigenvalue analysis in the Nastran-Marc interface where natural frequencies or buckling modes need to be calculated using the deformed geometry from a nonlinear analysis. The parameter is only necessary if the last nonlinear increment created a non-positive definite matrix. When MARCRBAL=1 is set, the system will be rebalanced and a positive-definite matrix is assured. Do not use this parameter unless it is known that a non-positive definite system occurs just prior to eigenvalue analysis.

### MARCRBAR

Default = 0, SOL 600 only.

Determines how RBAR is treated.

- |   |  |
|---|--|
| 0 | RBARS will be simulated using Marc's Servo Link. This option is best only for small incremental deformation and rotations. (Default)                     |
| 1 | RBARS will be converted to RBE2 with GN=GA, CM=123456, GM1=GB. The MARCRBE2 option specified will then be used to translate the converted RBARS to Marc. |

### MARCRBE2

Default = 3, SOL 600 only.

- |    |  |
|----|--|
| ID | Determines the type of RBE2 used. The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements.   |
| 0  | RBE2s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations.  |
| 1  | RBE2s will be simulated using Marc's TYING type 80 for translation and Servo Link for rotations. This option is capable of larger incremental deformations but requires small rotational increments. The MARCRBE2=1 option is only available if all 6 DOF's are specified in the CM field (4) of the RBE2 entry. |

- 2 RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 2D analysis.
- 3 RBE2s will be simulated using the new RBE2 element introduced into Marc Version 2003 for a 3D analysis.

**Note:** If RBAR, RROD or RTRPLT elements are found in the model and if MARCRBE2=2 or 3, these elements will be converted to equivalent RBE2's and used with the new Marc RBE2 element during the Marc execution (thus providing higher accuracy for large deformations and/or rotations).

The default should be used (parameter not entered) with models comprised of solid elements or a mixture of solid elements and other types of elements.

### MARCRBE3

Default = 3 SOL 600 only.

Determines the type of RBE3 used.

- 0 RBE3s will be simulated using Marc's Servo Link. This option is best only for small incremental deformations and rotations (see option 4 for a similar alternative).
- 2 RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 2D analysis.
- 3 RBE3s will be simulated using the new Marc RBE3 element introduced into Marc Version 2003 for a 3D analysis. (Default)
- 4 Same as MARCRBE3=0 except that all MPC's due to RBE3 will be placed ahead of all other MPC's. This option might improve the Marc solution for versions where Marc has not implemented AUTOMSET logic (those versions prior to MSC Nastran 2005 r3).

### MARCREVR

Default = 0, SOL 600 only.

Specifies that all rigid surfaces need to be modified.

- 0 Rigid contact surfaces are correct as entered and no changes are made by the translator. (Default)
- 1 All rigid surfaces are entered backwards and will be reversed.

### MARCREVRX

SOL 600 only.

Determines whether coordinates for NURBS2D (BCBODY) will be reversed or not if CQUADX, CTRIAX, and/or CTRIAX6 elements exist.



- 0 Reverse the coordinates (Y becomes X, X becomes Y).
- 1 In addition to reversing the coordinates (option 0) all points are also reversed. For example, if there are 4 points (1,2,3,4) they are reversed to (4,3,2,1). (Default)
- 1 Do not reverse the coordinates.
- 2 Reverse the coordinates for CQUADX and CTRIAX abut not for CTRIAX6.

## MARCRIGD

Default = 0, SOL 600 only.

The parameter should only be entered if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 0 All RBEi will not be converted to stiff beams or plates. (Default)
- 1 All RBEi will be converted to stiff beams or plates for the Marc input file using a stiffness scale factor from parameter MARCRSCL. This parameter allows such elements to have large rotations for versions of Marc which do not include large rotations of rigid elements.

### Remarks:

1. This option may not be used if RBAR, RROD or RTRPLT elements are in the model. See PARAM,MSTFBEAM for an alternative.
2. See PARAM,MARCSCLR to specify a scale factor for the “default” properties of these stiff beams.

## MARCSAME

Default = 0, SOL 600 only.

Determines whether SOL 600 runs with multiple subcase having the same LOAD ID or loads (see note below) in more than one subcase will be processed or not. SOL 600 will usually run under such circumstances but may get the wrong results.

**Important:** Do not use this parameter if the loading contains enforced displacements or the results may be incorrect.

- 0 The job will be aborted before Marc is spawned with a “Severe Warning” message. (Default)
- 1 The job will run to completion (if there are no other errors) and a standard Warning message will be issued.

It is recommended that if the same loads are to be used in multiple subcases that each subcase have a different LOAD ID. A typical file setup for SOL 600 should be setup in the following manner:

```
SOL 600,NLSTATIC PATH=1 STOP=1
CEND
DISP=ALL
STRESS=ALL
```

```

SPC=123
TEMP (INIT)=33
PARAM,MARCSAME,1
SUBCASE 1
LOAD=100
SUBCASE 2
LOAD=200
TEMP (LOAD)=300
BEGIN BULK
LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 1., 1.0, 1000, 1.0, 2000
PLOAD4, 1000, 10, 20.0
PLOAD4, 2000, 20, 25.0
(Other Bulk Data entries)
ENDDATA

```

**Note:** In the above example if the Bulk Data LOAD entries are changed to:

```

LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 0.9, 1.0, 1000, 1.0, 2000

```

The loads are the same before the overall scale factor (1.0 and 0.9, respectively) are applied and PARAM,MARCSAME,1 is needed. However if the entries were as follows, it is not needed because the loads before the overall scale factor is applied are different.

```

LOAD, 100, 1., 1.0, 1000, 1.0, 2000
LOAD, 200, 1.0, 0.9, 1000, 1.0, 2000

```

## MARCSCLR

SOL 600 only.

Sets property values for stiff beams when PARAM,MARCRIGD or PARAM,MSTFBEM are used in SOL 600.

The value of this parameter scales the “default” properties of stiff beams or plates if parameter MARCRIGD=1. The “default” (unscaled) values for the stiff beams are A = 10, I = 100 (both directions) J = 200, shear area = 5 and plate thickness of 1.5. Linear scaling is used for all areas and thickness, all inertia terms are multiplied by the square of the Value entered.

## MARCSETS

Integer, Default = 0, SOL 600 only.

Controls the type of sets used for the Marc input file element and node sets.

- |   |   |
|---|---|
| 0 | Sets will have the format DEFINE, ELEMENT, ... or DEFINE, NODE, ... |
| 1 | Sets will have the format DEFINE, ELSQ, ... or DEFINE, NDSQ, ...    |

### Remarks:

1. PARAM,MARCSETS,1 can speed up certain large Marc jobs by a considerable amount.
2. The parameter may be set in RC files.

## MARCSETT

Default = 0, SOL 600 only.

- |   |   |
|---|---|
| 0 | The current environment is not printed. (Default)   |
| 1 | The current environment is printed in the .f06 file. A user program named eodenv.f must be compiled, linked and placed in the input file directory. The contents of eodevn.f resembles the following: |

```
Program eodenv
call system("set")
stop
end
```

## MARCSINC

Default = 0, SOL 600 only.

This parameter controls how often a spline file is written if the spline option (analytical contact for deformable bodies) is requested. If this parameter is not entered or if it is 0 or -1, then a file is not written. If N is greater or equal to 1, then every nth time step is written. Spline files have the extension \*.mfd which may be processed by MSC Mentat.

## MARCSIZ3, Value

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered here is the third value on Marc's SIZING entry representing the maximum number of elements. |
|-------|---|

## MARCSIZ4, Value

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered here is the fourth value on Marc's SIZING entry representing the maximum number of grid points. |
|-------|---|

## MARCSIZ5, Value

Default = Program determines value, SOL 600 only.

- |       |   |
|-------|---|
| Value | If entered, the integer value entered here is the fifth value on Marc's SIZING entry representing the maximum number of constrained degrees-of-freedom. |
|-------|---|

## MARCSIZ6, Value

Default = Program determines value, SOL 600 only.

**Value**            If entered, the integer value entered here is the sixth value on Marc's SIZING entry representing the maximum number of elements in the largest list of distributed loads (the internal Marc to Nastran translator generates these one at a time, so this value is normally 1).

## MARCSLHT

Default = 5, SOL 600 only.

Number of layers through the shell thickness used to integrate shell and beam elements. For linear behavior, N=1 is sufficient. For most plasticity problems, N=5 is adequate. For extremely nonlinear plasticity problems N=11 should be used. SOL 600 requires that N be 5 or larger. If N is entered with a positive value less than 5, SOL 600 will set it to 5. To use values smaller than 5, enter N as a negative number. The absolute value will be used, however the job may fail or results may be incorrect if the model has plasticity.

**Note:** Use of PARAM,MARCDEF can effect the value of Marc's SHELL SECT parameter if PARAM,MARCSLHT is not entered. To eliminate SHELL SECT from the Marc file set N to -9999.

## MARCSOLV

Default = 8, SOL 600 only.

Determines the solver to use for Marc.

0	The profile solver will be used (solver 8 should normally be used instead)
2	The Sparse Iterative Solver will be used.
4	The Direct Sparse Solver will be used.
6	A hardware-provided solver will be used.
8	A sparse solver similar to the one used by Nastran will be used (Default)
9	The CASI element based iterative solver will be used.
10	The mixed direct/iterative solver will be used.
11	Pardiso direct solver (see param,mrthread)
12	MUMPS parallel direct solver (see param,mrthread)

**Note:** If any NLSTRAT entries are entered, the solver type must be specified using the IOLSVR option of NLSTRAT rather than this parameter.

## MARCSPCC

Default = 0 if no SPCD's are entered or 2 if there are SPCD's in the input.

Determines how SOL 600 will proceed if non-zero value enforced displacements are found in SPC entries for multiple subcase models.

- |   |   |
|---|---|
| 0 | A fatal error will be issued and the user will be requested to change all non-zero value enforced displacements to SPCD's.  |
| 1 | The job will continue without any changes (wrong results could be produced).  |
| 2 | All displacements on SPC entries will be set to blank and the job will continue (will usually produce the correct results if SPCD's with the correct values are specified). |

### Remark:

Non-zero enforced displacements should always be specified using SPCD (not SPC) for SOL 600. It is not necessary to specify SPC's for degrees of freedom where SPCD's are applied for SOL 600, but it will not do harm unless non-zero enforced displacements are specified on the SPC entries (either in addition to being specified on SPCD's or without having any SPCD's). If there is only one subcases, the non-zero enforced displacements may be specified on SPCD's and/or SPC's, however for multiple subcases wrong results can result even if the displacements specified on the SPC's and SPCD's are the same. The results will be even further off if the displacements are different. The remedy is to ensure that the SPCD's reflect the correct applied displacements, and then either remove the applied displacements from all SPC's or to set MARCSPCC=2.

## MARCSTIFF, Time

Default = 1.0, SOL 600 only.

- |      |  |
|------|--|
| Time | This parameter specifies what time matrices entered using PARAM,MARCFILi will be used in a Nastran solution. The file may contain matrices at several times, but only the matrices specified by the parameter will be used. This parameter is not usually used, MRMTXNAM,NAME is used instead. |
|------|--|

## MARCSTOP

SOL 600 only.

- |   |   |
|---|---|
| 0 | Normal Marc execution when spawned from Nastran.                                      |
| 1 | Marc will exit (with exit code 7) after phase 0 (corresponds to Marc parameter stop). |

## MARCSUMY

Default = 0, SOL 600 only.

Determines if the summary of maximum values is to be printed.

- 0 A summary of maximum displacements, stresses and strains will be printed in the Marc output file. (Default)
- 1 The summary of maximum values is not output.

## MARCT16

Default = 2, SOL 600 only.

Controls generation of a Marc t16 file. MARCT16=0, Marc does not generate a .t16 output file.

- 1 or 0 Does not generate a t16 file.  
Parameter omitted generates a t16 file. All entries are controlled by the MARCOUT Bulk Data entry, or if MARCOUT is not specified, by the default shown in option 2 below.
- 1 Generates a .t16 output file with the following post codes:  
11 11,1 11,N 12 12,1 12,N 13 13,1 13,N 14 14,1 14,N  
15, 15,1 15,N 16 16,1 16,N 17 17,1 17,N 18 18,1 18,N  
7 7,1 7,N 27 27,1 27,N 301 301,1 301,N 321 321,1 321,N  
341 341,1 341,N 401 401,1 401,N  
Nodal: 1, 2, 3, 4, 5, 6, 35, 36, 37, 38, 39, 40, 46, 48, 51, 52
- 2 Generates a .t16 output file with the following post codes. (Default)  
301, 301,1 301,N 341 341,1 341,N 47  
Nodal: 1, 2, 3, 4, 5, 6, 34, 35, 36, 37, 38, 39, 46, 48
- 3 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):  
301, 341 47  
Nodal: 1, 2, 5, 6, 34, 35, 36, 37, 38, 39
- 4 Generates a .t16 output file with the following post codes (op2, xdb, f06, punch results cannot be made using this option):  
301 341 47  
Nodal: 1

### Remarks:

1. This entry is used as an easy way to control which results are placed on Marc's op2 file. All entries can be overridden using the MARCOUT Bulk Data entry. For MARCOUT values of 1 and larger, MARCOUT should be omitted for the input file. MARCOUT should not be used if PARAM,MARCT16 is entered.
2. The frequency of output is controlled by the NLPARM or Bulk Data entries (variables INTOUT and NO respectively).
3. Items such as 341,1 and 341,N designate stresses at the bottom and top surfaces (for applicable elements). Items such as 341 designate stress at mid-thickness.

4. When stress 341 is specified for models with composite elements, the value will be changed to 391 to obtain stresses in the fiber (layer) direction.
5. Consult Marc Volume C documentation for the meaning of the above blocks. Option 1 provides most of the structural output anyone might want, option 2 provides total strain, Cauchy stress, displacement and contact information at the top, center and bottom of surfaces. Option 3 provides the information of option 2 but only at the center (not at the top and bottom).
6. It is necessary to generate a t16 file in order to produce op2, xdb, f06 or punch results.
7. Op2, xdb, punch and f06 results can only be created using option 1 and 2 although option 0 can also be used if the selected outputs are the same as option 1 or 2.
8. If a Bulk Data VCCT entry is included in the model, nodal post codes 57-64 and 74 will be added to any of the previous defaults if MARCOUT is not entered in the model.

## MARCT19

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | Marc does not generate a .t19 output file. (Default) |
| 1 | Generates a .t19 output file.                        |

## MARCTABL

Default = 0, SOL 600 only.

Determines if contact table is to be generated.

- |   |  |
|---|--|
| 0 | Contact tables will be generated for the main Marc input (phase 0) and for each subcase if specified by the user (Default). When MARCTABL=0, each subcase may have a BCONTACT Case Control command and a matching BCTABLE ID entry. In addition, Marc's "Phase 0" entry is supported by entering a BCTABLE with an ID of zero (or 1,000,000). Each separate BCTABLE will reference the BCBODY entries defined which, in turn, reference BSURF entries. (Default)                           |
| 1 | No contact tables will be generated and all contact bodies (if any) will be placed in the main input data section. Contact will thus be the same for all subcases. When MARCTABL=1, there must only be one BCTABLE entry in the file. There must only be one BCONTACT command in the Case Control and it must be above all subcases. The BCONTACT and BCTABLE entry must have the same ID. The BCTABLE entry can reference several BCBODY entries which, in turn, reference BSURF entries. |

## MARCTEDF

Character\*8, no Default, SOL 600 only.

Enter the Marc nthcnt file name without extension. Use this option only if PARAM,MARCTEDN,1 is entered (the file name without extension is limited to 8 characters). The characters ".nthcnt" will automatically be appended to the name specified.

## MARCTEDN

Default = 0, SOL 600 only.

Determines whether a thermal contact analysis will use the Marc `jid.nthcnt` file generated by this run or use one input by the user.

- 0 Analysis uses the Marc `jid.marc.nthcnt` file generated in this run. (Default)
- 1 Analysis uses a Marc `nthcnt` file generated by the user or in a previous run.

**Note:** If MARCTEDN=1, PARAM,MARCTEDF below must be entered to specify the file name.

## MARCTEMP

Default = 1, SOL 600 only.

- 0 The scratch files produced by Marc will be in the same directory as the Nastran input file.
- 1 The scratch files produced by Marc will be in the same directory as the Nastran scratch files. (Default)

**Note:** The Marc scratch files cannot be split.

## MARCTETT

Integer, Default = 0, SOL 600 only.

Determines how CTETRA elements will be mapped to marc.

- 0 4-node or 10-node Nastran CTETRA elements will be mapped to the Marc tet-4 or tet-10 elements.
- 11 4-node or 10-node Nastran CTETRA elements will be mapped to Marc solid element type 7. for tet-10 elements, the mid-side nodes are neglected. the 4 CTETRA nodes (N1,N2,N3,N4) are mapped to the 8 solid shell nodes as follows N1,N2,N3,N4,N4,N4,N4.
- 12 Same as MARCTETT=11 except debug is turned on (for the tetra conversion only)

### Remark:

MARCTETT should not be greater than zero for heat transfer or Herrmann elements.



## MARCTIEC

Default = 1, SOL 600 only.

- 1 Transient Time Integration Error Check for Marc's AUTO STEP method. A value of 1 turns the check on. (Default)
- 0 A value of 0 turns the check off. Turn the check off to match Marc results for version prior to Marc 2003 r1.

## MARCTOL

Default = 0, SOL 600 only.

Determines the method of convergence tolerance.

If parameter MARCTOL=0, convergence tolerances are based on residuals (loads). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy).

- 1 Convergence tolerances are based on displacement.
- 2 Convergence tolerances are based on strain energy.
- 4 Convergence is achieved when either residual or displacement satisfies the criteria.
- 5 Convergence is achieved when both residual and displacement satisfies the criteria.

## MARCTOTD

Default = 0, SOL 600 only

Determines whether for SOL 600 dynamic analyses will use full table association.

- 0 Full table vs time association is not used. (Default)
- 1 Full table vs time association will be used

### Remarks:

1. param,marctott,1 must also be set if MARCTOTD=1
2. This parameter may be set in RC files.

## MARCTOTL

Default = 0, SOL 600 only

Determines whether total or incremental loads are used in a SOL 600 static nonlinear analysis.

- 0 Incremental loads are used. (Default)
- 1 Total loads are used

**Remarks:**

1. Warning - If MARCTOTL=1, MRFOLOW3=1 will be set if possible (see PARAM,MRFOLOW3)
2. This parameter may be set in RC files.

**MARCTOTT**

Default = 0, SOL 600 only

Determines whether total loads, including pressures, gravity, spcd, etc., with associated tables are used in a SOL 600 static or dynamic analysis (Remark 3.)

- |   |   |
|---|---|
| 0 | Total loads with tables are not used. (Default) |
| 1 | Total loads with tables are used                |

**Remarks:**

1. Warning - If MARCTOTT=1, MRFOLOW3=1 will be set if possible (see PARAM,MRFOLOW3)
2. This parameter may be set in RC files.
3. For full table association, dynamic analyses require PARAM,MARCTOTD,1 to also be set.

**MARCTUBE**

Default = 0, SOL 600 only.

Determines whether CTUBE maps to Marc element 31 or 98 (Default = 0 if this parameter is omitted).

- |   |   |
|---|---|
| 0 | CTUBE elements map to Marc tube element 31 (see Remark) (Default) |
| 1 | CTUBE elements map to Marc element 98 (elastic beam)              |

**Remarks:**

Marc element 31 does not support thermal loading or creep. If the model contains thermal Loading or creep (even if not applicable to the CTUBE elements, option 1 is always used).

**MARCTVL, Value**

SOL 600 only.

- |       |  |
|-------|--|
| Value | If parameter MARCTVL is entered, it must have a real value. The value entered is the convergence tolerance used by Marc (see PARAM, <a href="#">MARCTOL</a> ). If parameters MARCTOL and MARCTVL are not entered, the tolerances are determined by and/or NLPARM Bulk Data entries. However, parameters MARCTOL and MARCTVL provide extra control over these convergence tolerances particularly in the case where or NLPARM specify more than one convergence type (such as load and energy). |
|-------|--|

## MARCUSUB, chr

No Default, SOL 600 only.

chr            chr is the name of a “user subroutine” to be included in the Marc run. chr is limited to 8 characters without the .f extension. This file must be located in the same directory as the Nastran input data. The subroutine name must have all lowercase letters. chr should be in lower case to prevent confusion. (Nastran will convert the file name to uppercase, but it will be reconverted to lower case when Marc is spawned.) Any user subroutine available to Marc may be specified. Multiple user subroutines must be combined into one file.

Restriction: The computer must have a Fortran compiler and Linker and the Fortran compiler must be the same as used to create the original Marc executable (see the Marc installation manual). The Marc input file does not as yet call out user subroutines, so manual editing of the Marc input file may be necessary in some cases to invoke them. Existing regular Marc subroutines can be modified and handled in the same manner if available to you.

**Note:** If more than one user subroutine is required, all should be combined into one file before execution.

## MARCVERS

SOL 600 only.

This parameter has been replaced by the following parameters:

param,mrconver  
param,mrcontab  
param,mrsprver

## MARCWDIS

Default = 1, SOL 600 only.

Determines whether Marc parameter section DIST LOADS Is written or not. If any of the three values for DIST LOADS are entered (see PARAM,MARCDIS2 PARAM,MARCDIS3 PARAM,MARCDIS4) it will be written. If PARAM,MARCWDIS,1 is entered, it will be written. If PARAM,MARCWDIS,-1 is entered, it will not be written.

**Caution:** Nastran cannot estimate these values very well and produces overly conservative numbers that sometimes leads to failure of the Marc run due to lack of memory. We suggest that the user should use this parameter sparingly and enter the MARCDIS2, MARCDIS3 and MARCDIS4 values for best Marc memory usage.

## MARCWELD

Default = 0 if this parameter is omitted, SOL 600 only.

Determines how CWELD/PWELD elements will be translated to Marc.

- 0            Add extra nodes and elements to simulate the weld.
- 1            Use Marc CWELD/PWELD formulation (available starting with MD Nastran R2 and MSC Nastran 2007).

This parameter may be set in RC files.

## MARELSTO

Default = 0, SOL 600 only.

Determines whether Marc's parameter ELSTO will be created.

- 1          ELSTO will not be created.
- 0          ELSTO will be created only for large models as determined by Nastran. The value of ELSTO will be 40960 except for parallel runs (where a PARAMARC entry exists in the input file) in which case ELSTO will not be created. (Default)
- >0        ELSTO will be created with the value specified used as the Marc ELSTO parameter whether or not the run uses parallel processing.

## MARGPFEL

Default = 0, SOL 600 only.

Used to determine if Marc GRID FORCE output will occur by element, by node, or both ways.

- 1          Output is by element only.
- 0          Output is by node only. (Default)
- 1          Output is by both element and node.

### Notes:

1. This parameter effects the contents of the Marc jid.marc.grd file. Option 0 or 1 is required for GPFORCE output in op2, xdb, punch, and/or f06 files.
2. This parameter can be set in RC files.

## MARFACEA

Default = 1, SOL 600 only.

Face number for "A" side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

## MARFACEB

Default = 1, SOL 600 only.

Face number for “B” side of weld if welds are made of solid elements. Not used if welds involve plates or shells.

## MARFATAL

Default = 1, SOL 600 only.

Determines whether non-existent grid id’s for BCBODY entries will cause fatal errors or not.

- 0 Non-existent grid ID in BCBODY entries will only cause warning messages. The grid ID will be reset to zero and the job will continue. The results should be checked carefully. In some cases this could result in an entire contact body being skipped.
- 1 Non-existent grid ID in BCBODY entries will cause FATAL ERRORS. (Default)

## MARGPFOR

Default = 0, SOL 600 only.

Used to determine whether GPFORCE is active for SOL 600. A Case Control request GPFORCE must be specified to obtain grid point forces (see note 4). The grid point forces are output from Marc on a file named `jid.marc.grd`. The `t16op2` program reads this file, puts the data on the `f11` file along with displacements, stresses, etc. After `t16op2` finishes, the `f11` file is brought into DBALL, from which DMAP generated on the fly can produce `op2`, `xdb`, `punch`, or `f06` output.

- 1 GPFORCE is ignored (Default).
- 0 GPFORCE is output for the last time in the last subcase only.
- 1 GPFORCE is output at all times given in the `.sts` file.
- N GPFORCE is output for times 1, 1+N, 1+2N, etc.

### Notes:

1. This parameter can be set in RC files.
2. Output is available only in the new Marc `jid.marc.gid` file.
3. The Case Control GPFORCE request must be above all subcases, or the same within all subcases.
4. The `jid.marc.grd` file can become very large if option 0 or N is not used.

## MARHEATM

Default = 0, SOL 600 only.

Determines whether a file named `heatm.rc` is necessary to run the second phase of SOL 600 heat transfer initial contact job.

- 0 heatm.rc is not required. Defaults will be used. The defaults are scr=yes batch=no mem=80mw
- 1 A heatm.rc file will be supplied by the user in the same directory as the original Nastran input file. The heatm.rc can contain any information used by other rc files except that batch=no. If the original Nastran input file is named jid.dat (or jid.bdf) and out=jid is specified, the final output will be in files such as jid.f06, jid.op2, jid.xdb. If out=jid is not specified the final output will be in files such as jid.nast.f06, jid.nast.op2, jid.nast.xdb.

## MARHTPRT

SOL 600 only.

Controls heat transfer output in the Marc .out file.

- 0 Do not print any output except for summary table.
- 1 Print the nodal temperatures.
- 2 Print all possible nodal heat transfer output.

## MARIBOOC

Default = 0, SOL 600 only.

- 0 Incremental backup data will be stored in memory. (Default)
- 1 For large problems, incremental backup data will be stored on disk. This option triggers Marc's parameter IBOOC.

## MARIPROJ

Default = 0, SOL 600 only.

Flag to determine if auxiliary nodes of a CWELD will be projected on the model or not. (If this parameter is not entered)

- 0 Nodes are not projected on to the model.
- 1 Nodes are projected on to the model.

## MARLDCMB

Default = -1, SOL 600 only.

Determines whether extraneous loads in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 105, 106.

- 1 Extra loads will not be filtered out.
- 2 Extra Bulk Data LOAD entries will be filtered out early in SOL 600 to save computer time.

**Note:** If PARAM,MARLDRMV,-1 is not entered, extra FORCE, MOMENT and PLOAD4 entries not used in the present analysis will also be filtered out early in SOL 600. Computer time saving can be appreciable for certain models with many extra loads.

## MARLDRMV

Default = 1, SOL 600 only.

Determines whether extraneous FORCE, MOMENT, and/or PLOAD4 entries in the input file will be filtered out at an early stage to save computer time. This applies only to SOL 600, ID where ID=1, 3, 6, 101, 103, 105, 106.

- 1 Extra loads will not be filtered out.
- 1 Extra Bulk Data FORCE, MOMENT, and/or PLOAD4 entries will be filtered out early in SOL 600 to save computer time. (Default)

**Note:** This option will be ignored (same as PARAM,MARLDRMV,-1) if PARAM,MARLDCMB,-1 is entered.

## MARMPCHK

Default = -1, SOL 600 only.

Determines whether Marc parameter MPC-CHECK is written.

- 1 MPC-CHECK is not written. (Default)
- 1 Apply the MPCs in the default order:
  1. MPCs obtained from SERVO LINK option.
  2. MPCs obtained from INSET option.
  3. MPCs obtained from TYING, RBE2, or RBE3 options (the actual order follows from the order of these options in the model definition block of the data file.)
  4. MPCs obtained from CYCLIC SYMMETRY option.
  5. MPCs obtained from CONTACT option.
 Print a warning message if a tied degree of freedom is being used by a subsequent MPC.
- 2 Same as 1, but instead of warning, a fatal error message is printed and the analysis will stop with exit 2001.

- 3 Try to rearrange the MPCs in such a way that a tied degree of freedom will not be used in a subsequent MPC. If this reordering cannot successfully be completed, print a fatal error message and stop the analysis with exit 2011.
- 999 Skip MPC-CHECK entirely, prevent Marc's error message "Exceed the max trial to order RBE2/3". This option should be used in conjunction with PARAM,AUTOMSET,YES only. (Marc feature,3901 is set)

**Note:** This parameter can be set in RC files.

## MARMPCID

Integer, Default = 1, SOL 600 only

Determines the ID of the MPC Case Control command for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see SOL 600 Executive Control statement, CONTINUE option).

- 0 or 1 A line MPC=1 will be inserted in the Case Control (remark)
- N A line MPC=N will be inserted in the Case Control (N must be between 2 and 9999)

### Remark:

When models with contact and if BCPARA,INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc\_xxxx where xxxx ranges from 0001 to the number of increments in the solution. The ID of these MPC's is usually equal to one. If for some reason it is not one, the actual value may be specified by this parameter.

## MARMTLCK

Default = 0, SOL 600 only.

Determines whether a check of various property-material combinations for SOL 600 will be made or not.

- 0 The checks will not be made. (Default)
- 1 Check will be made. These take extra computer processing time and for most models are not required. The user should turn on these check if he is in doubt if any property-type of material combinations entered into the model may be in error. Current check made are for the following illegal combination.  
PSOLID/MATHE (model should be PLSOLID/MATHE)  
PSOLID/MATHP (model should be PLSOLID/MATHP)



## MARNOCID

Default = 0, SOL 600 only.

SOL 600 does not support MCID defined by cylindrical or spherical coordinate systems. This parameter determines whether MCID defined by cylindrical or spherical coordinate systems will be ignored or “fataled out” for shell and solid elements.

- 0 The run will be “fataled out” if MCID defined by cylindrical or spherical coordinate systems are found for shell or solid elements. (Default)
- 1 MCID defined by cylindrical or spherical coordinate systems for shell or solid elements will be ignored.

## MARNOSET, Name

No Default, SOL 600 only.

- Name Character. If entered, this parameter will not write out a set with the specified name. This is useful, when portion of the model specify sets that are not actually used in the present analysis. Up to 20 of these can be specified.

## MAROFSET

Default = 1, SOL 600 only.

Determines how beam and shell offsets are applied.

- 0 Extra grids and rigid elements will be created to model the offsets.
- 1 Marc will automatically handle offsets for beam and shell elements. No extra grids or elements will be created. The offsets will be found in Marc’s GEOMETRY data. (Default)
- 2 Marc will automatically handle offsets for beam elements only.
- 3 Marc will automatically handle offsets for shell elements only.

<b>Note:</b>	If MAROFSET is 1 or 2, the beam orientation can be specified using the CBAR/CBEAM “BIT” flag. It is suggested that only combinations, GGG or BGG be used.
--------------	---

## MARPLANE

Default = 0, SOL 600 only.

For composite structures described using PCOMP, together with CQUAD4, CQUADR, etc., it is not possible to tell whether a standard 3D shell or a plane strain shell has been modeled. If MARPLANE is set to 1, such composite models will be assumed to be plane strain (as if a PLPLANE property had been entered rather than PCOMP).

## MARPROCS

Determines the number of domains to be used in a DDM parallel execution.

1	Use 1 processor.
2	Use 2 processors.
N	Use N processors.

### Remarks:

1. This parameter is not intended to be used by parallel jobs using several different machines.
2. This parameter and PARAMARC may not both be entered in the same run.
3. This parameter and PARAM,MRTTHREAD may both be used in the same job.
4. The parameter may be used with most of the solver types except for the MUMPS solver (type 12).

## MARRBAR2

Default = 1, SOL 600 only.

If MARCRBAR is set to 1, all CMA fields will be changed to 123456.

0	CMA field will remain as coded by user, Marc run may fail.
1	CMA field will be set to 123456. (Default)

## MARROUT

Default = -1, SOL 600 only.

Determines whether an inconsistent set of outputs between the Marc t16 file (selected using MARCOUT) and standard Nastran output selected using Case Control requests (and param,post) is allowed or not.

-1	Inconsistent output is not allowed, if the outputs are inconsistent, the job will fatal early before spawning Marc. (Default)
1	Inconsistent output is allowed. However, if the output requests are inconsistent, the job may fail during the t16op2 conversion after Marc has finished.

### Remark:

Nastran outputs in the op2, xdb, punch or f06 files are obtained by converting the Marc output from the t16 file. If output is not available in the t16 file, or if it is in the wrong form, errors will occur during the t16op2 conversion. For most problems, if op2, xdb, f06 and/or punch output is required, it is best not to enter MARCOUT. In most cases MARCOUT should only be used if post-processing using the Marc t16 file is to be done.

## MARSHRII

Integer  $\geq 0$ , Default = 0, SOL 600 only.

Controls how the properties of bars added to the edges of CSHEAR elements when f1 or f2 of PSHEAR is nonzero.

- |     |   |
|-----|---|
| 0   | CBAR elements at the edges of CSHEAR elements (if any) will only have area (I1, I2, J are all 0.0). See PSHEAR Remark 2 as to how the area of these bars is calculated. |
| > 0 | The value specified will be used to calculate I1, I2 and J from the area as<br>$I1=I2=A/MARSHRII$ . $J = 2.0 * I1$  |

### Remark:

This parameter must be entered in the Case Control above any subcases.

## MARTET10

Integer, Default = 10, SOL 600 only

Controls how to treat badly shaped 10-node tetra elements in SOL 600 if renumbering the element does not correct problems.

- |    |  |
|----|--|
| -1 | Stop the analysis if all possible re-numbering options do not correct the bad element. See Remark 4.         |
| 0  | Continue the analysis with the best re-numbering option and let Marc possibly fail with inside out messages. |
| 1  | If all possible re-numbering options fail, change the bad tet 10 element to tet 4. See Remarks 1 and 5.      |
| 2  | If all possible renumbering options fail, deactivate the bad tet 10 element. (See Remarks 2, 3, and 5.)      |
| 3  | Change the coordinates of bad tet 10 midside nodes to line at the midpoints of the vertices. (See Remark 6.) |

### Remarks:

1. Changing bad tet 10 elements to tet 4 may ignore continuity on the edges of adjacent tet 10 elements unless they also get changed to tet 4 elements. If many tet 10 elements are changed to tet 4 elements, the accuracy of the stresses could be inaccurate. In addition, if the elements are used in contact, the same contact body can't have mixed elements with midside and without midside nodes.
2. Deactivating bad tet 10 elements creates holes in the model. If only a few bad elements are deactivated in a fine-grid model, the overall behavior will usually be nearly the same as if the elements were good and remained in the model, however the stresses near the deactivated elements may be too large. If many bad elements are deactivated, all analysis results will probably be inaccurate.
3. Setting PARAM,MARTET10,2 will deactivate all types of elements for which Marc gets inside out elements (not just tet 10 elements). This option maps to Marc's 10-DEACT parameter.

4. This parameter is available starting with MD Nastran 2010. Prior to this release, only option 0 was available.
5. Use options 1 or 2 with extreme care and only when all other alternatives including remeshing using the gui have been exhausted.
6. If martet10=3, any midside nodes whose coordinates are changed and lie on the boundary of the structure will redefine the exterior of the structure by a small amount. Also, if the element is used in contact and the stress-free option is invoked, the coordinates may be changed again and the elements may become badly shaped again.

## MARTETIN

Integer, Default = 0, SOL 600 only.

Controls whether additional information messages are output to the .f06 file or not when param,martet10 is set to a positive value.

- |   |   |
|---|---|
| 0 | Do not output additional tet10 information messages.  |
| 1 | Output additional tot10 information messages regarding checking of all tet10 elements. (The .f06 file could be very large using this option.) |

## MARUPDAT

Default = 1, SOL 600 only.

If this parameter is omitted, the Updated Lagrange method will be used if plasticity is involved. Ensure that all elements in the analysis are capable of using the Updated Lagrange method. If not, enter PARAM,MARUPDAT,-1. The Updated Lagrange Method is more accurate for many problems and also runs faster for some problems.

- |    |   |
|----|---|
| -1 | The Total Lagrange solution procedure will be used when Marc is executed from Nastran.  |
| 1  | The Updated Lagrange solution procedure will be used when Marc is executed from Nastran. This corresponds to Marc parameter update. (Default)                   |
| 2  | The updated Lagrange solution with large rotations for beam elements. This corresponds to Marc parameter (see Marc VOL C documentation for details): update,0,2 |
| 3  | The following Marc parameter will be set, in which LARGE DISP need not be specified elsewhere (see Marc VOL C documentation for details): update,0,2,1          |
| 4  | The following Marc VOL C parameter will be set, in which case LARGE DISP need not be specified elsewhere. (see Marc documentation for details): update,0,,1     |
| 5  | The following Marc parameter will be set (see Marc VOL C documentation for details): update,0,1   |

## MARVFCUT

Default = 0.0001, SOL 600 only.

Controls the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored. (Used in SOL 600 radiation heat transfer only.)

## MAUTOSPC

Default = -1, SOL 600 only.

Determines whether Marc AUTOSPC will be added.

-1	Do not add AUTOSPC. (Default)
1	Add AUTOSPC to Marc's parameter's section. It will remain throughout the run.
2000	Turn on AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option
2001	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1
2002	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2
...	
2999	Turn on AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999
3000	Turn off AUTOSPC by adding the integer 1 to Marc's model definition SOLVER option.
3001	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 1.
3002	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 2.
...	
3999	Turn off AUTOSPC by adding the integer 1 to Marc's SOLVER option in subcase 999.

## MAXAPL

Default = 1000

MAXAPL specifies the maximum number of applied loads to be used as a basis for residual vector calculations. If the number of applied loads exceeds MAXAPL, User Warning Message 9166 is issued, and residual vector augmentation is disabled. Note that the computing requirements for a large number of residual vectors can be prohibitive. See also, the RESVEC Case Control command.

## MAXDAMP

Default = 1000

See [BEIGRED](#).

## MAXIREV

Default = 1, SOL 600 only.

Determines whether to reverse axisymmetric coordinates ( $x \rightarrow y \rightarrow x$ ) and reorder nodes in the elements or not.

- 1 Do not reverse coordinates and renumber elements
- 1 Reverse coordinates and renumber grid ID's for each element. (Default)

### Remark:

Experience has demonstrated that certain models require the -1 option and others require the +1 option. The majority require the +1 option. If one of the options produces inside out messages in the first increment or other error messages, switch to the other option.

## MAXLP

Default = 5

MAXLP specifies the maximum number of iterations for element relaxation and material point subincrement processes in SOLs 129 and 159. MAXLP is 10 in SOLs 106 and 153 and cannot be changed by the user.

## MAXRATIO

Default = 1.E7

The ratios of terms on the diagonal of the stiffness matrix to the corresponding terms on the diagonal of the triangular factor are computed. If, for any row, this ratio is greater than MAXRATIO, the matrix will be considered to be nearly singular (having mechanisms). If any diagonal terms of the factor are negative, the stiffness matrix is considered implausible (non-positive definite). The ratios greater than MAXRATIO and less than zero and their associated external grid identities will be printed out. The program will then take appropriate action as directed by the parameter BAILOUT.

By default, in the superelement solution sequences the program will terminate processing for that superelement. A negative value for BAILOUT directs the program to continue processing the superelement. Although forcing the program to continue with near-singularities is a useful modeling checkout technique, it may lead to solutions of poor quality or fatal messages later in the run. It is recommended that the default values be used for production runs. A related parameter is ERROR.

The value -1 of BAILOUT causes the program to continue processing with near singularities and a zero value will cause the program to exit if near singularities are detected.

In SOLs 101 through 200 when PARAM,CHECKOUT,YES is specified, PARAM,MAXRATIO sets the tolerance for detecting multipoint constraint equations with poor linear independence. (See [Superelement Analysis](#) in *the MSC Nastran Reference Guide*.)

BAILOUT and MAXRATIO may be specified in the Case Control Section in order to provide different values between superelements but not between different boundary conditions.

## MBENDCAP

Default = 1, SOL 600 only.

Determines how PBEND internal pressure will be treated.

- 1 Stress stiffening and axial loading due to pressure is ignored
- 0 Stress stiffening and axial loading occurs (this usually happens with end caps)
- 1 Only stress stiffening occurs. (Default)
- 2 Stress stiffening and axial loading occurs for CBEND elements with any nodes having SPC or SPC1 and stress stiffening occurs for all other CBEND elements regardless whether the SPC/SPC1 is referenced above the subcase level or within a subcase.

## MCNLPARM

Default = 1, SOL 600 only.

Determines whether to make all field 5 (KMETHOD) consistent if more than one NLPARM entry occurs in the input stream. If mixed KMETHOD fields are provided for different NLPARM entries, SOL 600 can sometimes encounter difficulties.

- 1 KMETHOD will not be changed to be consistent.
- 1 KMETHOD will be changed to be consistent with that used with the NLPARM entry with the lowest ID. (Default)

### Remark:

This parameter may be set in RC files.

## MCON2D3D

Integer, Default = -1, SOL 600 only.

Controls whether a mixture of 2D and 3D contact will be allowed in a SOL 600 model. For most models a mixture of 2D and 3D contact will fail but there are a few cases where it will run properly. The default will stop the run should a mixture of 2D and 3D contact as specified in field 3 of the first BCBODY line is found. PARAM,MCONF2D3D,1 will allow the job to continue but the job may fail later in the run.

- 0 Stop job with a FATAL ERRO if a mixture of 2D and 3D contact is found.
- 1 Allow job to continue if a mixture of 2D and 3D contact is found.

## MCORDUPD

Default = 0, SOL 600 only.

Determines if the coordinates will be updated if one of the CONTINUE options is specified on the SOL 600 Executive Control statement.

- 0 Original coordinates are used for the continue option. (Default)
- 1 Coordinates are revised to reflect the final position after the nonlinear analysis. These updated coordinates are used in the continue option.

**Remark:**

If `mcordupd=1`, `parampmarct19,1` is automatically set and `jid.marc.t19` will be generated (`jid.marc.t16` may also be generated if desired).

**MCSSHLCK**

Default = 1, SOL 600 only.

Case Control parameter that determines whether to check that the thickness direction of CSSHL elements is in the direction pointing from grids 1-2-3-4 to 5-6-7-8 or not.

- 1 Do not make the check.
- 1 Make the check. (Default)
- 2 Make the check only for 6-sided elements (looking like chexa) and ignore for 5-sided elements (looking like cpenta).

**Remark:**

This parameter must be set in the Case Control above the first subcase.

**MCSSHORR**

Default = 1, SOL 600 only.

Case Control parameter that sets the orientation of the thickness direction of CEHXA converted to solid shells.

- 1 The smallest CEHXA direction will be determined and the element will be renumbered such that the thickness of the renumbered elements points from grids 1-2-3-4 to 5-6-7-8
- 1 The thickness direction will be in the grid 1 to 5 direction (thickness points from grids 1-2-3-4 to 5-6-7-8). (Default)
- 2 The thickness direction will be in the grid 1 to 2 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,5,6,2,4,8,7,3)
- 3 The thickness direction will be in the grid 1 to 4 direction (grid numbering that originally was 1,2,3,4,5,6,7,8 will be changed to 1,4,8,5,2,3,7,6)
- 11 The thickness direction will be along the side closest to the basic X-direction
- 12 The thickness direction will be along the side closest to the basic Y direction
- 13 The thickness direction will be along the side closest to the basic Z direction.

**Remarks:**

1. This parameter must be set in the Case Control above the first subcase.



2. This entry is only used in conjunction with the CSSHLM entry only.
3. This entry should only be used if all elements converted to solid shells are CHEXA, none should be CPENTA.
4. See the CSSHL entry for a description of grids 1 to 8.
5. For options other than mcsshorr=1 the material properties and material orientation direction must be orientated properly.

### MDAREAMD

Default = 1, SOL 600 only.

Option to modify or not modify all DAREA entries which are not associated with any other loads (DAREA entries that supply the actual load).

- 1 Do not modify any DAREA entries.
- 1 Modify all DAREA entries that are not associated with any other load entries and supply the actual loading. (Default)

### MDEFSEPP

Integer, Default = see below

This SOL 600 parameter allows an easy way to set the defaults for contact separation control. Since this parameter may be placed in RC files, the user can set default separation controls appropriate to the type of problems normally run and apply the same values to all models without having to enter them in the bulk data for each run. These values are normally entered on the BCPARA entry. The form of MDEFSEPP is as follows:

PARAM,MDEFSEPPAABBBBCD

where AABBBBCD is a packed integer field comprised of the following 4 controls:

	Variable	Standard Default	Values that can be Entered	Max Digits	Min Digits
AA	NODSEP	5	0 to 99	2	0
BBBB	MAXSEP	9999	0 to 9999	4	4
C	ICSEP	0	0 to 3	1	1
D	IBSEP	0	0 to 4	1	1

#### Remarks:

1. All values of AA, BBBB, C, D override the standard defaults.
2. NODSEP, MAXSEP, ICSEP, IBSEP entered on BCPARA override values on this parameter.
3. This parameter can be set in RC files.

4. Max digits is the maximum number of digits that can be entered.  
Min digits is the minimum number of digits that must be entered if this parameter is used.
5. A common setting for moderately difficult separation problems is MDEFSEPP=5001004  
AA=NODSEP=5  
BBBB=MAXSEP=10  
C=ICSEP=0  
D=IBSEP=4
6. A common setting for more difficult separation problems is MDEFSEPP=2999934  
AA=NODSEP=5  
BBBB=MAXSEP=9999  
C=ICSEP=3  
D=IBSEP=4

### MDK4OPT

Default = 1

Performance parameter used in MDACMS for the calculation of K4. If MDK4OPT>0 the most common value of GE for structure elements will be internally applied to PARAM,G to speed up the calculation of K4. Set MDK4OPT to 0 to disable this feature. The traditional K4 calculation will be used instead.

### MDOPT14

Default = NO

Option 14 of the MDENZO functional module modifies degree of freedom (DOF) based domain decomposition such that DOF belonging to any one grid point are retained in a single domain. This action (PARAM,MDOPT14,YES) lends stability to the resulting solution at the possible expense of some efficiency. Setting MDOPT14 to NO bypasses this operation so that DOF may be distributed to any domain without restriction. This parameter applies to Matrix Domain ACMS.

### MDOTM

Default = AUTO

The default selects the most efficient method for data recovery in DOMAINSOLVER ACMS (PARTOPT=DOF). There are two methods: (1) Output transformation matrix (OTM) method is selected if the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) exceeds the number of degrees-of-freedom at which displacements are required in order to complete data recovery (DISP, STRESS, etc.) This method also requires PARAM,SPARSEPH,YES which is the default. This method may be forced with PARAM,MDOTM,YES. (2) If the number of modes multiplied by PARAM,MDOTMFAC (Default = 20) does not exceed the number of degrees-of-freedom at which displacements are required in order to complete data recovery condition is not satisfied then the non-OTM method is selected. This method may be forced with PARAM,MDOTM,NO.

### MDOTMFAC

Default = 20

See MDOTM.

## MDREDOPT

Default = -1

This parameter is used in ACMS for matrix reduction operations. It affects the way in which matrices are reduced to modal coordinates. Quantities affected are damping, loads, and acoustic coupling. By default, the ACMS-generated Q-size matrices are reduced to H-size (modal coordinates). This is the most efficient method. If MDREDOPT=1, a triple matrix multiplication is used.

## MDUMLoad

Default = 0, SOL 600 only.

- 0 For subcases with no applied loads and enforced displacement or velocity of rigid contact bodies, a small magnitude dummy load will be created. Without these dummy loads, Nastran becomes confused and does not produce the correct contact information. If there is only one subcase, the dummy loads are not necessary. (Default)
- 1 Dummy loads will not be produced for any subcase and rigid contact with enforced motion may be incorrectly described.

## MECHFIL

Default = 1.E-6

Criterion for discarding massless mechanism modes with small generalized mass. A smaller value will result in more marginal constraint modes being retained. This parameter is used for PARAM,MMMETH,OLD. See also [MECHFIX](#), [MMMETH](#), and [MMFIL](#).

## MECHFIX

Default = AUTO

Control for fixing the massless mechanism problem. This capability is provided automatically for the default of this parameter, listed above. This capability is executed only when the eigensolution does not provide answers because of symptoms consistent with the presence of massless mechanisms. If MECHFIX is set to YES, the constraint modes are removed before attempting an eigensolution. When set to NO, the capability is blocked, the eigensolution will fail, and User Fatal Message 9119 will be printed in the F06 file. See also [MMFIL](#) and [MMMETH](#).

## MECHPRT

Default = NO

For SOL 103 only, if massless mechanisms are found the constraint modes are printed with a format similar to eigenvectors when this parameter is set to YES. They are labeled CONSTRAINT MODES, and are numbered sequentially. Grid points with only zero values in a mode are not printed. This parameter should be used when performing initial checkout of a model and a goal is to remove all massless mechanisms before starting production analysis. The number of each "mode" matches the corresponding GID,C pair in the high ratio message. If there are many (thousands) of such modes the output file will be large. There is no method to plot these shapes at present.

## MESH

Default = NO

If MESH=YES is specified, then a summary of the shading status and the subelement mesh for each CHBDYi element referencing a VIEW Bulk Data entry is printed.

## METHCMRS

Default = 0

In dynamic analysis (SOLs 103, 107, 108, 109, 110, 111, 112, 145, 146, and 200), METHCMRS specifies the set identification number of an EIGR or EIGRL entry to be used in the calculation of the normal modes on the v-set of the residual structure.

By default, the residual structure v-set normal modes will be computed based on the METHOD Case Control command selection as long as q-set is present.

## MEXTRNOD

Default = 0, SOL 600 only.

Determines whether extra grids will be added to SOL 600 parallel analyses.

- 0 Extra grids will not be added. (Default)
- 1 Extra grids will be added so that all grids from 1 to the highest grid are defined this was necessary for certain version of Marc prior to the 2005 version. All extra grids that are added have coordinates of 0.0 in all three directions.

## MEXTSEE,N

Default = 0, SOL 600 only, Case Control Parameter

Determines whether SOL 600 external superelement residual runs contain loads in the main input file or not.

- 0 External Superelement residual runs will not contain any loads. (Default)
- 1 External Superelement residual runs will contain all loads.

## MFASTCMP

Default = 1, SOL 600 only.

Determines default composite shell integration method.

- 1 Standard integration method is used. Failure, plasticity, thermal loads, etc. are all allowed. (Default)
- 2 A “fast integration” through the thickness technique is used which ignores thermal strains, plasticity, and temperature-dependent material properties (which should not be entered in the model).
- 3 A “fast integration” through the thickness technique is used which only ignores plasticity. Thermal strains and temperature-dependent materials are allowed.

**Remarks:**

1. To override the default integration method, use Bulk Data entry PCOMP.
2. If option 2 is used and temperature loading is present in the model, the option will automatically be re-set to 3.

**MFEA5701**

Default = 1, SOL 600 only.

Determines whether feature 5701 is written to the Marc file. When this feature is set to 1, it disables default rotation checking set to 0.001, which was initially set for early versions of Marc when RBE2 and similar elements were added because the 0.001 value proved to be responsible for convergence problems. For models that need rotation checking, you can enter the value using the NLSTRAT variable RLROTT. We recommend always using MFEA5701,1 to turn off the default 0.001 checking value in Marc whether or not the value is entered using NLSTRAT RLROTT.

- |   |   |
|---|---|
| 0 | Rigid rotation checking of 0.001 is turned on.            |
| 1 | Rigid rotation checking of 0.001 is turned off. (Default) |

**MFORCOR1**

Default = 1, SOL 600 only.

Option to correct forces entered twice (at the same node) in multiple subcases. This is not commonly found in the input files.

- |   |                               |
|---|-------------------------------|
| 0 | Do not correct the forces.    |
| 1 | Correct the forces. (Default) |

**Remark:**

This parameter can be set in RC files.

**MFORDUPE**

Default = 0, SOL 600 only.

Controls how duplicate forces encountered for the same load case are handled in SOL 600.

- |   |   |
|---|---|
| 0 | Forces will be translated to Marc as encountered even if duplicates are present. In the case of duplicates (more than one set of forces for the same node in the same subcase), all forces found will be translated directly to the Marc input. (Default) |
| 1 | If duplicates exist, only the last set of forces will be translated.  |

**Note:** To prevent confusion, it is suggested that duplicate forces not be used.

## MFSKIPPP

Integer, Default = 0, SOL 600 only.

Controls memory near the end of SOL 600 translations will be freed or not.

- 0 Memory is freed.
- 1 Memory is not freed (can save computer time, and, in some cases may be necessary for some models.)

## MGAPINIT

Integer, Default = 0, SOL 600 only.

Controls whether CGAP elements are initially open or closed (during Marc increment zero).

- 0 Gaps are initially open.
- 1 Gaps are initially closed.

## MGLUETOL

Real, Default = 1.0E-6, SOL 600 only

Determines the tolerance for the PERMGLUP option of the SOL 600 Executive Control statement for situation where the primary method (same as SOL 101 with permanent glue) fails to find any glued contact possibilities (for example, for edge contact will usually find no glued contact using SOL 101). If not glued contact is found using the primary method a secondary method, which can sometimes take a significant amount of computer time for large models is used. The second method evaluates all grids that have the same coordinates as the grids within the tolerance specified by this parameter. They will be tied together unless they have SPC's, MPC's or are grids attached to rigid elements.

## MHEATSHL

Default = 0, SOL 600 only.

Determines whether a membrane or thick shell element formulation is used for heat transfer. This parameter can be overridden by individual PSHELL entries. In the current release, the membrane elements should not be used for thermal contact analysis.

- 1 All "quad" elements in the model will have membrane capability regardless of MIDi values on the PSHELL entries.
- 0 All "quad" elements in the model will have membrane capability. (Default)
- 1 All "quad" elements in the model will have thick shell capability (shell sect, 1).
- 2 Shells use 2 dof per node, linear variation of temperature through thickness (shell sect, 2).
- 3 Shells use 3 dof per node, quadratic variation of temperature through thickness (shell sect, 3).
- >3 Shells use  $2 \cdot n_i + 1$  dof per node (where  $n_i$  is the value of MHEATSHL specified). (shell sect,  $2 \cdot (n_i - 2) + 1$ )

**Notes:**

1. Membrane capability in heat transfer means that the temperature is constant throughout the thickness, The MHEATSHL=0 option can be overridden by entering a non-blank value for MID2, MID3 and/or MID4 on an applicable PSHELL entry in which case the MHEATSHL=1 option will be used if no MHEATSHL parameter is entered.
2. Post options will be added to be the same as the value for Marc's shell sect. For example, if shell sect, 3 is generated, post codes 9,1 9,2 and 9,3 will also be generated.

## MHEATUNT

Default = 2, SOL 600 only.

Specifies the units for heat transfer using SOL 600.

- |   |                          |
|---|--------------------------|
| 0 | SI mm units used         |
| 1 | SI m units used          |
| 2 | US units used. (Default) |

**Note:** This parameter is used by Marc's ISOTROPIC (heat transfer) third datablock, fourth field.

## MHEMPIX

Default = 500, SOL 600 only.

Controls the number of pixels used in radiation heat transfer for SOL 600 using the hemi-cube method.

## MHOUBOLT

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | SOL 600 transient dynamics will use the single step Houbolt numerical integration method. (Default)            |
| 1 | SOL 600 transient dynamics will use the Newmark Beta numerical integration method.                             |
| 2 | SOL 600 transient dynamics will use the standard Houbolt numerical integration method.                         |
| 7 | SOL 600 transient dynamics will use the generalized alpha (Hilber-Hughes Taylor) numerical integration method. |

## MHRED

Default = YES

MHRED=YES, selects the c-set and r-set component mode reduction method suggested by Dr. Arya Majed and Ed Henkel. See the *MSC Nastran V2004 Release Guide* for details.

## MICRO

Default = 10, SOL 700 only

Defines the accuracy of the initial conditions in Eulerian elements, when using the geometrical shape definition.

### Format:

PARAM,MICRO,VALUE

### Example:

PARAM,MICRO,15

VALUE            Micro-zoning parameter. (Integer > 0)

### Remarks:

1. MICRO is the number of micro zones into which an element is subdivided during initial condition generation.
2. The default MICRO = 10 results in material fractions as accurate as 0.001. If a higher accuracy is required, a greater value for MICRO can be used, but the CPU time for the generation increases rapidly.
3. Micro zoning is only used when the initial conditions of the Eulerian material are specified on a TICEUL1 entry.

## MIDNODE

Default = 0, SOL 200 only

- = 0            High order element mid-side grid is updated by shape basis vectors.
- = 1            High order element mid-side grid is smoothed by algorithm 1.
- = 2            High order element mid-side grid is smoothed by algorithm 2 (most robust).

## MINCLDSB

Integer, Default = 0, SOL 600 only

Determines whether the SB value in field 4 of MATF will be included or not in the stiffness formulation of the model (Marc COMPOSITE datablock 3 field 4):

- 0            SB will not be included.
- 1            SB will be included.

### Remarks:

1. For releases prior to MD Nastran 2010, SB was not included.



2. If SB is included, Marc parameter TSHEAR will also be included.

## MINIGOA

Default = NO

Allows for the reduction in the amount of disk space used when using superelements. When this parameter is set to YES, the transformation matrix GOA will contain terms only for the degrees-of-freedom in the U5 (USET, USET1, SEUSET, SEUSET1) set. This can allow for a significant reduction in the amount of disk space used by the database. The limitation of using this approach is that data recovery will be available only for these degrees-of-freedom and elements connected to them.

## MINRECC, N

Default = 0, SOL 600 only.

N Integer. Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

Minimum number of iterations per load step. This is the same as MINREC on the NLSTRAT entry. If no other NLSTRAT values are entered, it is easier to enter this parameter. The value can range from 0 to 9. For certain problems, the value should be 2 or greater or accuracy will be poor.

## MINSOUTT

Default = 0, SOL 600 only

Determines elements that deform so much that they go inside-out in an analysis will be deactivated.

0 Issue an error message and terminate the job if an element goes inside-out. (Default)  
1 Deactivate any element that goes inside-out and continue the analysis. This option is particularly useful for models which allow failure (for example those using MATF).

### Remark:

param,minsoutt,1 maps to Marc's parameter IO-DEACT.

## MINVASHF

Default = 1.0, SOL 600 only

Inverse Power "auto shift" value. A new shift point (in frequency squared) is determined as the highest frequency squared plus this entry times the difference between the highest and next highest distinct frequency squared.

## MINVCITR

Default = 40, SOL 600 only

Inverse Power method, number of iterations.

## MINVCSHF

Default = 0.0, SOL 600 only

Inverse Power shift frequency in Hz.)

## MINVCTOL

Default = 1.0E-5, SOL 600 only

Inverse Power convergence tolerance.

## MINVFMAX

no Default, use MINVNMATCH, SOL 600 only

Real. Inverse Power max frequency to extract in Hz.

## MINVNMATCH

Default = 5, SOL 600 only

Integer. Inverse Power max number of modes to extract.

## MLDALLOW

Integer, Default = 0

Determines whether forces (FORCE, MOMENT, PLOAD4, etc.) are allowed in SOL 600 if their ID is the same as the value N of any Case Control LOAD=N entry.

- |   |  |
|---|--|
| 0 | Forces with the same ID are not allowed. |
| 1 | Forces with the same ID are allowed.     |

### Remarks:

1. MLDALLOW=0 is the only option available for most other Nastran solution sequences.
2. MLDALLOW=1 was the only option available in SOL 600 prior to MD Nastran 2010.
3. MLDALLOW=1 cannot be used with PARAM,MNASTLDS if MNASTLDS>0.
4. This parameter can be set in RC files.

## MLSTRAIN

Default = -1, SOL 600 only.

Corresponds to Marc's "LARGE STRAIN" parameter and optionally allows large rotation of beams and shells.

- |    |  |
|----|--|
| -1 | Small strain, small rotation analysis is used. (Default)   |
| 1  | Large strain and large rotation analysis of beams and shells is implemented in a way to automatically select the best options for a large strain analysis based upon the element type (see the following table). |

- 2 Use large strain and large rotation analyses of beams and shells for hyperelasticity and multiplicative plasticity with radial return mapping
- 11 Same as option 1 except beams and shells are limited to small rotations (see Remark 1.)
- 12 Same as option 2 except beams and shells are limited to small rotations (see Remark 1.)

Element Type/ Material Model	1-Dimensional	Plane Stress or Membranes or Shell Elements	Plane Strain or Axisymmetric, or 3- Dimensional Displacement Form	Plane Strain or Axisymmetric, or 3- Dimensional Herrmann Form
Conventional elastic-plastic	Updated Lagrange additive plasticity; no finite strain	Updated Lagrange additive plasticity; includes finite strain	Updated Lagrange additive plasticity; includes finite strain utilized constant strain	Updated Lagrange multiplicative plasticity; includes finite strain
Mooney, Ogden, Gent, or Arruda- Boyce	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange
Foam	Total Lagrange	Total Lagrange	Updated Lagrange	Updated Lagrange; incompressibility neglected

**Remarks:**

1. Starting with MD Nastran 2010, if MLSTRAIN=1 or 2, feature,2103 will also be added to provide the best large rotation solution of beam and shell models using updated Lagrange. To prevent feature,2103 from being used MLSTRAIN can be set to 11 or 12 instead of 1 or 2 respectively if large strain small rotation is desired.
2. Starting with MD Nastran 2010, if hyperelastic materials are found in the model, PARAM,MLSTRAIN,12 will be set automatically unless a different value of MLSTRAIN has been set by the user.

**MLSTRAN2**

Integer, Default = 0, SOL 600 only

Determines how the LARGE STRAIN option is written in the Marc input file for SOL 600.

- 0 LARGE STRAIN, (other options if applicable)
- 1 LARGE STRA, (other options if applicable)

**Remark:**

Since SOL 600 writes the Marc input file using large-field free-format input either option will work properly. If the input file is modified by hand and this entry is changed to fixed-format, the LARGE STRA option must be used.



## MMAT2ANI

Default = 2, SOL 600 only.

Determines how MAT2 will be mapped to Marc.

- 0 MAT2 will be mapped to Marc's ORTHOTROPIC option.
- 1 MAT2 will be mapped to Marc's ANISOTROPIC option.
- 2 MAT2 will be mapped as explained in the following note. (Default)

**Note:** The default, MMAT2ANI=2, maps MAT2 to Marc's ORTHOTROPIC option if G13 and G23 are both zero or blank and to Marc's ANISOTROPIC option if G13 and/or G23 are non-zero.

## MMBOLTUS

Integer, Default = -1, SOL 600 only

Controls how the top and bottom nodes are placed in the Marc "tying 69" input when MBOLTUS is used in a SOL 600 model.

- 1 Same as pre-MD Nastran 2010 versions tying 69 node order is top, bottom, control
- 1 Reversed - tying 69 node order is bottom, top control

### Remarks:

1. If it appears that a bolt preload may be backwards for bolts modeled using MBOLTUS, this parameter may be entered to reverse the mpc equations internally generated in Marc by tying 69 instead of reversing the loads.
2. For the MBOLTUS entry to be consistent with the SOL 400 BOLT entry PARAM,MMBOLTUS,1 is usually required.

## MMEMDETT

Default = -1, SOL 600 only.

Determines the Nastran portion of SOL 600 determines how much usable memory is available just before spawning Marc and informs Marc through the -ml flag. If Marc needs more memory than this value, it will go out of core.

- 1 Do not determine the amount of usable memory. (Default)
- 1 Determine the amount of usable memory and set the -ml Marc command line option to that value

**Remarks:**

1. Marc can determine this memory, however the computations determine the amount of installed memory and do not take into account memory used by the operating system or by other user processes.
2. This parameter may be entered in RC files.

**MMFIL**

Default = 1.E-10

Filter value used to distinguish between massless mechanism modes and rigid body modes. A smaller value may discard rigid body modes. The default value has been effective on all problems solved to date. This parameter is used for PARAM,MMMETHOD,NEW and PARAM,MMMETHOD,BETA. See also [MMMETHOD](#).

**MMMETHOD**

Default = NEW

Selects method for searching for massless mechanisms. The default value is generally the most reliable method. Use PARAM,MMMETHOD,BETA to invoke a new procedure that can sometimes find massless mechanisms when the default does not find any. Specify PARAM,MMMETHOD,OLD to use the original procedure. See also [MECHFIX](#).

**MNASTLDS**

Integer, Default = See Remark 1., SOL 600 only.

Option to determine complex forces using OLOADS datablock or the forces, moments, pressures, etc. in the Nastran input file. Positive values of MNASTLDS may also significantly speed up the SOL 600 translation and, in the case of RFORCE, the marc portion of the run will also usually be faster.

-1 or 0	Loads for Marc are translated or calculated directly from the forces, moments, pressures, etc. in the Nastran input file. (Remark 1.)
1	Complex loads and moments are obtained from the OLOADS datablock, pressures, rforce and other distributed loads are translated from the values found in the Nastran input. (Remark 3.)
4	Same as MNASTLDS=1 if the model has Bulk Data LOAD entries with more than one term (L2, L3, ... on the Bulk Data LOAD entry is defined), otherwise same as mnastlds=0.
99	Same as MNASTLDS=1 regardless of whether complex loads or Bulk Data LOAD entries with more than one term exist in the mode.
777	Same as MNASTLDS=99 except that all type of distributed loads (pressures, rforce, etc.) will be converted to force and obtain from the OLOADS datablock. (Remark 5.)

**Remarks:**

1. Default=99 if any GRID entry has field 7 defined. Default=777 if there are multiple subcases and RFORCE or PLOAD4 with continuation lines exist in the model or if any FORCE entry with nonzero (or blank) field 4 is defined, otherwise the default is 4.
2. MNASTLDS=-1 (or 0) was the only option prior to MD Nastran 2010.

- Complex loads are defined as those forces or moments whose grid appears more than once when the Case Control and Bulk Data LOAD entries are evaluated. For example consider the following situation:

```
....
SUBCASE 1
LOAD = 10
SUBCASE 2
LOAD = 20
BEGIN BULK
LOAD, 10, 1.0, 2.0, 1, 3.0,2
LOAD, 20, 1.0, 4.0, 1, 5.0,2
FORCE,1,51, ,1.0,0.0,0.0,100.0
FORCE,2,51, ,1.0,0.0,0.0,200.0
```

Although the force at grid 51 occurs for different load ID's when combined using the LOAD entries it actually occurs twice in each subcase. This is the simplest form of a complex load and can be handled by either MNA STLDS option. However, some models will have much more complex situation, even having different entries for the Fx, Fy and Fz directions. In such circumstances, particularly if multiple subcases are involved MNA STLDS > 0 is recommended.

- This parameter may be set in RC files.
- When MNA STLDS=777 pressures (and other forms of distributed loads) are converted to nodal forces and the follower force effects will be lost. However, for multiple subcases the follower force effects are not available in SOL 600 unless the loading is the same between subcases except for a scale factor.

## MODACC

Default = -1

- |     |  |
|-----|--|
| 0   | Selects the mode acceleration method for data recovery in dynamic analysis. See <a href="#">Formulation of Dynamic Equations in SubDMAP GMA</a> in the <i>MSC Nastran Reference Guide</i> for further discussion. If PARAM,SPARSEDR,NO is specified, then PARAM,DDRMM,-1 must also be specified. |
| 1   | Is the same as MODACC = 0 except if SUPORTi entries are present then the displacements are corrected for rigid body motion and the local flexibility at the SUPORTi degrees-of-freedom.  |
| ≥ 0 | Is not recommended for use in hydroelastic problems.   |

## MODEL

Default = 0

This parameter also allows several models to be stored in the same graphics database created by PARAM,POST,0.

## MOFFCORE

Default = 1, SOL 600 only.

Determines how memory for PARAM,MARCOFFT above is to be allocated (for increased speed).

- 1 Additional memory is not allocated.
- 1 Additional memory is allocated if available. (Default)

## MOP2TITL

Default = 1, SOL 600 only.

Determines how titles are placed on the 146 word record for op2 output records generated by SOL 600.

- 1 Standard titles will not be placed, however titles of the form “CQUAD4 STRESS FROM MARC”, and similar titles for other element types, strains, displacements, etc. will be written. This option is useful for certain postprocessors.
- 1 Standard Nastran title, subtitles will be placed on op2 files generated by SOL 600. This option is useful for postprocessors that require SOL 600 op2 data to be exactly in the same format as that generated by other Nastran solution sequences. (Default)

**Note:** The SOL 600 op2 file follows that of SOL 109 as closely as possible.

## MOUTRCOR

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 OUTR options are disabled or not if any CORD2i references another coordinate system.

- 0 Disable OUTR options
- 1 Do not disable OUTR options

### Remarks:

1. In some cases, OUTR results are inaccurate if a CORD2i references another CORD.
2. CORD2i entries are not checked to see if they are actually used in the model.

## MP1SET

Character “F” or “G” (default, see below)

Controls the integration set used in the calculation of MONPNT1 results. For the value of “F”, the integrated values will reflect any redistribution of loads caused by rigid elements and MPCs. This is particularly useful in the static aeroelastic solution sequences (SOL 144) when RBE3s are used to distribute mass, applied loads and splining. For the value of “G”, no redistribution occurs. The default is a function of the solution sequence.

SOL 144 - Default = "F", option "G" is not available.

All other sequences - Default = "G", option "F" can be specified by the user.

### **MPCX**

Default = 0

See OLDSEQ.

### **MPERMPRT**

SOL 600 only.

Determines whether to print permanent glue MPC's in the f06 file.

- |   |                                   |
|---|-----------------------------------|
| 0 | Do not print permanent glue MPC's |
| 1 | Print permanent glue MPC's        |

### **MPTUNIT**

Default=0

MPTUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of monitor points results in OUTPUT2 format in SOL 146. Default=0 indicates that no monitor point data are to be stored

### **MQUATERN**

Integer, Default = 2, SOL 600 only

Controls whether quaternions will be used for SOL 600 models with large rotation.

- |    |   |
|----|---|
| -1 | Do not use quaternion "feature" for large rotation  |
| 1  | Use quaternion feature,2103 which allows large rotations for total Lagrange or updated Lagrange |
| 2  | Use quaternion feature,2105 which allows large rotations only if updated Lagrange is used       |

### **Remarks:**

1. The use of MQUATERN = -1 allows backwards compatibility such that most models will obtain the same results using MD Nastran R3 as were obtained using MD Nastran R3.
2. The use of MQUATERN = 1 is believed to yield the most accurate large rotation results for most models.
3. With PARAM,MQUATERN,1 large rotation may be used both with updated Lagrange and total Lagrange.
4. mquatern = -1 does not set Marc feature 2103 (feature 2105 is not set).
5. mquatern = 1 sets Marc feature 2103



## MRADUNIT

SOL 600 only.

Controls the units used in radiation heat transfer for SOL 600.

- |   |   |
|---|---|
| 1 | Degrees Celsius                                   |
| 2 | Degrees Kelvin (default if parameter not entered) |
| 3 | Degrees Fahrenheit                                |

**Note:** Degrees Rankin are not available.

## MRAFFLOR, N

Default = N0, SOL 600 only.

- |   |   |
|---|---|
| N | Integer. If N=0, a new AF_flowmat file containing temperature-dependent stress-strain curves will be generated during the current Nastran execution and also used in the spawned Marc run. If N=1, an existing AF_flowmat file will be used. The name of the file is always determined by the value of PARAM,MRAFFLOW, but PARAM,MRAFFLOT determines if other characters are added. |
|---|---|

## MRAFFLOT, N

Default = N0, SOL 600 only.

- |   |   |
|---|---|
| N | Integer. If N=0, the file name as specified using PARAM,MRAFFLOW,Name will be used with no changes except that all characters will be in lower case and the extension “.mat” will be added. If N=1, the characters “asm_” will be added at the beginning of Name, the first character of Name will be upper case (the other characters of Name will be lower case) and the extension “.mat” will be added. This will make the AF_flowmat file name compatible with many names in Marc’s AF_flowmat directory. |
|---|---|

## MRAFFLOW, Name

No Default, SOL 600 only.

- |      |  |
|------|--|
| Name | Character. Name of a file containing temperature dependent stress versus plastic strain curves in Marc’s AF_flowmat format. This file can be generated from the current Nastran run using TABLEST and TABLES1 entries or a pre-existing file can be used depending on the value of PARAM,MRAFFLOR. The extension “.mat” will be added to Name. If this is a new file, it will be saved in the directory from which the Nastran execution is submitted. If a pre-existing file is to be used, it can either be located in the directory where the Nastran execution is submitted or in the Marc AF_flowmat directory. |
|------|--|

## MRALIAS ID (MALIAS02, MALIAS03, etc.)

No Default, SOL 600 only. This parameter is not usually used.

The purpose of the parameter is to map the advance nonlinear element type selected by the internal Marc translator in Nastran to a different type. For example, if the element type 75 for CQUAD4 is normally used, a mapping to advance nonlinear element type 139 could be made. ID is a 6 digit number. The left 3 digits are the element type normally selected by the translator and the right 3 digits are the element type to be mapped. In the above example, element type 75 is to be mapped to 139. The user would enter ID=075139. If element type 165 is to be mapped to element type 1 (which is not a real case), ID=165001. Consult Marc Volume B for a list of elements and their meaning. The user is responsible for ensuring that the mapping selected is proper. There is a limit of 18 aliases that may be entered in any model. Since Nastran can only accept one parameter with a given name, the second alias should be named PARAM,MALIAS02 and the third PARAM,MALIAS03, etc. All original element types mapped must actually exist in the model.

### Remark:

These entries should only be used if the Marc GEOMETRY entries are identical for the original and new element types.

**Note:** If you use one of the MRALIAS parameters, certain “parameters” in the Marc file may no longer be correct. For example, an element originally capable of using the updated Lagrange method may be aliased to one that must use the total Lagrange method. Such conditions are not checked by the translator when you use alias and you will need to make modifications to the Marc input file yourself to reflect them. To resolve this use PARAM,MLSTRAIN.

The Bulk Data entry, ALIASM, is available and subsequent version are more powerful.

## MRALLOGC

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for general Marc memory when Marc is spawned from Nastran. It specifies the initial allocation of “general memory”. This is used for storing element stiffness matrices and part or all of the matrix solver workspace among other things. Please note that element data like stresses and strains are not part of the general memory. Solvers 6, 8, and 9 use the main part of the workspace in separate memory. Initial allocation of the general memory can be used for avoiding reallocation (increase of the workspace). For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

## MRALLOCS

No Default, SOL 600 only.

Integer. The value entered here is the amount of memory (MB) allocated for Marc solver memory when Marc is spawned from Nastran. It specifies the initial allocation of memory for solver 8. By giving a value that is more than the maximum used during the run, one avoids that the solver workspace is increased (reallocated). This can be particularly useful for large contact jobs, where additional memory may be allocated due to

contact. If the given workspace is less than what is needed, it is automatically increased. This option is only for use with solver type 8. No check is done to see if solver type 8 is used in the job. For parallel processing the amount specified is the total for the job. It is divided by the number of domains used.

## MRBE3SNG

SOL 600 only.

Option to check the singularity of RBE3's in SOL 600. The real value entered is the singularity threshold allowed. If MRBE3SNG is entered as a positive value, all RBE3's with poor singularity values above the value entered will be output in the `jid.marc.out` file as warning messages. If MRBE3SNG is entered as a negative value, all RBE3's with poor singularity values above the absolute value entered will be output in the `jid.marc.out` file as error messages and the job will abort if they are found in increment zero, but if they are found after increment zero the messages are warning messages and the job will continue.

**Note:** A "good" RBE3 element using the original geometry in increment zero could become singular as the structure deforms in subsequent increments.

## MRBEAMB

Default = 0, SOL 600 only.

- |    |  |
|----|--|
| 0  | Write equivalent radius for all beams (see PARAM, BEAMBEA) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc's GEOMETRY values for beam type elements. (Default) |
| -1 | Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.   |

## MRBEPARM, IJK

Default = 0, SOL 600 only.

IJK provides settings for Marc's RBE parameter. If PARAM, MRBEPARM parameter is entered, Marc's RBE parameter will be set using this IJK. If PARAM < MRBEAMPM is not entered, PARAM, MARCRBE2 or PARAM, MARCRBE3 can be used to set Marc's RBE parameter.

IJK is a combination of three variables. For example 311. Descriptions for individual entries are as follows:

- |   |   |
|---|---|
| I | Enter 3 or 6 to control the number of degrees-of-freedom. For the dependent grid (reference grid) of each rbe2 or rbe3. The independent grids can have 3 or 6 dof and can be different than what is specified by I. |
| J | Enter 1 to use large displacement formulation of rbe2. Enter 3 to deactivate automatic convergence test for rbe2.   |
| K | Enter 1 to use large displacement formulation of rbe3. Enter 2 to activate non-normalized rotation constraint coefficient for rbe3.   |

**Note:** If IJK values other than specified above are entered, IJK will be set to zero and the parameter will not be used. This parameter should not be entered unless there are rbe2's or rbe3's in the model and they are to be used as such in Marc (rather than mpc or stiff beams).

## MRBDYCVT

Default = 0, SOL 600 only.

Determines if CHBDYG is converted to CHBDYE for SOL 600 heat transfer.

- |   |   |
|---|---|
| 0 | Do not convert. (Default)   |
| 1 | Attempt to convert CHBDYG to CHBDYE. All grids specified by all CHBDYG entries must reference actual conduction elements (CQUAD4, CHEXE, CTETRA, etc.) If even one CHBDYG does not reference an existing conduction element, this parameter will be re-set to zero. |

**Note:** If MRBDYCVT=0, CHBDYG will normally result in point heat transfer loads rather than distributed heat transfer loads.

## MRBIGMEM

Integer, Default = 0, SOL 600 only.

- |   |   |
|---|---|
| N | If N=0, memory allocations during loads translation phases are sized for computers with limited memory and swap space (paging space). Some large problems and/or unusual problems may not run. If this happens, use a newer modern computer with lots of memory and disk space (and lots of swap space) and set N=1. Larger memory allocations will then be available. This parameter is not usually required unless the available memory is extremely small. |
|---|---|

## MRBUKMTM

Default = 2, SOL 600 only

- |   |   |
|---|---|
| 1 | Buckling modes will be computed using the Inverse Power method.   |
| 2 | Buckling modes will be computed using the Lanczos method. Matrices must be positive-definite for this option. (Default) |
| 3 | Use Lanczos if EIGRL is specified, Inverse Power if EIGB is specified.  |

**Note:** MRBUKMTH should be specified if EIGR or EIGRL is used for buckling unless SOL 600,105 is the solution sequence. An alternative is to use ANALYSIS=BUCK. In SOL 600, it is not possible to compute natural frequencies and buckling modes in the same run. If ANALYSIS=BUCK is specified anywhere in the Case Control and if PARAM,MRBUKMTH is omitted, param,mrbukmth=3 will be set automatically.

## MRC2DADD

Default = 0, SOL 600 only.

Allows an offset to be added to all coordinates for 2D analyses so that X and Y will always be positive.

- 0 Offsets will not be added. (Default)
- 1 Offsets will be determined so that all Marc X and Y coordinates are positive (will exceed 0.1).

## MRCKBODY

Integer, Default = 1, SOL 600 only.

Determines whether each BCBODY will be checked to see if the 2nd line is present when more than 2 lines exist. If the 2nd line is missing for BCBODY entries with more than 2 lines, errors can occur.

- 1 BCBODY entries are not checked.
- 1 BCBODY entries are checked. If the 2nd line is not present for BCBODY entries having more than 2 lines, a 2nd line will be added internally.

### Remark:

This parameter can be set in RC files.

## MRCKLOAD

Integer, Default = 0, SOL 600 only.

Controls whether each subcase has LOAD or DLOAD specified. SOL 600 will usually not run correctly unless each subcase has some type of load (dummy or real). If a subcase only has a change in SPC, MPC, or contact to obtain the correct results a dummy load is usually required.

- 0 No check is made to see if each subcase has a Case Control LOAD or DLOAD entry.
- 1 A check is made to see if each subcase has a Case Control LOAD or DLOAD entry and if any subcase does not have such entry, a fatal error will be issued.

### Remarks:

1. This parameter must be entered in the Case Control above any subcases.
2. This parameter may be set in RC files.

## MRCONADD

Integer, Default = 1, SOL 600 only.

Controls whether PARAM,MRCONVER,11 will be added automatically if any BCTABLE with FBSH, BKGL and/or SEGS sub-headers is found. Also controls whether a BCTABLE with ID=0 will be added if one is not present in the input file.

- 1 Do not change value of PARAM,MRCONVER and do not add a BCTABLE with ID=0 if it is missing.
- 1 Change PARAM,MRCONVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Also add BCTABLE=0 if there are other BCTABLEs and non with ID=0 exists.
- 2 Change PARAM,MRCONVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found but do not add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.
- 3 Do not change PARAM,MRCONVER to 11 if any sub-headers FBSH, BKGL and/or SEGS are found. Add BCTABLE=0 if there are other BCTABLEs and none with ID=0 exists.

### Remarks:

1. PARAM,MRCONVER,11 is necessary if BCTABLE sub-headers FBSH, BKGL and/or SEGS exist in the model.
2. BCTABLE with ID=0 is sometimes necessary for certain contact problems. It never hurts to have one as it will be ignored for those contact problems where it is not needed.
3. This parameter may be used in MD Nastran 2010 and subsequent versions. The behavior of previous versions was the same as PARAM,MRCONADD,-1.
4. This parameter may be set in RC files.
5. This parameter, if used, must be placed in the Case Control above any subcases.

## MRCONRES

Default = 0, SOL 600 only.

For restart runs, Marc will produce “continuous” results files (t16/t19) which contain the results of the original run(s) as well as the results of the restart run. If Nastran postprocessing is requested to generate op2, xdb, etc. files, they will also contain the results from prior runs as well as the restart run.

MRCONRES=1, output results files will contain the results of the restart run only.

## MRCONTAB

Default = 0, SOL 600 only.

Determines whether CONTACT and CONTACT TABLE for SOL 600 use table-driven form or not.

- 0 Do not use table-driven form. (Default)
- 11 Use table-driven form of CONTACT and CONTACT TABLE

**Remarks:**

1. If MRCONTAB=11 then MRCONVER must also be set to 11
2. This parameter should is not necessary for SOL 600 heat transfer analysis and should not be set

**MRCONUSE**

Integer, Default = 0, SOL 600 only

Determines which increment for which contact MPC's will be used for SOL 600 jobs using a CONTINUE=101 to CONTINUE=159 (see the SOL 600 Executive Control statement, CONTINUE option).

- |   |   |
|---|---|
| 0 | The last increment will be used (remark). |
| N | Increment N will be used.                 |

**Remark:**

When models with contact and if BCPARA<INITCON,4 is specified, MPC's for each increment will be produced and placed in files named jid.marc.conmpc\_xxxx where xxxx ranges from 0001 to the number of increments in the solution. This parameter allows the user to specify which increment will be used in the continuation run spawned by the primary Nastran run after Marc finished. N must be less than or equal to the number of actual increments.

**MRCONVER**

Default = 0, SOL 600 only.

Determines version of Marc to use for CONTACT and CONTACT TABLE for structural analysis.

- |    |   |
|----|---|
| 0  | Uses enhanced Marc version 9 (Marc 2001 with added fields). |
| 11 | Users Marc version 11 (Marc 2005).                          |

**Remarks:**

1. This parameter only affects contact and only options 0 and 11 are available. It is not needed unless BCTABLE with FBSH and/or BKGL is entered.
2. This parameter may be set in RC files.
3. If PARAM,MRCONVER,11 is entered it is usually also necessary to enter PARAM,MRSRVER,11.

**MRCOORDS**

Default = 0, SOL 600 only.

Determines whether Marc COORD SYSTEM will be added if any CORD1i or CORD2i entries are in the model and if CBUSH elements, orientation vectors or other items requiring coordinate system transformation are present in the model or if PARAM,MRPLOAD4,2 is entered which signifies that PLOAD4 with continuation entries with CID, N1, N2, N3 are to be taken into account in the model.

- 0 Add Marc COORD SYSTEM. (Default)
- 1 Do not add MARC COORD SYSTEM

### MRCQUAD4

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CQUAD4 elements in SOL 600.

- 75 Advance nonlinear element type 75 is used. (Default)
- 139 Advance nonlinear element type 139 is used.

**Note:** Element 75 is capable of thick shell behavior and element type 139 is applicable to thin shells.

### MRCTRIA3

Default = 75, SOL 600 only.

Controls the “Default” advance nonlinear element type for CTRIA3 elements in SOL 600.

- 75 Advance nonlinear element type 75 with a duplicate node is used. (Default)
- 138 Advance nonlinear element type 138 with 3 nodes is used.

**Note:** Element 75 is capable of thick shell behavior and compatible with the default element 75 used for CQUAD4, but since one node is degenerate (repeated) stresses may not be accurate. Element 138 is accurate for thin shell behavior but may not be accurate for thick shell behavior. Currently there is no triangle thick shell in Marc.

### MRCWANGL

Default = 0.0, SOL 600 only.

Angle in degrees over which to rotate the cross-section about the beam axis to obtain its final orientation.

### MRDELTTT

Default = 3, SOL 600 only.

Determines how delta time is set for each “step” of a SOL 600 transient nonlinear analysis.

- 0 DTI=Ti/N
- 1 DTI=Ttot/N
- 2 DTi=min(Ttot/N, Ti/2.0)



- 3 DTi=min(Ttot/N, Ti/10.0) (Default)
- 4 DTi=min(Ttot/N, Ti/100.0)
- 5 DTi=min(Ti/N, Ti/2.0)
- 6 DTi=min(Ti/N, Ti/10.0)
- 7 DTi=min(Ti/N, Ti/100.0)

- SOL 600 transient dynamics is run by taking each pair of points on the applicable TABLED1 entry for the applied forcing function as a step. For example, if the following TABLED1 is given, the time steps will be as shown below:

```
TABLED1, 1
+, 0.0, 0.0, 1.0E-3, .2 4.0E-3, .3, 6.0e-3, -.3
+, 1.0E-2, 0.0
```

Step	Final Time	Delta Step Time (Ti)
1	1.0E-3	1.0E-3
2	4.0E-3	3.0E-3
3	6.0E-3	2.0E-3
4	1.0E-2	4.0E-3

If the entry for this model is (N=100, Ttot=1.0E-2)

DTi is the initial delta time (AUTO STEP (2,1) for the particular step, the six selections of MRDELTTT would give for the DTi values shown in the table below. Note that option 3 gives at least 10 points per step and the old option 0 is probably too conservative.

Step	Final Time	Time Step (Ti)	DTi for MRDELTTT							
			0	1	2	3	4	5	6	7
1	1.0E-3	1.0E-3	1.0E-5	1.0E-4	1.0E-4	1.0E-4	1.0E-5	1.0E-5	1.0E-5	1.0E-5
2	4.0E-3	3.0E-3	3.0E-5	1.0E-4	1.0E-4	1.0E-4	3.0E-5	3.0E-5	3.0E-5	3.0E-5
3	6.0E-3	2.0E-3	2.0E-5	1.0E-4	1.0E-4	1.0E-4	2.0E-5	2.0E-5	2.0E-5	2.0E-5
4	1.0E-2	4.0E-3	4.0E-5	1.0E-4	1.0E-4	1.0E-4	4.0E-5	4.0E-5	4.0E-5	4.0E-5

- If DTi is set using NLAUTO variable TINIT, this parameter will be ignored.

### MRDISCMB

Default = 0 without gravity, =1 with gravity, SOL 600 only.

Determines whether distributed loads, such as pressure, are combined in the Marc input file into one list if the magnitude of the distributed loads are the same. This parameter applies to MSC Nastran 2005 only. For previous MSC Nastran versions, MRDISCMB was 1. Option 0 save Marc memory and processing time.

- 0 Distributed loads with the same magnitude are combined.
- 1 All distributed loads are input individually. Must be used if multiple subcases with the same loadings are present in the model.

**Caution:** This parameter should be used with caution. For multiple load cases, MRDISCMB=0 may produce the wrong results particularly if gravity loading is present. MRDISCMB=1 must be used if multiple subcases with the same loadings are present in the model.

## MRDUPMAT

Default = 1 without gravity loading and -1 with gravity loading, SOL 600 only.

Controls whether to create duplicate materials for elements used by both the Marc pshell (PARAM,MRPSHELL,1 or the smear option on the SOL 600, ID statement) or not. If the user knows in advance that no materials are used by shells and any other types of elements, this step can be skipped to save considerable computer time.

- 1 Check for materials used by shells and other types of elements and create duplicate materials if any are used by both.
- 1 Do not check or create duplicate materials.

## MRDYNOLD

Default = 1, SOL 600 only.

Determines whether dynamic loads created by SOL 600 are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases or uses a new calculation method.

- 1 Dynamic loads are the same as in MD Nastran R3 (and MSC Nastran 2008) and prior releases.
- 1 Dynamic loads are “total loads” and may be more accurate for multiple subcases. (Default)

## MREIGMTH

Default = 1, SOL 600 only.

- 1 Eigenvalue analysis will be done in Marc using the Lanczos method. (Default)
- 0 Eigenvalue analysis will be done in Marc using the inverse power sweep with double eigenvalue extraction.
- 3 Eigenvalue analysis will be done in Marc using the inverse power sweep with single eigenvalue extraction.

## MREL1103

Default = 0, SOL 600 only.

- 0 Maps CQUAD4 to Marc's element type 11 for plane strain problems. (Default)
- 3 Maps CQUAD4 to Marc's element type 3 for plane stress problems.

## MRELRB

Default = 0, SOL 600 only.

- 0 If BCMOVE with the release option is specified, IDRBODY (see the BCMOVE entry) will refer to the actual IDs of BCBODY entries. (Default)
- 1 The release option of BCMOVE is used, IDRBODY (see the BCMOVE entry) will be in the order of occurrence of the BCBODY entries in the sorted Nastran Bulk Data file. For example, if there are two BCBODY entries with ID 12 and 22, the MRELRB=1 option means that you should specify IDRBODY on the BCMOVE entry using values of 1 and 2. If MRELRB=0, the IDRBODY values should be 12 and 22.

## MRELSPCD

Integer, Default = 0, SOL 600.

Controls whether SPCD for the second or subsequent subcase represents total (absolute) displacements or incremental displacements (relative to the displacement at the end of the previous subcase). SPCD for the first subcase is always total (absolute).

- 0 SPCD represents relative displacements (same as SPCR) for subcases 2 and above unless PARAM,MARCTOTT,1 has been set, in which case the SPCD's represent absolute displacements.
- 1 SPCD represents absolute displacements for subcases 2 and above (relative displacements for subcases 2 and above must be entered using SPCR).

### Remarks:

1. This parameter may be set in RC files.
2. PARAM,MRELSPCD,1 will automatically set PARAM,MARCTOTT,1 which means total loads using table-driven input will be used in the analysis. This is the only way total (absolute) displacements for subcases 2 and above can be used in Marc.
3. The behavior of SPCD prior to the MD Nastran 2010 release was to consider SPCD as relative displacements for subcases 2 and above.

## MRENUCLE

Default = -1 if parameter is not entered and MRENUCLE is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUCLE is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with largest element number less than approximately 20000). (Default)
- 1 All elements will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All elements will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the maximum element number is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUCLE on the SOL 600 statement).
3. MRENUCLE should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUCLE=1 an equivalence list of original and re-numbered element numbers is output on file elenum.txt.

## MRENUGRD

Default = -1 if parameter is not entered and MRENUGRD is not entered on the SOL 600 statement, SOL 600 only.

It is best if MRENUGRD is specified in the SOL statement. Some models will not have memory allocated properly if this parameter is placed in the bulk data.

- 1 No renumbering will occur (suggested for models with grid IDs less than approximately 20000). (Default)
- 1 All grid IDs will be renumbered and the new numbers will be used in the Marc analysis (see Remark 4.)
- 2 All grid IDs will be renumbered, the new numbers are used internally during translation however the original element numbers will be used in the Marc input file and Marc analysis.

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 statement (see MRENUGRD on the SOL 600 entry).
3. MRENUGRD should not be set both on the SOL 600 statement and as a parameter.

4. For MRENUGRD=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

## MRENUMBR

Default = 0, SOL 600 only.

Determines if both grid and element IDs for SOL 600 will be renumbered or not.

- |   |  |
|---|--|
| 0 | No renumbering will occur (suggested for models with largest grid ID less than approximately 20000). (Default)   |
| 1 | All grid and element IDs will be renumbered and the new numbers will be used in the Marc analysis. An equivalence list will be output on file grdid.txt                |
| 2 | All grid and element IDs will be renumbered internally during translation, however the original element numbers will be used in the Marc input file and Marc analysis. |

### Remarks:

1. This parameter can be set in RC files or on the SOL 600 statement.
2. To use models where the grid ID is greater than 9,999,999 this parameter must be set on the SOL 600 entry (see MRENUMBR on the SOL 600 statement).
3. MRENUMBR should not be set both on the SOL 600 statement and as a parameter.
4. For MRENUMBR=1 an equivalence list of original and re-numbered grid IDs is output on file gridnum.txt.

## MRESTALL

Default = 1, SOL 600 only.

Controls rotational restraints for solid element-only models.

- |   |  |
|---|--|
| 0 | SPCs for DOFs 4-6 will be ignored if entered in the Nastran file             |
| 1 | SPCs for DOFs 4-6 will be included if entered in the Nastran file. (Default) |

**Note:** This option might produce an input-data error in the Marc run but is sometimes required if RBEs or other special items are included in the model.

## MRESULTS

Default = 0, SOL 600 only.

If MRESULTS is set to 3, postprocessing of a previously-generated Marc t16 file to a results-only op2 file (normally called an f11 file) will be accomplished. OUTR=f11 and STOP=3 should also be set on the SOL 600 command line. This capability is available starting with MSC Nastran 2004.1.0. If MRESULTS=0 the t16 file from the current job will be processed if requested by OUTR options on the SOL 600 statement.

## MREVPLST

Default = -1, SOL 600 only.

Determines whether 2D plain stress triangular element node numbers will be reversed or not.

-1 or 0	Do not reverse the node numbers
1	Reverse the node numbers

## MRFINITE

No Default, SOL 600 only.

Controls Marc's FINITE parameter. If entered, with an integer value of 1, Marc's FINITE option will be employed. If this option is entered, parameters MRFOLOW1, MRFOLOW3 and MARUPDAT should also be entered. Other parameters to be considered are MARCDILT, MARCASUM and LGDISP.

## MRFOLLOW

Default = 0, SOL 600 only.

- 0 FORCE1, FORCE2, MOMENT1, MOMENT2 will act as non-follower forces. (Default)
- 1 Follower forces entered using FORCE1, FORCE2, MOMENT1, MOMENT2 will be mapped to Marc's new follower force option.
- 1 Follower forces will be turned off even if requested to be on using other options. This is sometimes necessary for multiple load cases where pressures are applied to different elements in the different load cases.

## MRFOLLOW2

Default = 0, SOL 600 only.

Controls whether Marc's follow for, 2 is used when multiple subcases are present.

- 0 Marc's follow for, 2 will not be used when multiple subcases are present. (Default)
- 1 Marc's follow for, 2 will be used when multiple subcases are present

## MRFOLOW1

No Default, SOL 600 only.

Controls the second field of Marc's follow for, N, M, L parameter (in this case, value N). If entered, options 1, 2, 3 and -1 are currently available. Enter:

- 0 If follower force due to distributed loads (pressure) is not to be considered.
- 1 If follower force stiffness due to distributed loads is not required.
- 2 If follower force stiffness due to distributed loads is to be included.

- 3 If the follower force for distributed loads is based upon the displacement at the beginning of the increment, as opposed to the last iteration.
- 1 If the undeformed geometry is required but total values of distributed loads are to be used (not presently available)

### MRFOLOW3

Default = 0 for statics and 1 for dynamics.

Controls the 3rd field of Marc's follow for N, M, L parameter (in this case, value M). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remarks.

#### Remarks:

1. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLOW3 must be 1 for SOL 600 dynamics and if entered as zero for dynamics will be reset to 1. If the user enters dynamic loads using MARCIN they should be in the form of total loads rather than incremental loads.
2. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLOW3 must be 1 for SOL 600 statics if PARAM,MARCTOTL,1 is entered. IF PARAM,MRFOLOW3 is entered as zero when PARAM,MARCTOTL,1 is entered, it will be reset to 1 if possible.
3. Starting with MD Nastran 2010 and MSC Nastran 2009, MRFOLOW3 must be 1 for SOL 600 if PARAM,MARCTOTT,1 is entered. If PARAM,MRFOLOW3 is entered as zero when PARAM,MARCTOTT,1 is entered, it will be reset to 1 if possible.

### MRFOLOW4

SOL 600 only.

Controls the 4th field of Marc's follow for N, M, L parameter (in this case, value L). This parameter is normally set internally by SOL 600 and should not normally be set by users, please see the following remark.

- 0 If follower force for point loads is not required. (Default)
- 1 If follower force for point loads is to be considered.

#### Remark:

To activate follower forces (concentrated loads) PARAM,MRFOLOW,1 must be entered as well as the forces using FORCE1, FORCE2, MOMENT1 and/or MOMENT2. PARAM,MRFOLOW4 should also be set to one.

### MRGAPUSE

Default = 0, SOL 600 only.

Determines whether Nastran CGAP elements will be approximated as Marc gap elements. The default is to fatal SOL 600 analyses if CGAP elements are found in the model.

- 0 Do not translate Nastran models using Nastran CGAP elements.
- 1 Translate Nastran models using CGAP elements. Marc gap elements are quite different than Nastran elements and usually can't be translated. In a few limited cases the Nastran and Marc gap elements are equivalent. It is up to the user to determine whether the gap Nastran elements can be used with SOL 600 or not. It is suggested that the user read Marc Volume A and C and run small test models to access each particular use of gaps. If gaps can be used, set PARAM,MRGAPUSE,1 and re-run the analysis. See also [MARCAPP](#), [MARCAPN](#), and [MARCAPD](#).

## MRHERRMN

Default = 1, SOL 600 only.

This entry controls whether extra grid created for such items as hyperelastic Herrmann elements, CWELD, et.c are output or not in the op2, f06, punch and/or xdb files. When Herrmann grids are output, the displacement value is actually pressure which might be confusing when looking at an f06 file.

- 1 Nodal output for extra grids is not provided.
- 1 Nodal output for extra grids is provided when the above files are requested. (Default)

## MRHYPMID

Default = 0, SOL 600 only.

- 0 Hyperelastic models with mid-side nodes will be translated to Marc including the mid-side nodes. Such models might run quite slowly. (Default)
- 1 Hyperelastic models with mid-side nodes will be translated to Marc leaving out the mid-side nodes. These models will normally run faster, but the displacements of the mid-side nodes will be zero and thus plots might look strange unless the model without mid-side nodes is read into the postprocessor using the .marc.dat or .t16 files.

## MRITYPE,

Default = 0, SOL 600 only.

Type of "constraint" used to connect the auxiliary nodes in all CWELDs.

- 0 RBE3 constraints will be used. (Default)
- 44 Kirchhoff constraints will be used
- 80 Combined RBE2 and RBE3 constraints will be used.



### MRMAT8A3, Value

Default = 1.0, SOL 600 only.

Value            If solid composites are modeled using MAT8, the third coefficient of thermal expansion, ALPHA3 is not defined. ALPHA3 is calculated as  $ALPHA3=value * ALPHA1$ .

### MRMAT8E3, Value

Default = 0.1, SOL 600 only.

Value            If solid composites are modeled using MAT8, the third modulus, E3 is not defined. E3 is calculated as  $E3=value * E1$ .

### MRMAT8N1, Value

Default = 0.5, SOL 600 only.

Value            If solid composites are modeled using MAT8, NU31 is not defined. NU31 will be calculated as  $NU31=value * NU12$ .

### MRMAT8N3, Value

Default = 1.0, SOL 600 only.

Value            If solid composites are modeled using MAT8, the NU23 is not defined. NU23 is calculated as  $NU23=value * NU12$ .

### MRMAXISZ

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as `-maxsize N` where N is the integer entered.

### MRMAXNUM

Default = Value in Marc include file in tools directory. SOL 600 only.

If this value is entered, the integer value will be used on the command line to run Marc as `-maxnum N` where N is the integer entered.

### MRMEMSUM

Default = 0, SOL 600 only.

If `MRMEMSUM=1`, a summary of memory used by the internal Nastran-to-Marc translator will be printed in the `f06` file. Each line will contain four numbers (all are in 4-byte words). The first number is the current memory request, the second the current memory (in addition to standard MD Nastran open core), the third is the memory limit with zero meaning no memory limit, and the fourth is the high water memory used so far by the Nastran-to-Marc translator and/or the `t160p2` results conversion.

This parameter must be entered in the Case Control.

### MRMTXKGG, Name

No Default, SOL 600 only.

**Name** Character. The NAME will be used for K2GG entries if entered. This parameter is similar to MRMTXNAM. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

### MRMTXNAM, Name

No Default, SOL 600 only.

**Name** Character. The NAME (field 2 of the DMIG entry) that will be used for DMIG values in a spawned Nastran execution. For example, a common name used frequently is K2XX. The Case Control command K2PP=Name will be added (in this example K2PP=K2XX will be added) at the end of the Case Control of the spawned job. DMIG entries with other names may exist on the file, but only those with NAME will be used in the spawned execution. Either MRMTXKGG or MRMTXNAM can usually be entered. The proper Case Control K2GG=NAME or K2PP=NAME will be selected automatically by SOL 600 as follows: If the continue option involves dynamic response analysis, K2PP will be used. If the continue option involves eigenvalue extraction or static analysis, K2GG will be used.

### MRNOCOMP

Default = 1, SOL 600 only.

Determines whether n-layer composite will be created if CORDM is defined on the PSOLID entry. This allows output in material coordinate systems.

- 1 Composite are not created (automatically activated for brake squeal). (Default)
- 1 1-layer composites are created.
- N N-layer composites are created.

## MRNOCOR

Default = 0, SOL 600 only.

0	Marc parameters ELASTICITY, PLASTICITY, UPDATE, LARGE DISP, CONSTANT DILATATION will be automatically adjusted as recommended by the Marc developers.
1	the above parameters will be adjusted as determined to be the most consistent in correlation between Nastran and Marc results for similar problems. This parameter should be entered for buckling problems without plasticity. PARAM,LGDISP,1 should also be entered.

## MRNOECHO

Default = 0, SOL 600 only.

Determines whether various outputs are placed in the Marc file.

0	Output is not suppressed (unless other entries are made to suppress it). (Default)
1	Suppress echo of nodes and element lists
2	Suppress echo of boundary conditions
3	Suppress echo of nurbs data

**Note:** Enter any combination to suppress whatever is desired. To suppress all items, enter 123.

## MRORINTS

Default = 99, SOL 600 only.

Controls orientation type for all solid elements in model (see Marc Volume C documentation, ORIENTATION option).

1	Edge 1-2
2	Edge 2-3
3	Edge 3-4
4	Edge 3-1
5	Edge 4-1
6	XY Plane
7	YZ Plane
8	ZX Plane
9	XU Plane
10	YU Plane

- 11 ZU Plane
- 12 UU Plane
- 13 UORIENT Define transformation matrix with orient.f user subroutine
- 14 3D ANIO
- 91 Composite orientations will be Edge 1-2 for shells and 3D ANSO for solids and all non-composites will use option 99. This option provides compatibility with earlier versions and requires that all referenced coordinate systems by composite elements be rectangular.
- 99 All orientations are specified by coordinate systems (CORD2R, CORD2C, etc.). (Default)
- 101-112 Shell elements will have options 1-12, solid element will have option 14. (See Note 2)

**Notes:**

1. For solid composites, it is necessary to use the PSOLID entry as well as the PCOMP entry. The PSOLID entry should normally have an entry in field 4 which specifies a CORDi entry to use for the material alignment direction.
2. If option 101 is picked, shells will have edge 1-2 and solids will have 3D ANISO. If option 102 is picked, shells will have edge 2-3 and solids will have 3D ANISO, etc.

**MROUTLAY**

Default = -9999, SOL 600 only.

Indicates which shell or composite layers are to be output using the MARCOUT Bulk Data entry or by default. Options are as follows:

- 9999 All layers will be output. (Default)
- N Layers 1 and abs(N) will be output. (N must not be -1.)
- 1 If the number of layers is less than 21, same as -N option. If the number of layers is greater than 21, layers 1, 11, 21, 31, ..., N will be output.
- 0 Top and Bottom layers only will be output.
- N Layers 1 through N will be output.
- 9999 Stresses at the element center only will be output (top and bottom are not output). This option produces output with the assumption that the element has constant stress and strain throughout the element.

**Remarks:**

1. Options -9999, -N, -1 and N also produce output at the element center.
2. MROUTLAY = 0 is the same as MROUTLAY = -1.
3. Do not enter this parameter if Bulk Data entry MAYOUT is used.



## MRPARALL

Default = 0, SOL 600 only.

Determines whether parallel processing for SOL 600 will be forced even if Nastran detects that it might fail prior to spawning Marc.

- 0           Spawn Marc in single-processor mode if Nastran detects Marc may fail in DDM mode. (Default)
- 1           Spawn Marc in DDM mode even if Nastran detects Marc may fail. Leave any COORD SYS entries in the Marc input file. If COORD SYS occur in the input file a DDM Marc job may fail depending on the version of Marc being used. Please read the following remark carefully before using this option.
- 2           Spawn Marc in DDM mode even if Nastran detects Marc may fail. Remove all COORD SYS entries from the Marc input file. Please read the following remark carefully before using this option.

### Remark:

Known situations where Marc does not presently work in parallel are as follows:

When local coordinate systems are specified for the following:

Field 7 of any GRID entry.

Material coordinate systems (for example field 8 of CQUAD4).

If CBUSH uses coordinate systems.

For the following cases, this situation can be avoided and PARAM,MRPARALL does not need to be entered:

If coordinate systems are used only for input (field 3 of the GRID entry) or just entered in the Nastran input but not actually used, enter PARAM,MRCOORDS,1 to allow parallel processing to proceed.

If coordinate systems are only used for shell orientation (such as field 8 of CQUAD4) and if all such coordinate systems are rectangular the following two parameters may be used to allow parallel processing.

PARAM,MRCOORDS,1

PARAM,MRORINTS,1

## MRPBUSHT

Default = 0, SOL 600 only.

Determines whether the requirement that when PBUSHT and contact occur in the same model, BCTABLE must be specified for each subcase.

- 0           This requirement is enforced and the job will abort if BCTABLE's are not entered
- 1           This requirement will be ignored and the job will proceed. Wrong results may occur and/or the job may terminate later in the run.

## MRPIDCHK

Integer, Default = 1, SOL 600 only.

Controls whether a check for duplicate PIDs (property IDs) will be made in SOL 600.

- 1 Do not make a check for duplicate PIDs.
- 1 Check for duplicate PIDs for structural elements - if any are found fatal the job.

### Remarks:

1. This parameter may be set in RC files.
2. This parameter may be used in the MD Nastran 2010 and subsequent versions.
3. Duplicate PIDs will usually cause SOL 600 to fail or obtain the wrong results if the duplicates occur for structural elements. It is usually acceptable to have a duplicate PID for a spring or damper and for a structural element.
4. For versions prior to MD Nastran 2010, SOL 600 behaved as if PARAM,MRPIDCHK,-1 was set.

## MRPELAST

Default = -1, SOL 600 only.

Determines whether PELAST will be skipped or cause the job to abort for SOL 600. SOL 600 does not support PELAST. PBUSHT along with CBUSH and PBUSH should be used instead.

- 1 SOL 600 jobs with PELAST that are referenced by any CELAS will abort.
- 1 PELAST entries will be skipped (ignored).

## MRPLOAD4

Default = 2, SOL 600 only.

Determines whether continuation entry for PLOAD4 will cause the run to stop or if the continuation line(s) are to be ignored.

- 0 The job will stop and a “Severe Warning” message will be issued.
- 1 The job will continue and the continuation lines will be ignored. A warning message will be issued for the first few such entries.
- 2 CID, N1, N2, N3 and SOLR will be used. If SOLR=LINE, the direction must be specified using the CID, N1, N2, N3 fields for SOL 600. LDIR is ignored. (Default)
- 3 If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

**Remarks:**

1. MRPLOAD4 should not be set to 2 for axisymmetric or plain strain analyses or if parallel processing is used unless the “single file Marc input file” option is used. In some cases, but not all, if MRPLOAD4=2 is set it will automatically be changed to zero one of these conditions exist.
2. See parameter MRPLOD4R for a companion control.
3. If any CSSHL elements are used, the pressures at the corners will be averaged and applied to the entire element for all element types in the model.

**MRPLOD4R**

Integer, Default = 0, SOL 600 only.

Determines how PLOAD4 pressures are treated in Marc when PARAM,MRPLOAD4,2 is set.

- |   |  |
|---|--|
| 0 | Pressures are treated as constant over the element if the pressure over the element varies, the average is applied.      |
| 1 | The Marc PLOAD4 style of loads is used which allows the pressures at the four corners of a quad surface to be different. |

**Remark:**

Versions of Marc including MDR3 and Marc 2008R1 and versions previous to these versions produce incorrect results for some models using MRPLOD4R=1 if total loads (specified by PARAM,MARCTOTL,1 or MARCTOTT,1) are used even if the pressure is the same at the four corners of each quad element.

**MRPLSUPD**

Default = 1, SOL 600 only.

Determines whether Marc’s PLASTICITY,3 will be used or not for fast integrated composites or smeared composites. Wrong results may be obtained if PLASTICITY,3 is used for these types of analyses even if the plasticity is in non-composite portions of the model (Default = 1 if this parameter is omitted).

- |    |   |
|----|---|
| -1 | Use PLASTICITY,3 if it is necessary for non-composite regions of the model                |
| 1  | Replace PLASTICITY,3 with UPDATE (FINITE and CONSTATN DIALITATION are not used. (Default) |

**Remark:**

SOL 600 determines if smeared composites are used by the presence or lack of the SMEAR word on the SOL 600,ID entry. Fast composites are determine if PARAM,MFASTCMP is set to 2 or 3 or if any PCOMPF Bulk Data entries are present in the model.

**MRPOISCK**

Integer, Default = 2, SOL 600 only

Controls whether to check if a “bad” Poisson ratio has been entered in SOL 600 for MAT1 entries.

- 0 Do not check or correct “bad” Poisson ratios.
- 1 Check if Poisson ratio is negative or > 0.5, if so change it to 0.333.
- 2 Check if Poisson ratio is negative or > 0.5, if so change MAT1 to MATORT with all three E values, the same as E for MAT1, all three G values the same as G for MAT1 and  $v12=v223=v31=0.333$

**Remarks:**

1. Current Marc versions do not allow isotropic materials with Poisson ratios having negative values or values greater than 0.5. Previous Marc versions may have allowed this.
2. “Bad” Poisson ratios may occur if the user specifies E and G and leaves the Poisson field on MAT1 blank.
3. If mrposick=1, the value of G will change. The value of E remains as originally specified.
4. If mrposick=0, the job will error out in the Marc phase of SOL 600.
5. This parameter is available starting the MD Nastran 2010.

**MRPREFER**

Default = 1, SOL 600 only.

Determines to output SOL 600 stresses on the t16 file in the standard Marc coordinate system for the element or the “preferred” (layer) coordinate system when the model contains composite elements.

- 0 Stresses are output in the standard coordinate for the element
- 1 Stresses are output in the “preferred” (layer) coordinate system. (Default)

**Remarks:**

1. The standard coordinate system for solids is usually the basic coordinate system. The standard coordinate system for shells and beams is usually the element coordinate. See Marc documentation volumes A and B for further details.
2. Some versions of Patran cannot plot stresses for MRPREFER=1.

**MRPRSAC**

Default = 0.0, SOL 600 only.

This parameter is primarily used by Versions of SOL 600 prior to MD Nastran R2 and MSC Nastran 2007 which could not support different pressures at the different corners, pressures applied in directions that are not normal to a surface or edge loads. This parameter is not necessary starting with the MD Nastran R2 and MSC Nastran 2007 Version 5 if PARAM,MRPLOAD4,2 is set.

Factor by which pressure loads are scaled for SOL 600. Prior to MD Nastran 2006 R1 and MSC Nastran 2006, Marc was not capable of handling different pressure applied to different corners of a surface. In other words, different values of p1, p2, p3, p4 on the PLOAD4 entry could not be handled. Approximations can be made by assuming the pressure is uniform over the surface if the surface is small enough. For SOL 600, the pressure (P) used by Marc is calculated using the following:





If p1 is not blank (or zero) regardless of the value of mrprsfac  
 $P = p1$

If mrprsfac is positive and p1 is blank or zero and one or more of p2, p3, or p4 are not zero or blank,  
 $P = mrprsfac * (p1 + p2 + p3 + p4)$  (This default if p2 or p3 or p4 are not zero or blank)

If mrprsfac is zero or negative,

If none of p1, p2, p3, p4 are blank or zero  $P = 0.25 * (p1 + p2 + p3 + p4)$

Otherwise,

$p = (+/-)pp$  where pp is the value largest absolute value of p1, p2, p3, p4 and P will have the proper associated sign.

### Remarks:

1. If MRPLOAD4=2, MRPRSFAC is ignored and p1,p2,p3,p4 are used as entered.
2. If MRPLOAD4 is not 2, the default for mrprsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrprsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by PLOAD4.

**Note:** The default for mrpsfac varies from element to element. For each element it is 1.0 divided by the number of Pi defined. For example, if only one of p1,p2,p3,p4 is defined, the default mrprsfac is 1.0. If two of p1,p2,p3,p4 are defined, the default for mrpsfac is 0.5. If three are defined the default is 0.33333 and if all four are defined the default is 0.25. When PARAM,MRPRSFAC is entered, the value is the same for all elements with pressure specified by pload4.

## MRPSHCMP

Integer, Default = 0, SOL 600 only

If MRPSHELL=1 and there is a Marc PSHELL and composite for the same elements, this option controls which will be used.

- 0 Use Marc composite and ignore Marc pshell. (Default)
- 1 Use Marc pshell and ignore Marc composite.
- 2 Do not make any corrections (this works for some models and fails for others)

## MRPSHELL

Default = 0, SOL 600 only.

Used to control shell property specifications for Marc in SOL 600.

- 0 Classical GEOMETRY options will be used for all shell elements. (Default)
- 1 The Marc PSHELL option will be used, (can only be used with CQUAD4, CQUAD8, CTRIA3, and CTRIA6).

**Important:** If MRPSHELL=1, shell elements will use Marc's new PSHELL option. No materials used by PSHELL may be referenced by other types of elements that this option has used. For commonly used elements, duplicate materials will automatically be created by SOL 600 to satisfy this criteria, however this can take considerable computer time. If the user knows that no materials are used by both PSHELL and other properties, he can set PARAM,MRDUPMAT,-1 to bypass the checking and creation for duplicate materials. To speed up this process, set PARAM,MSPEEDPS,1. The Marc PSHELL formulation is more stable if MID3=MID2. To set MID3=MID2, set PARAM,MARCMID3,1.

### MRRBE3TR

Default = 0, SOL 600 only.

Controls whether all translations or all translation and rotations will be used for the reference degrees of freedom or whether fewer than 3 are allowed.

- 0 Fewer than 3 reference degrees-of-freedom may be used. (Default)
- 3 All RBE3's with REF C (field 5 of the RBE3 entry) not having 123 will have all translations for the Marc REFC (field 5 of the Marc RBE3 entry) set to 123.
- 6 All RBE3's with REF C (field 5 of the Nastran RBE3 entry) not having 123456 will have all translations and rotation for the Marc REFC (field 5 of the Marc RBE3 entry) set to 123456.

#### Remarks:

1. Marc frequently gives a singular tying message if there are not at least degrees of freedom 123 for REFC. This sometimes results in poor singularity ratios and incorrect results for nonlinear analyses. It also can prevent eigenvalues from being calculated.
2. It is recommended that option 3 or 6 be used for all SOL 600 analyses unless the user is certain that REFC can have fewer than three dof's for the particular analysis.

### MRRCF, RCF

No Default, SOL 600 only.

- RCF Character. Name of RCF file name (limited to 8 characters) used in conjunction with another MSC Nastran run spawned from an original MSC Nastran run as specified by the CONTINUE option on the SOL 600 command. The RCF file may contain any information required (such as scratch=yes, exe=, etc.) as discussed in [Executing MSC Nastran, 2](#) of this guide. This rcf file does not have to use the same options as the primary rcf file and should normally set batch=no as one of the options.

## MRRELNOD

Default = -1, SOL 600 only.

Controls enforced displacements using SPCD.

- |    |  |
|----|--|
| 0  | For multiple subcases with SPCDs, the SPCDs from the previous subcase will be released at the start of the current subcase.  |
| 1  | SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be released gradually during the current subcase.   |
| -1 | SPC's and SPCD's from the previous subcase, not specified again in the current subcase, will be subtracted out. For example, if the previous subcase applied a SPCD of 0.1 to a particular dof, the current subcase will apply -1.0. This will bring the displacement of that dof to zero for the current subcase. |

## MRRSTOP2

Default = 0, SOL 600 only

Integer, Normally op2, xdb, punch and f06 output is not available for SOL 600 restart analyses. Setting this parameter to 1 will allow the program to attempt to create one or more of these files. Only in limited cases will the job be successful.

- |   |   |
|---|---|
| 0 | Do not attempt to create .op2, xdb, punch, f06 output for restart runs (default if parameter is not entered). (Default) |
| 1 | Attempt to create .op2, xdb, punch, f06 output as specified by other options.   |

## MRSCMOD

Default = 0.0, SOL 600 only.

Solution scaling factor for linear buckling analysis (SOLs 600, 105) using the Lanczos method. If the applied load in the first subcase is too large, the Lanczos procedure may fail. This number may be used to scale the solution for numerical reasons. The collapse load will be output based upon the total load applied.

### Remark:

This parameter applies to rigid surfaces described using 4-node patches only.

## MRSETNA1, N

Default = Program calculated, SOL 600 only.

- N        If this parameter is entered with  $N > 0$ , the integer value entered will be used in Marc's SETNAME parameter section as the first value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

## MRSETNA2, M

Default = Program calculated, SOL 600 only.

- M        If this parameter is entered with  $M > 0$ , the integer value entered will be used in Marc's SETNAME parameter section as the second value of SETNAME,N,M which is an undocumented Marc option. N is the number of sets and M is the largest number of items in any set. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. Both MRSETNA1 and MRSETNA2 must be included for either to take effect. This option is no longer required for MSC Nastran 2005 r2 and subsequent releases.

## MRSETNAM, N

Integer, Default = Program calculated, SOL 600 only.

- N        If this parameter is entered with  $N > 0$ , the value entered will be used in Marc's SETNAME parameter section. This entry is sometimes necessary if large lists of elements or nodes are used to describe materials, properties or contact. To get around a Marc bug, if a computed setname value is large, Nastran will normally use the undocumented form (see MRSETNA1). To prevent this, set N to -1.

## MRSPAWN2, CMD

Default = Nastran, SOL 600 only.

**CMD** Character. Name a command to run D Nastran (limited to 8 characters single field or 16 for large field) used in conjunction with the CONTINUE options on the SOL 600 command. The Nastran run to be spawned will have the form:

```
CMD jid.nast.dat rcf=RCF
```

where file RCF is provided by PARAM,MRRCFILE,RCF. As an example, if CMD is nastran, jid is myjob (original file myjob.dat) and RCF=nast.rc, the spawned run will execute using:

```
nastran myjob.nast.dat rcf=nast.rc
```

### Notes:

1. See PARAM\*,HEATCMD for SOL 600 thermal contact heat transfer analysis.
2. CMD will be converted to lower case regardless of the case entered.

## MRSPLINE

Default = 2, SOL 600 only.

Determines whether RSPLINE will be converted to CBAR or to MPC's.

- 1 RSPLINE will ignored (this option should only be used for model checkout)
- 1 RSPLINE will be converted to stiff CBAR elements
- 2 RSPLINE will be converted to MPC's (Marc's servo link) using the same equations used by other Nastran solutions, except that the MPC's are capable of large rotation. This option requires somewhat longer translation times than the others but can save computer time during the Marc execution phase. (Default)

## MRSRING

Default = 0.0, SOL 600 only.

Specifies a stiffness value to be added to the main diagonal of each translational term of the stiffness matrix. This option is useful in nonlinear static analysis with 3D contact of two or more separate structures. Some of the pieces may not be grounded until contact occurs. By adding a small spring to ground, such as K=1.0, these pieces are stabilized until contact occurs. This option applies to SOL 600 only and generates SPRINGS in the Marc input file for all nodes in the model and all three translational degrees-of-freedom. IF K is entered as a negative number, the absolute value of K will be added to all 6 degrees of freedom of each grid in the model.

If the run is for heat transfer and K is positive, the spring will only be added to DOF 1.

## MRSRVER

Integer, Default = -1, SOL 600 only

Controls how CELAS and all other items map to the Marc input data.

- 0 Non-table form is used which generally means the Marc input file version is less than 11.
- 11 The table form for SPRINGS is used which requires two lines, the first with spring rates and the second with table ID's. PARAM,MRSRVER,11 is usually required if PARAM,MRCONVER,11 is entered.

## MRSTEADY

Default = 1, SOL 600 only.

Controls the solution method for SOL 600 steady state heat transfer.

- 1 Marc STEADY STATE is used with TIME STEP of 1.0. (Default)
- 2 AUTO STEP is used.

## MRT16STP, N

Default = 0, SOL 600 only.

Enter in Case Control at the subcase level.

- N Number of load increments to put on the t16/t19 files for each subcase when the AUTO STEP method is used. The default of 0 puts all adaptive increments on the t16/t19 file. If a value of N is entered, load steps for times=tmax/N will always be introduced into the auto stepping process and the t16/t19 files will have outputs only at zero and those times. This corresponds to field 1 of Marc's AUTO STEP 2nd option and can also be set using the NLAUTO option. If this is the only non-default NLAUTO variable to set, it is more easily accomplished using this parameter. It is suggested that this parameter always be used for large models and that N be 10 or greater, otherwise the size of the t16/t19 files may become very large. This is especially important for Windows systems, which presently has a 4GB limit in converting t16 to op2 files due to compiler limitations.

## MRTABLIM

Integer, Default = 0, SOL 600 only

Determines the maximum number of points in any TABLES1 used when tables are specified or MATHE input.

- 1 Unlimited, not recommended - If used could lead to numerical errors or program aborts
- 0 The limit is 500 points.
- N The limit is set to N points. It is recommended that N does not exceed 500.

**Remark:**

If more than about 500 points are used for the tables, the curve filling processes used to determine the material constants may encounter numerical errors and the program could abort. If a table with more than the number points specified by this parameter is entered, the number of points will be reduced to the specified limit so that numerical errors are unlikely to occur.

**MRTABLS1**

Default = 0, SOL 600 only.

TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves according to the following formulas for the value entered for MRTABLS1. In the formulas *s* is the stress entered for TABLES1, *e* is the strain entered in TABLES1, *S* is the Marc WORK HARD stress and *E* is the Marc WORK HARD Plastic strain. *e<sub>y</sub>* is the yield strain (*s<sub>y</sub>*/*E*) where *EE* is Young's modulus, *s<sub>y</sub>* is the yield stress. The first point of the Nastran curve will be skipped unless MRTABLS2 is set to 1.

0 (Default)	$S=s$ $E=e - e_y$
1	$S=s(1+e)$ $E=\ln(e+1)$
2	$S=s$ $E=e - s/E$
3	$S=s(1+e)$ $E=\ln(1+e) - s/EE$
4	$S=s$ $E=e$
5	$S=s$ $E=\ln(1+e)$
6	$S=s$ $E=\ln(1+e-s/E)$
7	$S=s(1+e)$ $E=\ln(1+e)-S/EE$
8	$S=s(1+E)$ $E=\ln(1+e)$
9	$S=s(1+E)$ $E=\ln(1+e)-\ln(1+e_y)$

**Note:** This parameter (like any other parameter) can only be entered once in an input file.

## MRTABLS2

Default = 0, SOL 600 only.

- |   |   |
|---|---|
| 0 | TABLES1 stress-strain curves are converted to Marc WORK HARD stress-plastic strain curves starting with the yield point. The first point will be skipped. (Default) |
| 1 | All points in TABLES1 will be connected to WORK HARD, however, the first plastic strain will be set to zero if entered as non-zero.                                 |

**Note:** This parameter (like any other parameter) can only be entered once in an input file.

## MRTABPRC

Integer, Default = 0, SOL 600 only

Determines the precision of stress-strain curves when translated from Nastran to Marc using SOL 600.

- |   |   |
|---|---|
| 0 | Standard precision (8 significant figures) is used. |
| 1 | Maximum precision (14 significant figures) is used. |

### Remarks:

1. This parameter is available starting with the 2010 entry. For MD Nastran 2010 it only applies to stress-strain curves entered in conjunction with MATEP.
2. Prior to MD Nastran 2010, only option zero was available.
3. In most cases, the stress-strain values are not known to even 8 significant figures. However, if the stress-strain is calculated from other data the extra precision might be useful.

## MRTFINAL

SOL 600 only.

Real. In some dynamic problems, due to numerical roundoff an extra time step with very small initial and final times is generated. If these times are too small, Marc sometimes will diverge even though the “final time” actually desired by the user has been reached within reasonable accuracy. MRTFINAL is a value below which this extra step will be eliminated to prevent excess computations and/or possible divergence. If this parameter is not entered, the default value is 1.0E-8. This parameter is only used with the AUTO STEP procedure. The ‘extra’ step will also be eliminated if the initial time step is less than MRTFINAL/100.0.

## MRTHREAD

Integer > 0, Default = 1, SOL 600 only.

Determines the number of threads to be used by the run. For multi-core processors speed can frequently be gained by specifying a number of threads equal to the number of cores assuming the job has exclusive use of the machine.



- 1            Use 1 thread.
- 2            Use 2 threads.
- N            Use N threads.

**Remarks:**

1. This parameter may only be used by solver types 8 (multifrontal direct sparse) and 11 (Pardiso).
2. This parameter may be combined with PARAM,MARPROCS to use several processors each using several threads.

**MRTIMING**

Default = 0, SOL 600 only.

If MRTIMING is 1, timing summaries for various portions of the internal Marc translator will be provided in the f06 and log files.

**MRTSHEAR**

Default = 0, SOL 600 only.

By default, parabolic shear deformation is not included in the formulation of beam and shell elements if there are no composite elements. If composite shell elements are present, parabolic shear deformation is included by default. This parameter can be used to override these defaults.

- 1            Parabolic shear deformation is included in the formulation of beam and shell elements.
- 1 or 0      Parabolic shear deformation is not included even if composite elements are present.  
(Default)

**Remark:**

This parameter maps to Marc TSHEAR parameter.

**MRUSDATA**

Default = 0, SOL 600 only.

Determines whether user subroutine USDATA will be activated and if so the size of data.

- 0            User subroutine USDATA is not active in the current run. (Default)
- N            User subroutine USDATA will be active the in the current run and N is the number or real\*4 words needed for the data stored in common block /usdam/

**Remark:**

In addition to adding this parameter, the user must enter USDATA on Bulk Data entry USRSUB6.

## MRVFIMPL

Default = 0.01, SOL 600 only.

Controls the fraction of the maximum view factor that is to be treated implicitly (contribute to operator matrix). View factor values smaller than this cutoff are treated explicitly.

## MRV09V11

Default = 1, SOL 600 only.

Determines whether certain Marc “features” which are default in Marc “version 11” are added for SOL 600-generated models that use “version 9”.

- 1 Do not add the features.
- 1 Add the following features:
  - feature,4703 to speed up DDM jobs for one-processor jobs, it has no effect)
  - feature,5701 to disable old rigid rotation checking which was too stringent
  - feature,601 to improve contact
  - feature,5301 to improve deformable-deformable contact
  - feature,3201 to improve contact friction types 6 and 7
  - feature,5601 to improve thickness updating when the updated Lagrange method is used
  - feature5801 to improve in-plane bending of advance nonlinear element type 140
  - feature,6001 to improve concrete cracking analysis

<b>Note:</b>	The above features are only used for certain problems, even though all are included with the default option, they have no effect on models that do not use advance nonlinear element type 140, feature 601 has no effect on models that do not have contact, etc.
--------------	---

## MSG10625

Default=0

- 0 USER FATAL MESSAGE 10625 will stop any frequency response run **with frequency dependent elements**, with the DMAP path **PARAM, BUSHNM, YES** (default) if multiple subcases with different Case Control **FREQ=n** are detected.
- 1 The run will be allowed to proceed.

The frequency dependent path BUSHNM, NO path is only viable when the frequency dependences are specified on the PBUSHT, PELAST, PDAMPT, PAABSF.

When the frequency dependences include MATiF entries, only the path PARAM, BUSHNM, YES path is viable. This path is also viable only for new features such as extended GEij on MAT2 or MAT9 entries, or the use of PDISTB/PDISTBM entries. The frequency dependences provided by PBUSHT, PELAST, PDAMPT, PAABSF also work for this path. Also, since the required element level  $\Delta K$ ,  $\Delta K4$ ,  $\Delta B$  matrices required for frequency dependency are computed in a completely different manner between PARAM, BUSHNM, YES/NO paths, the GPFORCE calculation is only available in the new PARAM, BUSHNM, YES path.

When the path PARAM, BUSHNM, YES has any type of frequency dependency, several new solution techniques are turned on. These include, by default, the use of master frequencies and extended residual vector calculations. Also, via PARAM, NMNLFREQ, value, in modal frequency the eigenvalue analysis may be performed using material stiffness values other than the base values defined on the basic MATi entries.

Automatic Master frequencies, controlled via the Case Control MFREQ, are based on the presence of MAT1F, MAT2F, MAT8F, MAT9F, PBUSHT, PELAST, PDAMPT entries and their associated TABLEDi entries.

For each forcing frequency corresponding to a determined master frequency, MSC Nastran forms complete element stiffness, structural damping and viscous damping matrices based on material and property table lookup at that frequency.

For forcing frequencies between master frequencies, the fully assembled g or modal sized stiffness, structural damping, and viscous damping matrices are computed by interpolation.

When multiple subcases are present, and these subcases have different FREQ=n selection, MASTER frequencies, computed based on the first subcase, may not have good correlation with forcing frequencies in subsequent subcases. Thus, USER FATAL MESSAGE 10625 will be issued by default.

## MSIZOVRD

Default = -1, for small models and +1 for large models, SOL 600 only.

Determines whether Marc SIZING values for large models will be updated or not. Nastran includes Marc directories with a file named “include” in the tools directory which includes a line MAXNUM=N, where N is some value like 1000000. If the maximum number of nodes or elements in the model exceeds N, memory overwrites or job aborts are possible unless either N is set larger than the actual max node or element number in the model or the values are specified on the SIZING entry (field 3 for max element and field 4 for max node).

- 1 The sizing entry will not be updated (either the model is not large, MAXNUM in the include file has been updated or parameters such as MARCSIZ3 and/or MARCSIZ4 have been entered into the run to provide values that are large enough. (Default)
- 1 If the number of nodes or elements in the model exceeds 1,000,000 the sizing entry will be updated to the max number of nodes and elements actually in the model. Extra nodes and or elements to account for welds, pinned members, Herrmann elements, etc. will be included.

**Note:** For PARAM,MSIZOVRD large models are considered to be those with more than 1,000,000 nodes and/or elements.

## MSOLMEM, MBYTE

Default = Program determined value, SOL 600 only.

**MBYTE** If entered, the integer value entered here is the 8th field of Marc's SOLVER option, and is the maximum memory in Mega Bytes that can be used by Marc's solver types 6 (hardware provided direct) and 8 (multi-frontal direct, which is the default solver for SOL 600) before going out of core. This parameter is the same as the MBYTE field on the NLSTRAT entry may be easier to enter as a parameter if no other NLSTRAT values are needed.

**Note:** This parameter should only be entered if NLSTRAT entries are not required. If any NLSTRAT entries are made, use the MBYTE field instead of this parameter.

## MSPCCCHK

Integer, Default = 0, SOL 600 only

Effects multiple subcases where SPCD's are applied and SPC's potentially change between the subcases.

- 0 A comprehensive check is made to ensure that all SPC changes are accounted for all subcases.
- 1 The check is not made. The check can take significant computer time and in most cases it is not actually needed. If the user knows it is not needed, computer time can be saved by turning it off.

### Remarks:

1. This parameter is available starting with MD Nastran 2010.
2. This parameter may be set in rc files.

## MSPCKEEP

Default = 0, SOL 600 only.

Determines which SPC's entered in the bdf file are retained in the Marc input file for 2D models.

- 0 Only retain those dof's that are normally associated with the type of 2D analysis (normally 2 translations) (Default)
- 1 Retain all dof entered in the bdf file in the Marc input file.

### Remark:

Some types of Marc 2D analyses support one rotation in addition to two translations and it is important to restrain this rotation.

## MSPEEDCB

Default = -1, SOL 600 only.

Determines whether CBEAM increased speed options are to be applied. This option may be necessary for models with a large number of beams whose element IDs are large.

- 1 No increased speed options are used for CBEAM. (Default)
- 1 cbeam.prp file will be standard unformatted rather than direct access. A small table to provide the location of each entry is saved in memory.
- 2 The entire cbeam.prp file is saved in memory

**Note:** See param,marccbar,1 to change CBAR to CBEAM.

### MSPEEDCW

Default = 0, SOL 600 only.

Determines whether CWELD elements will be translated to Marc in core (for increased speed) or out of core. This parameter is needed if many CWELD elements are present in the model to avoid large translation times.

- 0 Processed out of core. (Default)
- 1 Process in core.

### MSPEEDOU

Default = 1, SOL 600 only.

Determines whether speed enhancements are activated for the t16op2 conversion.

- 0 Speed enhancements are not activated.
- 1 Speed enhancements are activated which will place certain scratch data in memory. For large models make sure enough memory is available (if PARAM,MSPEEDSE and/or PARAM,MSPEEDP4 are used, there is probably sufficient memory unless there is a large number of output “time” points. (Default)

### MSPEEDP4

Default = -1, SOL 600 only.

Determines whether PLOAD4 entries will be translated to Marc in core (for increased speed) or out of core.

- 1 PLOAD4 will be processed in core.
- 1 PLOAD4 will be processed out of core. (Default)

**Note:** The MSPEEDP4=1 option may require more memory than is available on certain computers for large models. Do not use if CWELD elements are present.

## MSPEEDPS

Default = -1, SOL 600 only.

Determines whether additional memory will be used if the Marc PSHELL option is invoked (PARAM,MRPSHELL,1) or smear option on SOL 600,ID in order to speed up processing. This will be beneficial if there are many materials and properties in the model and not very beneficial if all elements use just a few properties and materials.

- 1 No additional memory will be used. (Default)
- 1 Additional memory will be used if available.

## MSPEEDSE

Default = -1, SOL 600 only.

Determines whether speed enhancements are activated using extra memory and/or special low level I/O routines.

- 1 No speed enhancements will be activated. (Default)
- 1 Solid elements 2D elements data will be processed in core.
- 2 All elements will be processed in core.
- 3 2D and 3D elements will be processed in core, 1D elements will be processed using bioxxx (low level direct access routines used by gino).

**Note:** The MSPEEDSE=1 option may require more memory than is available on certain computers for large models. MSPEEDSE > 0 should not be used for 64-bit integer versions of Nastran or DEC Alpha computer systems.

## MSPEEDS2

Default = -1, SOL 600 only.

Determines whether additional memory will be used when PARAM,MSPEEDSE with options 1, 2, 3 is entered to obtain additional speed.

- 1 Additional memory will not be used. (Default)
- 1 Additional memory will be used and additional speed will usually be obtained.

## MSPEEDSH

Default = -1, SOL 600 only.

Determines whether additional speed enhancements will be used to process shell/plate elements (CQUAD4, CTRIA3, etc.) using incore memory.

- 1 No additional speed enhancements are used. (Default)
- 1 Additional speed in-core enhancements will be used. Sufficient memory must be available. If not, the job will abort with an appropriate message.

**Note:** This parameter should only be used if shell speed enhancements are selected with one of the PARAM,MSPEEDSE options.

## MSPLINC0

Integer, Default = 0, SOL 600. For SOL 400, MSPLINC0 =1 is always used.

This parameter controls whether to enforce C0 continuity for all spline options if any are requested by setting IDSPL=1 on any BCBODY entry.

- 0 C0 continuity is not enforced.
- 1 C0 continuity is enforced.

## MSTFBEAM

Default = 0, SOL 600 only.

Determines if all rigid elements (RBE2, RBE3, RBAR, RTRPLT) will be converted to stiff beams. This parameter should only be used if PARAM,MARMPCHK and/or PARAM,AUTOMSET options fail during a Marc execution.

- 1 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. All 6 dof's at each end of the rigid beam will be used.
- 2 All rigid elements will be converted to stiff beams or plates after they have initially been formed in the Marc file as RBE2 or RBE3 elements. Pin codes specified by the original Nastran rigid elements will be used when possible.

### Remark:

See PARAM,MARCSCCLR to specify a scale factor for the "Default" properties of these stiff beams.

## PARFREE

Integer, Default = 0

Determines whether the nodal coordinates in the GEOM1 datablock of SOL 600 OOP2 files will have the original coordinates or revised stress-free coordinates.

- 0 GEOM1 will have the original coordinates (same as defined on GRID entries)
- 1 GEOM1 will have modified coordinates to enforce stress-free conditions if applicable

**Remarks:**

1. ICCOORD on the BCTABLE with ID=0 entry must be set to 1 or 3 for at least one secondary-primary combination.
2. The parameter only addresses op2 files (t16 and t19 files automatically use PARFREE=1)
3. This capability is available starting with MD Nastran 2010. Prior version always used PARFREE=0.

**MSTTDYNE**

Integer, Default = 0, SOL 600 only

Controls whether SOL 600 may have static and dynamic load cases in the same analysis.

- 0 All subcases must either be static or dynamic
- 1 Mixed static and dynamic load cases are allowed.

**Remarks:**

1. Use of this entry requires SOL 600,106 or SOL 600,NLSTATIC (not SOL 600,NLTRAN).
2. Do not enter , DLOAD, LOADSET, LSEQ, TLOADi in the Case Control or Bulk Data.
3. Each subcase must have NLAUTO entries. Whether the subcase is static or dynamic is controlled by IFLAG on each NLAUTO entry. For static subcases IFLAG=3 and for dynamic subcases IFLAG=0.
4. All loads in each subcase build up from an initial value at the end of the previous subcase to the full value entered on the various load-type entries (such as FORCE, PLOAD4, SPCD, etc.) at the end of the subcase. Time histories for individual load cases are not allowed with this parameter. If all loads for a particular subcase use the same time history, this subcase may be divided into N-1 smaller subcases where N is the number of points on the time history curve. For this situation, the loads for each smaller subcase should be scaled by the amplitude of each point on the time history curve. This technique may also be used to simulate force vs. time for an analysis with all static subcases.
5. When MSTTDYNE=1, the following will automatically be added to the Bulk Data:

```
marcin,-1,0,dynamic,6
param,marcauov,0
```

These two entries may be added by the user in place of PARAM,MSTTDYNE,1 however both should not be done.

**MTABIGNR**

Default = 0, SOL 600 only.

Determines whether tables for VCCT analyses will be ignored or used.

- 1 VCCT tables are not used
- 0 VCCT tables are used. (Default)



**Remark:**

Use of tables for VCCT SOL 600 2D analyses with contact is not supported. If the table variation is not important, this parameter may be used to obtain an approximate solution.

**MTABLD1M**

Default = 1, SOL 600 only.

Option to modify or not modify all TABLED1 entries which do not start with the first point of (0.0, 0.0).

- 1 Do not modify any TABLED1 entries
- 1 Modify all TABLD1 entries that do not start with (0.0, 0.0). (Default)

**Remark:**

See PARAM,MTABLD1T.

**MTABLD1T**

Default = 0.01, SOL 600 only.

Specifies the second time value of all TABLED1 entries that do not start with the first point being (0.0, 0.0) if PARAM,MTABLD1M=1. Modified tables will start with the first point of (0.0, 0.0), the second point will be at the time specified by this parameter with an amplitude of the original first amplitude. The 3<sup>rd</sup> point will be at time 2\*MTABLD1T and amplitude of the original first amplitude.

**Remark:**

The proper value of MTABLD1T to enter to analyze a step input depends on the fundamental natural frequency of the model. If MTABLD1T is too small, the response will not fully build up and if it is too large, it will act like a ramp input rather than a step. The best value to use is approximately 0.1/fn, where fn is the first linear natural frequency in Hz.

**MTEMPCHK**

Integer, Default = 0, SOL 600 only

Controls how temperature-dependent properties are checked in the Marc portion of SOL 600.

- 0 Check at zero degrees only. (Marc feature 9601)
- 1 Check at all temperatures, stop checking and fatal the job when the first problem is found. (Marc feature 9600)
- 2 Check at all temperatures and report all problems, then fatal the job if any problems were found. (Marc feature 9602)

**Remarks:**

1. This parameter is available starting with MD Nastran 2010.
2. The default in Marc previously was feature 9601.

3. The default in SOL 600 is different than in Marc. The default in Marc is 9600.
4. Checking in Marc will occur from the lowest temperature of any curve to the highest of any curve. Thus, to ensure that all curves pass, define all curves from the lowest temperature of any curve to the highest temperature of any curve (even though one particular curve can never reach the lowest or highest temperature specified on another curve).
5. features,960i will be entered if any TABLEMi or TABL3Di are found.

## MTEMPDWN

Default = -1, SOL 600 only.

Option to automatically choose the FeFp multiplicative decomposition plasticity model (PARAM,MARCPLAS,5) for plasticity problems with thermal loading when the temperature decreases (see PARAM,MARCPLAS).

- |    |   |
|----|---|
| -1 | Use the standard mean normal with large strain method unless PARAM,MARCPLAS is entered. (Default)   |
| 1  | Attempt to determine whether the temperature at the start of the run is lower than the initial temperature, and if so choose the FeFp method. |

### Remarks:

1. For some problems with very soft materials, the FeFp method is required to achieve convergence for plasticity cases where the temperature decreases during the early stages of the run. Since most users would not know this, PARAM,MTEMPDWN is offered to automatically let Nastran decide to invoke this method or not.
2. The FeFp method takes substantially more computer time than the default mean normal method.
3. This parameter can be entered in RC files if it is desired to be used for a large number of runs.

## MTET4HYP

Integer, Default = 0, SOL 600 only

Controls settings for TET4 elements with hyperelasticity.

- |   |   |
|---|---|
| 0 | Use Marc element 157 which uses the Herrmann formulation.   |
| 1 | Use Marc element 134 and automatically set PARAM,MARUPDAT,1 |

### Remarks:

1. MTET4HYP=1 should normally only be set if any tet4 elements use MATHE or MATHP and if one or more of the grids in such tet4 have field 7 set to a cylindrical or spherical coordinate system.
2. If Remark 1 does not apply, MTET4HYP=0 is more accurate, although it is highly recommended that tet10 elements be used for all SOL 600 models which will be even more accurate.

## MTLD1SID

Integer, Default = -1 for SOL 600

Determines whether the SID of TLOAD1s entered in the input will be changed or not. (SOL 600 only)

-1	Do not change the SID of any TLOAD1 entry
0	Change the SID of all TLOAD1s to match the lowest SID of any TLOAD1 found in the original input.
N	Change the SID of all TLOAD1s to the value N.

**Remark:**

This parameter may only be set in the input file.

**MUALLUDS**

Integer, Default = 0, SOL 600 only

Controls how material, contact and element-related user subroutines are specified in SOL 600.

- 0 User subroutines are entered using the MATUSR entry (material), contact-related (BCBODY, BCTABLE, etc.) or element entries as well as the USRSUB6 entry.
- 1 User subroutines are entered using the MATUDS, BCONDUDS and/or ELEMUDS. They should not be specified on MATUSR. BCxxx, element entries or by using the USRSUB6 entry.

**Remark:**

For version prior to MD Nastran 2010 only option 0 was available.

**MULRFORC**

Default = -1, SOL 600 only.

Option to activate multiple RFORCE entries for different portions of The model in the same subcase.

- 1 Do not allow this capability (only one RFORCE entry for subcase can be entered). (Default)
- 1 Allow multiple RFORCE entries for each subcase

**MUMPSOLV**

Integer > 0, Default = 1, SOL 600 only.

This can also be set by the Bulk Data entry PARAMARC.

- 1 Use 1 processor.
- 2 Use 2 processor.
- N Use N processor.

**Remarks:**

1. This parameter may only be used by solver the MUMPS solver (solver type 12).

2. The PARAMARC Bulk Data entry cannot specify this type of parallel processing.

## MUSBKEEP

Default = 0, SOL 600 only.

- |   |  |
|---|--|
| 0 | If user subroutines are included, they are compiled and linked to form a new version of Marc if MUSBKEEP=0, the new version of Marc will be deleted at the end of the run. (Default) |
| 1 | The new Marc executable will be saved on disk in the same directory as the Nastran input file. Its name will be the name used in PARAM,MARCUSUB with the extension marc.             |

## MVERMOON

Integer, Default = 0, SOL 600 only

Controls whether 5-term Mooney series or 5-constant Mooney will be used in the Marc portion of SOL 600.

- |   |  |
|---|--|
| 0 | 5-constant Mooney formulation will be used.    |
| 1 | 5-term series Mooney formulation will be used. |

### Remark:

Prior to MD Nastran 2010, the Marc only allowed a 5-constant Mooney formulation with no volumetric terms. Now Marc allows a full 5-term series for both distortional and volumetric behavior for both MATHP and MATHE entries. If a version of Marc prior to this is to be used, set param,mvermoon,0 or use the default.

## MWINQUOT

Integer, Default = 0, SOL 600 only

Determines whether SOL 600 run on Windows will attempt to place quote strings in the command to execute Marc when PATH is not specified on the SOL 600 Executive Control statement.

- |   |   |
|---|---|
| 0 | Attempt to place quote strings around various fields. |
| 1 | Do not place any quote strings.                       |

### Remarks:

1. Quotes are normally only required if spaces are needed in the file names or directory names.
2. This parameter may be placed in rc files.
3. Depending on the complexity of the command to execute Marc, quote strings may sometimes be incorrect. If the command is complex, it is suggested that PATH=1 be placed on the SOL 600 Executive statement and the statement to execute Marc be placed in the marcrun.pth file.

## MWLDGSGA

Integer, Default = 1, SOL 600.

Controls CWELD type ELEMID GS, GA, GB usage in SOL 600.

- 0 When GS, GA and GB are all greater than zero for CWELD type ELEMID, Marc ignores GS and uses GA and GB to determine if a connection is possible. MWLDGSA=0 retains this default behavior, however experience shows that some models will fail with a message that the weld connection could not be achieved.
- 1 This option will re-set GA and GB to zero so that GS will be used to achieve the connection. Experience shows that this option may be more reliable for many models with CWELD type ELEMID.

## MXICODE0

Default = 5

In nonlinear harmonic response analysis, if the solution fails to converge more than MXICODE0 times in succession, a new trial displacement vector is calculated.

## MXLAGM1

Default = 0.0

MXLAGM1 is to override the maximum Lagrange Multiplier that is used by the merit function in Trust Region.

## NASPRT

Default = 0

NASPRT specifies how often data recovery is performed and printed in SOL 200. By default, SOL 200, in addition to performing an analysis and optimization, will perform full data recovery operations at the first design cycle and upon completion of the last design cycle.

If NASPRT > 0, then data recovery operations are performed at the first design cycle; at every design cycle that is a multiple of NASPRT; and the last design cycle. For example, if PARAM,NASPRT,2 and the maximum number of design iterations requested is 5, then data recovery is obtained at design iterations 1, 2, 4, and 5.

If NASPRT < 0, then no data recovery operations are performed.

## NDAMP, NDAMPM

NDAMP:

Default = 0.01 for SOLs 129 and 159, -0.05 for SOL 400 heat transfer and mechanical non-contact analysis, 0.0 for SOL 400 contact.

NDAMPM:

Default = 0.0 for SOL 400 non-contact, 1.0 for SOL 400 contact. Not used for SOL 400 heat transfer.

NDAMP/NDAMPM:

In SOLs 129 and 159, numerical damping may be specified for the METHODS = "ADAPT" on the entry through the NDAMP entry in order to achieve numerical stability. A value of zero for NDAMP requests no numerical damping. The recommended range of NDAMP values is from 0.0 to 0.1.

SOL 400 Mechanical: In SOL 400, numerical damping may be specified for all METHODS (ADAPT, FNT, etc. on the entry. NDAMP and NDAMPM are two parameters that control the damping scheme and the associated dynamic operator. The equilibrium equation for the most general operator (the Generalized- $\alpha$  operator) is given by

$$M\ddot{u}_{n+1+\alpha_m} + C\dot{u}_{n+1+\alpha_f} + F_{n+1+\alpha_f}^{int} = F_{n+1+\alpha_f}^{ext}$$

where  $\alpha_m$  is NDAMPM and  $\alpha_f$  is NDAMP. In the previous equation, a typical quantity  $F_{n+1+\alpha}$  is given by the expanded form

$$F_{n+1+\alpha} = (1 + \alpha)F_{n+1} - \alpha F_n$$

Depending on the values of NDAMP and NDAMPM, the equilibrium equations can reduce to the HHT- $\alpha$  scheme (NDAMPM = 0) or the WBZ- $\alpha$  scheme (NDAMP = 0) or the Generalized- $\alpha$  scheme (NDAMPM  $\neq$  0, NDAMP  $\neq$  0). For the HHT- $\alpha$  scheme, NDAMP can vary in the range of  $-0.33 \leq \text{NDAMP} \leq 0.0$ . For the WBZ- $\alpha$  scheme, NDAMPM can vary in the range of  $0.0 \leq \text{NDAMPM} \leq 1.0$ . For the Generalized- $\alpha$  scheme, NDAMP can vary in the range of  $-0.5 \leq \text{NDAMP} \leq 0.0$  and NDAMPM in the range of  $-0.5 \leq \text{NDAMPM} \leq 1.0$ .

For problems involving no contact, the HHT- $\alpha$  is used in SOL 400 with default NDAMP = -0.05 and NDAMPM = 0.0 except that the model is linear, or no mass and no damping matrix. In this case, the defaults are NDAMP = 0.0 and NDAMPM = 0.0. For problems involving contact, the WBZ- $\alpha$  scheme is used in SOL 400 with default NDAMPM = 1.0 and NDAMP = 0.0

SOL 400 Heat Transfer: Numerical damping may be specified through the value of NDAMP only. NDAMPM is not used. NDAMP can be varied in the range of (-2.414,0.414). At these outer limits, the transient scheme reduces to the Backward-Euler method. Any value that is outside this range is automatically reset to the closest outer limit. For NDAMP = 0.0, the transient scheme reduces to the Crank-Nicholson scheme. The default value of NDAMP is -0.05.

SOL 400 Thermo-Mechanical Coupled Analysis: If NDAMP and NDAMPM are unspecified, then the default that is appropriate for the particular sub-step is used. For e.g., for a coupled transient contact problem, NDAMP = -0.05 during the heat transfer sub-step and NDAMP = 0.0, NDAMPM = 1.0 during the mechanical sub-step. If the values of NDAMP and NDAMPM are specified by the user in the Bulk Data Section, then these values would be applicable for both the thermal and mechanical sub-steps. To make the values of the NDAMP/NDAMPM physics-dependent, the user can define the values under each sub-step of the Case Control Section. E.g.,

```
SUBSTEP 1
  ANALYSIS=HTRAN
  PARAM,NDAMP,-2.414
```

```
SUBSTEP 2
  ANALYSIS=NLTRAN
  PARAM,NDAMP,-0.05
  PARAM,NDAMPM,0.0
```

## NEWMARK

Default = NO

See CQC under [SCRSPEC](#).

## NFDOPTS

Default = 2

PARAM, NFDOPTS, 2 computes frequency dependent matrices at master frequencies. In general, master frequencies via MFREQ in case control section of input deck is less than the number of forcing frequencies. Hence, PARAM, NFDOPTS, 2 can improve performance as well as reduce disk space requirement for large frequency response jobs. With PARAM, NFDOPTS, 1, frequency dependent matrices are computed at forcing frequencies.

## NHPLUS

Default = 20

In nonlinear harmonic response analysis, in order to avoid aliasing in the calculation of the Fourier coefficients, a certain number of extra evaluation points are used. NHPLUS allows the number of extra points to be defined.

## NINTPTS

Default = 10

N1NTPPTS requests interpolation over the NINTPTS elements closest to each grid point. NINTPTS=0 requests interpolation over all elements, which can be computationally intensive. Related parameters include BIGER, CURV, NUMOUT, OG, OUTOPT, SIG, S1M, S1AG and S1AM.

## NLAYERS

Default = 5 for CQUAD4 and CTRIAR, 7 for CQUADR and CTRIAR, 3 for CAXISYM (minimum=1, maximum=12) (SOLs 106, 129, 153, 159, and 400)

NLAYERS is used in material nonlinear analysis only and is the number of layers used to integrate through the thickness of CQUAD4, CTRIA3 CQUADR, CTRIAR and CAXISYM elements with isotropic material properties. Set NLAYERS=1 for efficiency if no bending is selected (MID2=0 or -1 on all PSHELL entries). Do not specify NLAYERS=1 if MID2 is greater than zero. A larger value of NLAYERS will give greater accuracy at the cost of computing time and storage requirements. For CQUADR and CTRIAR, the maximum is 11.

## NLHTLS

Default = 0

See Remarks under Case Control command, [TSTRU \(Case\)](#).

## NLHTOL

Default = 1.0E-5

During the iteration procedure of nonlinear harmonic response, the norm of the residual load vector for the current step is divided by the norm of the residual load vector for the previously converged step; this value is

then compared with NLHTOL. If the value is smaller than NLHTOL, the system is assumed to have converged.

## NLHTWK

Default = 1.0E-5

In nonlinear harmonic response analysis, if convergence is not obtained, a line search procedure is initiated to calculate a scaling factor for the displacement vector from which updated nonlinear loads are subsequently calculated. If the solution fails to converge more than 5 times in succession a new trial displacement vector is calculated using a push off factor the size of which is defined by NLHTWK. See also parameter [MXICODEO](#).

## NLMAX

Default = 60

The number of suspected massless mechanisms is determined from the number of high ratio messages. If this number exceeds NLMAX, the number of trial massless mechanisms is reduced to NLMAX. This avoids an expensive debug run when there may be thousands of massless mechanisms due to systematic modeling error, such as having CONM2 elements on many grid points for which structural elements have been left out through oversight. The value of this parameter may be increased on initial debug runs where many high ratio DOFs may be present, and the user prefers to see them all at once, rather than on successive runs where only a part are removed at one time. See also [MECHFIX](#).

## NLMIN

Default = 10

In the case of only one or a few high ratio DOFs, more MMs may be present resulting in more trial MM vectors being used, and those trial vectors that do not indicate true problems are discarded by the code. A smaller value could be considered on a stable model undergoing small modeling changes. See also [MECHFIX](#).

## NLPACK

Default=100

NLPACK is used in the nonlinear solution sequence SOL 400 with ANALYSIS=NLTRAN for transient analysis only.

In transient analysis SOL 400 will pack output data and restart data for NLPACK output time steps together at the last time step as a single data package. For example, if NLPACK=100 (the default), then one data package have output data for 100 output time steps and restart data for the last time step. Later usage such as restart or initial condition for later step can be performed only at data package boundaries. If NLPACK=-1, all output data for a STEP and restart data for end of the STEP are grouped into a single package. In this case, the restart can be performed only at STEP boundaries. NLPACK=0 is illegal. If NLPACK=1, each package of data on the database includes output data for one output time step and restart data. In this case, restart can be performed at every output time step. Please note that the output time step is controlled by the NO field on the Bulk Data entry, or NO filed of FIXED time step control as well INTOUT field of ADAPT



time step control in NLSTEP Bulk Data entry. NLPACK also influences the intermediate output selected by NLOPRM.

Please note that NLPACK is based on the number of the output load steps (concerning INTOUT in NLSTEP, and so on), but not the number of the calculated load steps. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With “intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

## NLTOL

Default =  $\begin{cases} 2 & \text{(SOL 106 and MD Nastran SOL 400)} \\ 0 & \text{(SOL 153, nonlinear heat transfer)} \end{cases}$

NLTOL establishes defaults for the CONV, EPSU, EPSP and EPSW fields of NLPARM Bulk Data entry for nonlinear static and heat transfer analysis according to the following table:

NLTOL	Level of Accuracy
0	Very high
1	High
2	Engineering Design
3	Preliminary Design

See Remark 16. of the NLPARM entry for further details and corresponding default NLPARM field values.

## NMLOOP

Default = 0

In SOL 106 nonlinear static analysis, normal modes will be computed with the updated nonlinear stiffness if PARAM,NMLOOP is requested in the subcase section, it must be for all subcases. The nonlinear normal modes will be computed at the last iteration loop of each subcase in which a METHOD command appears.

## NMNLFRQ

Default = 0.0

PARAM, NMNLFRQ, Real\_Value allows users to select material or property values for frequency dependent materials or properties at a desired frequency other than the “nominal” values specified on the MATi or PBUSH, PCOMP, PCOMPG, PDAMP, or PELAS entries.

If there is only CBUSH frequency dependency and NO other type of material dependency, then Nastran will not use/update the CBUSH properties at the specified NMNLFRQ.

## NOCOMPS

Default = +1

NOCOMPS controls the computation and printout of composite element ply stresses, strains and failure indices. If NOCOMPS = 1, composite element ply stresses, strains and failure indices are printed. If NOCOMPS = 0, the same quantities plus element stresses and strains for the equivalent homogeneous element are printed. If NOCOMPS=-1, only element stresses and strains are printed. MAT1, MAT2, or MAT8 entries with MIDs>99999999 run the risk of conflicting with internally generated MAT2 entries when composites are present and should be avoided. Even if no composites are present, MIDs>99999999 signal to shell element stress computations that no STRESS is to be computed. In the case of MIDs>99999999 if the user desires stress calculations for shell elements, even if no composites are present, then PARAM, NOCOMPS, -1 is required.

Homogenous stresses are based upon a smeared representation of the laminate’s properties and in general will be incorrect. Element strains are correct however.

NOCOMPS only supports conventional elements, not advanced elements. For advanced nonlinear element, only ply stress/strain will be output.

## NODCMP

Default = 0

For some type of nonlinear transient problems, including heat transfer, the decomposition of the solution matrix is not required. In order to increase efficiency, NODCMP is created to determine whether the solution matrix will be decomposed. If NODCMP=0, the solution matrix will be decomposed. If NODCMP=1, the solution matrix will NOT be decomposed. In this case, MAXBIS and DJUST on the Bulk Data entry, NLPARM, must be set zero on the Bulk Data entry. NODCMP is available for SOL 129 and 159 only.

## NOELOF

Default = -1

If NOELOF > 0, then the grid point forces (GPFORCE Case Control command) are computed along the edges of the two-dimensional elements. The default value will suppress this output.

## NOELOP

Default = -1

If  $NOELOP > 0$ , then the sum of the grid point forces (GPFORCE Case Control command) are computed parallel to the edges of adjacent elements. The effect of CBAR element offsets is not considered in the calculation of the forces. The default value will suppress this output.

## NOGPF

Default = 1

NOGPF controls the printout of grid point forces (GPFORCE Case Control command).  $NOGPF > 0$  specifies that the grid point forces are to be printed.  $NOGPF < 0$  suppresses the printing of the grid point forces.

## NONCUP

Default = -1

In SOLs 111 and 112, NONCUP selects either a coupled or uncoupled solution algorithm in the modal dynamic response analysis. This parameter has the following meanings in both solution sequences:

NONCUP = -1 (Default)      Use the uncoupled solution algorithm if there are no off-diagonal terms in any of the modal matrices (MHH, BHH and KHH); otherwise use the coupled solution algorithm. However, in order to ensure correct results, the coupled solution algorithm is used when large mass enforced displacement, large mass enforced velocity or SPC/SPCD enforced motion is employed in SOL 112, regardless of the existence of off-diagonal terms in the modal matrices.

If **enforced motion** is requested in **modal transient analysis**, NONCUP = 0 is the default.

NONCUP > -1      Use the coupled solution algorithm unconditionally

NONCUP = -2      Use the uncoupled solution algorithm unconditionally

User Information Message 5222 indicates which solution algorithm is used in the analysis.

## NQSET

Default = 0

If  $NQSET > 0$ , and the PARAM entry is in Case Control, all part superelements that do not contain QSET entries, or are not referenced by SENQSET entries in the main Bulk Data Section, have NQSET generalized coordinates assigned to them. These QSET variables are functionally equivalent to those generated by SENQSET entries.

## NUMOUT, NUMOUT1, NUMOUT2, NUMOUT3

See S1, S1G, S1M.

## OELMOPT

Integer; Default=3.

PARAM,OELMOPT selects the class of elements to include in the search for grid points:

- 0 All regular elements but not PLOTEL and rigid elements.
- 1 All regular elements and PLOTEL.
- 2 All regular elements and rigid elements.
- 3 Both 1 and 2 (default).

## OELMSET

Integer; Default = 0

Identification number of a Case Control command SET definition. The members of the specified SET represent the identification numbers of the finite elements that are to be retained in the “reduced” op2 file element connection data block.

## OG

Default=0

See CURV.

## OGEOM

Default = YES

See POST < 0.

## OGRDOPT

Integer; Default = 1

Selects the method used to create the set of grid points retained in the reduced grid point geometry data block. The default simply uses the set of grid point IDs listed in the OGRDSET Case Control SET. Set consistency is checked. OGRDOPT=2 uses the list of grid point IDs that are connected to elements in the OELMSET Case Control SET. OGRDOPT=3 merges the contents of the OGRDSET Case Control SET with the contents of the grid point list connected to the elements in the OELMSET Case Control SET. There is no consistency check for OGRDOPT=2 or OGRDOPT=3. OGRDOPT=0 turns the SET consistency check off altogether. For this case, the grid points retained are those specified in the OGRDSET SET and the elements retained are those specified in the OELMSET SET. See also PARAM,OELMOPT description.

## OGRDSET

Integer; Default = 0

Identification number of a case control command SET definition. The members of the specified SET represent the identification numbers of the grid points that are to be retained in the “reduced” op2 file grid geometry data block.

## OLDSEQ

$$\text{Default} = \begin{cases} -1 & \text{for non-iterative-distributed-parallel solutions} \\ 5 & \text{for iterative solutions using distributed parallel methods} \\ & \text{(NASTRAN ITER=1 and system(231)>0)} \\ 6 & \text{if SUPER=2} \end{cases}$$

OLDSEQ selects from the following options for resequencing:

- 1 No resequencing is performed.
- 1 Use the active/passive option.
- 2 Use the band option.
- 3 For the active/passive and the band option select the option giving the lowest RMS value of the active columns for each group of grid points.
- 4 Use the wavefront (Levy) option.
- 5 Use the Gibbs-King option even if the CPU estimate is higher than for nonsequencing.
- 6 Use the automatic nested dissection option even if the CPU estimate is higher than for no resequencing. See the following SUPER=2 description.
- 8 Semiautomatic selection. The program will compute estimates for two options that are suitable for the decomposition method selected by the PARALLEL and SPARSE keywords on the NASTRAN statement and select the option with the lowest estimate. The following table shows the suitable options for each decomposition method.

Decomposition Method	Suitable Options
regular	1 and 4
parallel	2 and 5
sparse	6 and 7

- 9 The extreme partitioning method is used to partition the model into domains.
- 10 The METIS partitioning method is used to partition the model into domains.
- 11 The MSCMLV partitioning method is used to partition the model into domains.

**Notes:**

1. The model partitioning options make sense only when running with the DOMAINSOLVER command in the Executive Control Section. For DOMAINSOLVER (PARTOPT=GRID), param,oldseq,9 is the default. For all other DOMAINSOLVER options, the default is param,oldseq,11.
2. The wavefront option does not support superelement resequencing or starting nodes. Also note that the automatic nested dissection option uses starting nodes only to establish the root of the initial connectivity tree.

If the value of OLDSEQ is changed in superelement analysis, an SEALL=ALL restart is required.

PARAM,FACTOR is used to generate the sequenced identification number (SEQID) on the SEQGP entry as follows:

$$\text{SEQID}=\text{FACTOR}*\text{GRP}+\text{SEQ}$$

where:

SEQ = generated sequence number

GRP = group sequence number

If GRP=0, use GRP(MAX)+1 where GRP(MAX) is the largest group sequence number in the database.

PARAM,MPCX controls whether the grid point connectivity created by the MPC, MPCADD, and MPCAX entries and/or the rigid element entries (e.g., RBAR) is considered during resequencing:

- |    |  |
|----|--|
| -1 | Do not consider the connectivity of the MPC, MPCADD, MPCAX, or rigid element entries.  |
| 0  | Consider the connectivity of the rigid element entries only. (Default).  |
| >0 | Consider the connectivity of the rigid element entries and the MPC, MPCADD, and MPCAX entries with the set identification number set to the value of this parameter. |

PARAM,SEQOUT controls the output options as follows:

- |   |   |
|---|---|
| 0 | Do not generate any printed or punched output for the new sequence (Default). |
| 1 | Print a table of the internal/external sequence in internal order.            |
| 2 | Write the SEQGP entries to the PUNCH file.                                    |
| 3 | Perform SEQOUT=1 and 2.   |

PARAM,START specifies the number of the grid points at the beginning of the input sequence. The input sequence will be the sorted order of the grid point numbers including the effect of any SEQGP entries input by the user. A single SEQGP entry can be input to select the starting point for the new sequence. Otherwise, the first point of lowest connectivity will be used as the starting point.

If PARAM,SUPER<0, all grid points from the connection table that are not part of the group currently being processed are deleted. This option provides for sequencing only the interior points of a superelement. If any

superelements are present, the residual structure is not resequenced. If all of the grid points are in the residual structure, they are resequenced.

If PARAM,SUPER=0 or 1, all grid points in the connection table are considered. This option provides for the recognition of passive columns.

If PARAM,SUPER=2, then all points that are connected to multipoint constraints (via MPC entries) or rigid elements (e.g., the RBAR entry) are placed in a special group at the end of the sequence. This option also forces OLDSEQ=6 and may not be selected with other values of OLDSEQ. This option is intended primarily for models that have many active columns due to MPCs or rigid elements; e.g., a model with disjoint structures connected only by MPCs or rigid elements.

## OLDWELD

Default = NO

By default, the program uses new formulations for connector elements (CFAST, CSEAM and CWELD). To acquire identical connector results as those from previous versions before v2017.1, specify OLDWELD=YES.

## OMACHPR

Default = NO

See POST < 0.

## OMAXR

Default = 2 · BUFFSIZE

OMAXR specifies the maximum record length of data written by the OUTPUT2 module under PARAM,POST,<0 and PARAM,OPTEXIT,-4. BUFFSIZE is a machine-dependent value defined in the *MSC Nastran Configuration and Operations Guide*. For further information, see the OMAXR parameter description under the OUTPUT2 module description in [MSC Nastran DMAP Programmer's Guide](#).

## OMID

Default = NO

To print or punch the results in the material coordinate system, set the parameter OMID to yes. Applicable to forces, strains, and stresses for TRIA3, QUAD4, TRIAR6, and QUAD8. This is available for linear analysis only. Other elements and outputs are not supported. This capability is not supported by post-processors (xdb and op2 output are not changed), by grid point stress output that assumes output is in element coordinate system, random response analysis, heat transfer analysis, stress sorting, MAXMIN Case Control, composite elements, MONPNT3 Bulk Data and external superelement data recovery in the assembly run.

## OMSGLVL

Integer; Default = 0

Set consistency check error message severity flag. The default causes FATAL messages to be generated if the grid set is not consistent with the element-related grid point set and the job is terminated. If OMSGLVL=1, the FATAL messages are reduced to WARNINGS and the job is allowed to continue.

## OP2GM34

Default = TRUE

By default, the GEOM3 and GEOM4 are written in OP2 file. To avoid GEOM3 and GEOM4 output, set PARAM, OP2GM34, FALSE.

## OPCHSET

Integer; Default = 0

SET punch request flag. If OPCHSET=1, then the list of grid points used to reduce the grid point geometry data block will be punched in case control SET definition format.

## OPGEOM

Default = -1

OPGEOM > -1 prints the aerodynamic set definitions for all degrees-of-freedom. To print structural degree-of-freedom sets, please see [USETPRT](#).

Sequence	Print	USETPRT
None	None (Default)	-1
	Row sort only	0
Internal	Column sort only	1
	Row and Column sort	2
	Row sort only	10
External	Column sort only	11
	Row and Column sort	12

The [USETPRT](#) parameter description contains further explanation that is relevant to OPGEOM as well.

## OPGTKG

Default = -1

OPGTKG > -1 prints the matrix for the interpolation between the structural and aerodynamic degrees-of-freedom.

## OPPHIB

Default = -1

In the flutter (SOLs 145 and 200) and dynamic aeroelastic (SOLs 146 and 200) solution sequences, OPPHIB > -1 and a DISPLACEMENT request in the Case Control Section will output the real vibration modes with the structural displacement components transformed to the basic coordinate system.

## OPPHIPA

Default = -1

In the flutter (SOLs 145 and 200) and the dynamic aeroelastic (SOL 146) solution sequences, OPPHIPA > -1 and a DISPLACEMENT command in the Case Control Section will output the real



structural vibration modes at all degrees-of-freedom, including the aerodynamic degrees-of-freedom in the global coordinate system. Use PARAM,OPPHIPB to output in the basic system.

## OPTEXIT

Default = 0

In SOL 200, especially during the checkout of the analysis model and the design optimization input data (design model), it may be desirable to exit the solution sequence at certain points before proceeding with full optimization. OPTEXIT may be set to values of 1 through 7 and -4. The DSAPRT Case Control command overrides the specification of PARAM,OPTEXIT,4, -4, or 7. The description of OPTEXIT values follow.

OPTEXIT Value	Description
0	Do not exit. Proceed with optimization.
1	Exit after the initialization of the analysis and design model but before finite element analysis begins.
2	Exit after finite element analysis and initial design response and shape basis vector processing.
3	Exit after design constraint evaluation and screening.
4	Exit after design sensitivity analysis and print the matrix of design sensitivity coefficients (DSCM2). This is equivalent to the DSAPRT (UNIFORM,END=SENS) Case Control command.
-4	Exit after design sensitivity analysis and write the data blocks related to sensitivity coefficients (DSCM2 and DSCMCOL) to an external file using the OUTPUT2 and OUTPUT4 modules. This is equivalent to the DSAPRT (NOPRINT,EXPORT END=SENS) Case Control command. See related parameters ITAPE, IUNIT, and OMAXR.
5	Exit after the first approximate optimization of the design model.
6	Exit after the first update of the analysis model based on the first approximate optimization of the design model.
7	Compute and output design sensitivity coefficients at the end of normal program termination: hard convergence, soft convergence, or maximum design cycles. This is equivalent to the DSAPRT (UNIFORM,START=LAST) Case Control command. If the final design is a discrete design optimization, no sensitivity is performed and the OPTEXIT=7 request is not honored.

## OPTION

Default = ABS

See SCRSPEC.

## OSETELE

Default = 2

See [AUTOGOUT](#).

## **OSSETGRD**

Default = 1

See [AUTOGOUT](#).

## **OSWELM**

Default = -1

Offset for identification numbers of internally generated RBE3 rigid body elements (generated by all formats of CFAST and CSEAM as well as CWELD with formats ELEMID, GRIDID, ELPAT and PARTPAT). By default, OSWELM=-1, the numbering starts with SYSTEM(182) + 1. The default of system cell 182 is SYSTEM(182)=100,001,001. If the user defines OSWELM > 0, then the numbering starts with OSWELM + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD element, a pair of RWELD constraint elements is generated if MSET=ON is specified, see the entry PWELD, 3329 for an explanation.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated RBE3 rigid body elements (generated by CWELD and CFAST, all formats) and CONM2 mass elements (generated by CFAST when it has nonzero mass). The default behavior in SOL 400 is the same as in the linear solution sequences.

## **OSWPPT**

Integer, Default = -1

Offset for internally generated grid identification numbers for connector elements (GA and GB for CWELD and CFAST if these are not specified and all auxiliary grids for CWELD, CFAST and CSEAM). By default, OSWPPT=-1, the numbering starts with SYSTEM(178) + 1. The default of system cell 178 is SYSTEM(178)=101,000,000. If the user provides OSWPPT > 0, then the numbering starts with OSWPPT + 1.

Active ONLY for "PARAM,OLDWELD,YES", for each CWELD or CFAST element, a pair of grid points GA and GB is generated internally if the formats ELEMID, GRIDID, ELPAT or PARTPAT are used and if no identification numbers for GA and GB are specified, see the entry CWELD, 1696 and CFAST, 1505 for a definition of the formats.

Active ONLY for "PARAM,OLDWELD,YES", in an SOL 400 nonlinear analysis this defines the offset for identification numbers of internally generated grids of connector elements CWELD and CFAST (GA and GB if these are not specified in the bulk data input and all auxiliary grids). The default behavior in SOL 400 is the same as in the linear solution sequences.

## **OUGCORD**

See [POST](#).

## **OUGSPLIT**

Default=YES

By default, displacements, velocities, and accelerations are written to separate tables. PARAM, OUGSPLIT, NO will combine these tables into a single table and is intended for third-party post-processors which are not able to read the separate tables. The value of OUGSPLIT does not affect the results in the f06 file.

## OUNIT1

Default = Value of OUNIT2

For PARAM, POST, -1 and -2 defines the unit that geometry data blocks are output to. See PARAM, POST. This parameter should not be specified after BEGIN SUPER.

## OUNIT2

Default = 12

For PARAM, POST, -1 and -2 defines the unit that results data blocks are output to. See PARAM, POST. This parameter should not be specified after BEGIN SUPER.

## OUTOPT

Default = 0

See CURV.

## PACINFID

Integer, no default.

This parameter is used only in data recovery operations for elements in acoustic field point mesh bulk data. PACINFID is the ID of the PACINF entry that contains the location of the pole used to determine element orientation. Data recovery operations require that the element surface normal vector point away from the pole location. This parameter is required only if the acoustic model references multiple PACINF bulk data entries and the pole locations on the PACINF entries are not coincidental.

## PANELMP

Replaced by a keyword on the FLSPOUT Case Control command.

## PATPLUS

Default = NO

PARAM, PATPLUS, YES may be used with PARAM, POST, 0 to allow the user to also write data related to geometry, connectivity, and properties and results from GPFORCE, WETSENS, ELSENS, and PACCE to a Fortran unit as is done under PARAM, POST, -1.

## PATVER

Default = 3.0

See POST = -1.

## PCOMPRM

Default = 0

PCOMPRM controls the computation and printout of composite element extreme ply stresses, strains, failure Indices and strength ratios. Note that extreme ply output encompasses ply stresses, ply strains, ply failure indices and ply strength ratios together. Punch output is not available for extreme ply results.

PCOMPRM = 0 (default) , STANDARD composite ply responses are output.

PCOMPRM = 1 STANDARD & EXTREME composite ply responses are output

PCOMPRM = 2 ONLY EXTREME composite ply responses are output

## PDRMSG

Default = 1

PDRMSG controls the printout of messages associated with deformed plots, including error messages. PDRMSG = 0 suppresses the printout. Contour values will not be displayed unless the default value is used.

## PEMDMP

Default = 'NO'

Setting the value of character parameter PEMDMP to 'YES' (PARAM,PEMDMP,YES in bulk data) will invoke the distributed Nastran-PEM solution logic. PEMDMP is intended for large Nastran models with multiple PEM trim components.

The PEMDMP solution strategy is to distribute Trim Master Frequencies evenly among distributed Nastran DMP processes. The PEMDMP solution is possible when all the following are true:

- Nastran SOL 111 solution specification
- DOMAINSOLVER ACMS is used
- Command line option "dmp=N" where N>1
- Multiple hosts; or use of RUNOPT=MULTIPAR on the DOMAINSOLVER entry (Multiple-Parent DMP processing)
- Number of forcing frequencies greater than NDMP (number of DMP processes)
- Number of trim master frequencies greater than NDMP+1
- PARAM,PEMDMP,YES in bulk data

If one of the above items is not true, or if RUNOPT=PARCHILD is specified on the DOMAINSOLVER entry, the PEMDMP solution is not deployed. If the PEMDMP solution is active, User Information Messages 10544 and 10545 are printed in the F06 file. Otherwise, no messages are printed.

## PEMFRIM

PARAM,PEMFRIM,1 is used to request full Reduced Impedance Matrix (RIM) from Actran, instead of lower-triangular RIM. By default (PARAM,PEMFRIM,0), Actran returns lower-triangular RIM. Note that PEMFRIM will increase disk space demand of PEM job. In addition, OOC field of ACPEMCP will cause Actran to generate full RIM when set to an integer greater than 1.

## PEMMAPC

PEMMAPC is used to generate lists of GRD IDs of the coupling surfaces. 'PARAM,PEMMAPC,1' (default is 0) will produce following files in 'mapping\_control' subdirectory of job submit directory.

For	SET ID	Generic File name
Structure GID coupled with all TRMCs	100000	<deckname>.<job seq>.x_nas.structure_component1.set
Fluid GID coupled with all TRMCs	200000	<deckname>.<job seq>.x_nas.fluid_component2.set
Structure GID coupled with TRIMID=xxxx only	y00018xxxx for GLUED or z00018xxxx for SLIDING	<deckname>.<job seq>.>.x_nas.structure_component1.trim_coupling.<xxxx>.<y or z>.set
Fluid GID coupled with TRIMID=xxxx only	Y00028xxxx for OPEN or Z00028xxxx for IMPER	<deckname>.<job seq>.>.x_nas.fluid_component2.trim_coupling.<xxxx>.<Y or Z>.set
TRIMID=xxxx GID coupled with structure	xxxx90001	<deckname>.<job seq>.>.x_nas.interface<xxxx>.surface.1.set
TRIMID=xxxx GID coupled with fluid	xxxx90002	<deckname>.<job seq>.>.x_nas.interface<xxxx>.surface.2.set

Guidelines for PARAM,PEMMPAC,1

1. PARAM,PEMMPAC,1' must be placed in the main bulk data section or under 'BEGIN BULK' to be effective.
2. Without 'PARAM,PEMNPART,n', PEMMPAC logic has a TRIM ID limit of less than 10000.
3. With 'PARAM,PEMNPART,n', PEMMPAC logic has a TRIM ID limit of less than 100.

## PEMNCOP4

PARAM,PEMNCOP4,1 (default) is used to request each Reduced Impedance Matrix (RIM) from Actran to be written into individual file. With PARAM,PEMNCOP4,0, Actran writes all RIMs into a single file. Note that PARAM,PEMNCOP4,1 can reduce footprint on disk space of large PEM jobs.

## PEMNPART

PEMNPART can be utilized to partition a large trim component. The default value for PEMNPART is 1. 'PARAM,PEMNPART,n' will partition a large trim component into 'n' parts. The guidelines are:

1. PARAM,PEMNPART,n' must be placed under 'BEGIN TRMC=xx' to be effective for trim component 'xx'.
2. Each trim component can have its own 'PARAM,PEMNPART,n'
3. When value of 'n' for PEMNPART is greater than 1, new trim components are generated with ID equal to 'xx'\*100+<seq no> where <seq no> goes from 0 to n-1.
4. For no loss of accuracy, setting 'n' to a value equal to the number of disconnected parts of a large trim component.
5. For accepting loss of accuracy to run on a machine with limited memory, setting 'n' to a value larger than the number of disconnected parts of a large trim component.

6. 'PARAM,PEMNPART,n' with  $n > 1$  should be utilized only as an alternative for running PEM jobs with large trim component and insufficient memory on the computer. Other alternatives for running PEM job with large trim component and insufficient memory are:
  - a. Split large trim component into smaller trim components and/or
  - b. Acquire more memory for the computer
7. For a trim component with 'PARAM,PEMNPART,n' where  $n > 1$  and 'PARAM,TRMBIM,physical', data recovery will be skipped for all partitioned trim components.

## PEMSKIN

'PARAM,PEMSKIN,1' is used for Actran to couple the trim bodies only to the surface of the cavity and will not search any node inside the cavities. The default value for PEMSIN is 0 which allows Actran to target both surface and interior nodes of the cavities for trim bodies coupling.

## PENFN

Default =  $1.0e+5$  (See LMFACT)

## PERCENT

Default = 40

See CQC under [SCRSPEC](#).

## PH2OUT

Default = 0, SOL 400 only

For nonlinear solution sequence, SOL 400, in addition to the regular phase III output, the user can also request the phase II output. This is useful when the run is terminated abnormally before the phase III outputs are formatted and printed. The phase II output consists of all outputs requested by the Case Control commands in the input file and prints in sort1 format. If there's no PH2OUT, MSC Nastran outputs phase III outputs only. This is the regular output. If PH2OUT=1, MSC Nastran outputs phase II outputs only. In this case, there will be no output for the upstream superelements. If PH2OUT=3, MSC Nastran outputs both phase II and phase III outputs. In this case, some of the outputs for the residual structure may be redundant.

Case Control command, NLOPRM OUTCTRL, takes precedence over PH2OUT. In other words, when NLOPRM OUTCTRL is present, PH2OUT is no longer functional. NLOPRM OUTCTRL=SOLUTION is equivalent to PH2OUT=1 and NLOPRM OUTCTRL=(STD,SOLUTION) to PH2OUT=3.

## PKRSP

Default = -1

If PKRSP=0, the magnitude of the output quantities at the time of peak acceleration of the modal variables is output. This option is available only for modal transient analysis.

## PLTMSG

Default = 1

PARAM,PLTMSG,0 suppresses messages associated with undeformed plot requests, including error messages.

## POST

Default = 9999999

If PARAM,POST,0, then the following parameters and discussion apply (not support in SOL 400):

The data blocks often used for pre- and postprocessing will be stored in the database and also converted, by the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)), to a format suitable for processing by MSC Patran. These data blocks include input data related to geometry, connectivity, element and material properties, and static loads; they also include output data requested through the Case Control commands OLOAD, SPCF, DISP, VELO, ACCE, THERMAL, ELSTRESS, ELFORCE, FLUX, GPSTRESS, GPFORCE, ESE, GPSDCON, and ELSDCON.

The converted data is written to logical FORTRAN units, which may be assigned to physical files in the File Management Section. The FORTRAN unit numbers are specified by the parameters GEOMU, POSTU, and LOADU. By default, all data is written to the logical FORTRAN unit indicated by GEOMU. If LOADU > 0, static load data may be diverted to another unit indicated by LOADU. If POSTU > 0, then output data requested with the Case Control commands listed above will be diverted to the logical unit indicated by POSTU. See [Database Concepts in the MSC Nastran Reference Guide](#) for the procedure for assigning physical files.

By default, if converted data already exists on the files indicated by GEOMU, POSTU, and LOADU, then the DBC module will overwrite the old data. If this is not desirable, then PARAM,DBCOVWRT,NO must be entered. The parameters MODEL and SOLID may be used to store more than one model and solution in the graphics database. These parameters are not supported by MSC Patran.

PARAM,DBCDIAG > 0 requests the printing of various diagnostic messages from the DBC module (see [MSC Nastran DMAP Programmer's Guide](#)) during the data conversion. By default, no messages are printed.

If PARAM,PATPLUS,YES is specified along with PARAM,POST,0 then the PARAM,POST,-1, operation will also be performed.

If PARAM,POST,< 0 or 1, then the following parameters and discussion apply:

- PARAM,POST,1 outputs the appropriate files for the SimXpert and Patran programs and should be used to obtain results from the latest result data blocks from SOL 400 and other recent MSC Nastran developments such as NEF.
- PARAM,POST,-1 outputs the appropriate files for the Patran program (PARAM,POST,1 should be used for more recent versions of MSC Nastran > 2012).
- PARAM,POST,-2 outputs the appropriate files for the Siemens I-deas<sup>®</sup> program.
- PARAM,POST,-4 outputs the files indicated below along with OPHIG for the MSC\_NF interface by LMS International.
- PARAM,POST,-5 outputs the files indicated in the table below along with LAMA and OPHG1 for the FEMtools interface by Dynamic Design Solutions.
- POST=-4 and -5 are intended for SOL 103 only.
- PARAM,POST,-6 outputs the files indicated below for Siemens Unigraphics<sup>®</sup>.

- PARAM,POST,-7 outputs the files for Free Field Technologies.

An OUTPUT2 file for FORTRAN unit 12 in binary format is automatically created in the same directory and with the same name as the input file and with the extension “.op2”. For example, if the input file is fender.dat then the OUTPUT2 file will be called fender.op2.

An ASSIGN statement is required in the FMS Section only if neutral file format is desired as follows:

ASSIGN OP2='filename of FORTRAN file' FORM

Geometry data blocks are output with PARAM,OGEOM,YES (Default) and are written to a FORTRAN unit specified by PARAM,OUNIT1 (Default = OUNIT2) for POST = -1, -2, -4, and -6. PARAM,OUNIT2K (Default = 91) specifies the unit number for KELM and KDICT with PARAM,POST,-5. PARAM,OUNIT2M (Default = 92) specifies the unit number for MELM and MDICT with PARAM,POST,-5. Note that PARAM,POST,-5 is not supported with DMP. See the following table for the specific geometry data blocks written for different values for POST.

See also the PARAM,POSTEXT description for additional data blocks written to the .op2 file.

POST					Geometry Data Block	Description
-1,1 or -7	-2	-4	-5	-6		
YES	NO	NO	NO	NO	GEOM1S	Grid Point Definitions (Superelement)
NO	YES	YES	NO	YES	CSTM	Coordinate System Transformations
NO	YES	YES	NO	YES	GPL	Grid Point List
NO	YES	YES	NO	YES	GPDT	Grid Point Definitions
NO	YES	YES	NO	NO	EPT	Element Properties
NO	YES	YES	NO	NO	MPT	Material Properties
YES	YES	YES	NO	NO	GEOM2	Element Definitions
YES*	YES	NO	NO	NO	GEOM3	Load Definitions
YES*	YES	NO	NO	NO	GEOM4	Constraint Definitions
YES	NO	NO	NO	NO	DIT	Dynamic Table Input
YES	NO	NO	NO	NO	DYNAMICS	Dynamic Loads Definition
NO	NO	YES	YES	NO	KDICT	Element Stiffness Dictionary
NO	NO	YES	YES	NO	KELM	Element Stiffness Matrices
NO	NO	YES	YES	NO	MDICT	Element Mass Dictionary
NO	NO	YES	YES	NO	MELM	Element Mass Matrices
NO	NO	NO	NO	YES	ECTS	Element Connections
YES	NO	NO	NO	NO	VIEWTB	View Element Table
YES	NO	NO	NO	NO	EDOM	Design Model Input
YES	NO	NO	NO	NO	GEOM2S	Same as GEOM2 for superelements



POST					Geometry Data Block	Description
-1,1 or -7	-2	-4	-5	-6		
YES	NO	NO	NO	NO	CSTMS	Same as CSTM for superelements
YES	NO	NO	NO	NO	EPTS	Same as EPT for superelements
YES	NO	NO	NO	NO	MPTS	Same as MPT for superelements

\* To avoid having GEOM3 and GEOM4 written if POST=-1, set PARAM,OP2GM34,FALSE.

PARAM,OMACHPR,NO selects the single precision format for GPDT, CSTM, and GEOM1.

PARAM,OMACHPR,YES selects the machine precision format. With PARAM,POST,< 0,

PARAM,OMACHPR,NO is the default. With PARAM,POST,1, PARAM,OMACHPR,YES is the default.

For PARAM,POST = -1 and -2, results data blocks are output to a FORTRAN unit specified by

PARAM,OUNIT2 (Default = 12). This parameter is allowed to vary between superelements. In buckling solution sequence (SOL 105), a unique value of OUNIT2 should be specified for the buckling subcase. For all other solution sequences or within a given analysis type, changes to PARAM,OUNIT2 between subcases is not supported. See the POST Case Control command to change the unit number between subcases. See also the related parameter OMAXR.

By default the displacements are output in the global coordinate system. To output in the basic coordinate system, specify PARAM,OUGCORD,BASIC.

PARAM,POST,-1 or 1: Results Data Blocks for Patran and SimXpert

By default, the following data blocks are output under PARAM,POST,-1. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file. It is recommended when processing large post files in Patran and SimXpert that the following environment variable setting is used when starting the session: DRANAS\_NAST\_MEM=2048MB.

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
< 3.0	≥ 3.0				
YES	YES	OQG	SPCFORCE	OQG1	Forces of single-point constraint
YES	NO	OUG	DISP	OUGV1PAT	Displacements in the basic coordinate system
YES	YES	OUG	DISP	OUGV1	Displacements in the global coordinate system
YES	NO	OES	STRESS	OES1	Element stresses (linear elements only)
YES	NO	OEF	FORCE	OEF1	Element forces or heat flux (linear elements only)
YES	YES	OEE	STRAIN	OSTR1	Element strains
YES	YES	OGPS	GPSTRESS	OGS1	Grid point stresses
YES	YES	OESE	ESE	ONRGY1	Element strain energy
YES	YES	OGPF	GPFORCE	OGPFB1	Grid point force balance table

PARAM,PATVER		Parameter Name	Case Control	Data Block Name	Description
< 3.0	≥ 3.0				
NO	YES	OEFX	FORCE	OEF1X	Element forces with intermediate (CBAR and CBEAM) station forces and forces on nonlinear elements
NO	YES	OESX	STRESS	OES1X	Element stresses with intermediate (CBAR and CBEAM) station stresses and stresses on nonlinear elements
NO	YES	OPG	OLOAD	OPG1	Applied static loads
NO	YES	OCMP	STRESS	OES1C	Ply stresses
NO	YES	OCMP	STRAIN	OSTR1C	Ply strains
NO	YES	none	DISP SPCFORCE FORCE STRESS STRAIN	OUPV1 OQP1 DOEF1 DOES1 DOSTR1	Scaled Response Spectra
			none	LAMA	Nonlinear Buckling
NO	YES	none	DISP OLOAD	OCRUG OCRPG	
NO	YES	none	NLSTRESS	OESNLXR	Nonlinear static stresses
NO	YES	none	BOUTPUT	OESNLBR	Slideline stresses
NO	YES	none	NLLOAD	OPNL1	Nonlinear loads
NO	YES	none	STRESS	OESNLXD	Nonlinear transient stresses

PARAM,POST,-2: Results Data Blocks for Siemens I-deas®

By default, the following data blocks are output under PARAM,POST,-2. By default, the displacements are output in the basic coordinate system. To output in the global coordinate system, specify PARAM,OUGCORD,GLOBAL. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, if PARAM,OQG,NO is specified, then the SPCFORCE output is not written to the OUTPUT2 file.

PARAMeter Name	Case Control	Results Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraint
OUG	DISPLACE	BOUGV1	Displacements in the basic coordinate system
		BOPHIG	Eigenvectors in the basic coordinate system
		OUGV1	Displacements in the global coordinate system
		TOUGV1	Grid point temperatures
OES	STRESS	OES1	Element stresses (linear elements only)

PARAMeter Name	Case Control	Results Data Block Name	Description
OEF	FORCE	OEF1	Element forces (linear elements only)
	FLUX	HOEF1	Element heat flux
OEE	STRAIN	OSTR1	Element strains
OESE	ESE	ONRGY1	Element strain energy
OCMP	STRESS	OEFIT	Failure indices
	STRESS	OES1C	Ply stresses
	STRAIN	OSTR1C	Ply strains
OUMU	ESE	LAMA	Eigenvalue summary
		ONRGY2	Element strain energy
OEFX	FORCE	OEF1X	Element forces (nonlinear elements only)
OESX	STRESS	OES1X	Element stresses (nonlinear elements only)
none	none	ODELBGPD	Shape optimization geometry changes

PARAM, POST, -4: Results Data blocks for LMS International/MSC NF

By default, the following data blocks are output under PARAM,POST,-4. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OUG	DISPLAC	OPHIG	Eigenvectors in the global coordinate system.

PARAM, POST, -5: Results Data blocks for Dynamic Design Solutions/FEMtools

By default, the following data blocks are output under PARAM,POST,-5. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that eigenvectors not be written to the OUTPUT2 file. PARAM,OUNIT2O (Default51) specifies the unit number of the OUTPUT2 file.

PARAMeter name	Case Control	Data Block Name	Descriptions
OUG	DISPLAC	OUGV1	Eigenvectors in the global coordinate system.
		LAMA	Eigenvalue summary.

PARAM, POST, -6: Results Data Blocks for Siemens Unigraphics®

By default, the following data blocks are output under PARAM,POST,-6. The following parameters may be used to disable the output of data blocks to the OUTPUT2 file. For example, PARAM,OUG,NO requests that displacements not be written to the OUTPUT2 file.

PARAMeter Name	Case Control	Data Block Name	Description
OQG	SPCFORCE	OQG1	Forces of single-point constraints
OUG	DISPLAC	OUGV1	Displacements
OES	STRESS	OES1	Element stresses
OEF	STRESS	OEF1	Element forces
OEE	STRAIN	OSTR1	Element strains
OESX	STRESS	OES1X	Element stresses with intermediate station stresses and stresses on nonlinear elements
OEFX	STRESS	OEF1X	Element forces with intermediate station forces and forces on nonlinear elements
OPG	OLOAD	OPG1	Applied static loads
none	none	DVPTAB	Designed property table
none	none	OPTPRMG	Optimization parameters
none	none	PROPO	Final element properties
none	none	DBCLOPT	Optimization summary data

#### PARAM.POST,-7 Results Data Blocks for Free Field Technologies

Except eigenvectors, all other output data blocks are the same as PARAM,POST,-1. For eigenvectors, the matrix form is written to the OUTPUT2 file.

### POSTEXT

Default = NO

Under PARAM,POST,-1 and -2, and if PARAM,POSTEXT,YES is specified, then the following data blocks are also written to the .op2 file specified by PARAM,OUNIT2.

Data Block Name	PARAM POST	Description
FRL	-1, -2	Frequency response list (modal frequency response only).
BHH	-1, -2	Modal damping matrix (modal frequency response only).
KHH	-1, -2	Modal stiffness matrix (modal frequency response only).
BGPDT	-1	Basic grid point definition table.
PVT0	-1, -2	User parameter value table
CASECC	-1	Case Control table
EQEXIN(S)	-1	Equivalence external to internal grid ID table

Data Block Name	PARAM POST	Description
CLAMA	-1, -2	Complex eigenvalue table
OEDE1	-1, -2	Element energy loss output table
OEKE1	-1, -2	Element kinetic energy output table
OUGV2	-1, -2	Displacement output table in SORT2
PSDF	-1, -2	Power spectral density table
OGPWG	-1, -2	Grid point weight generator output table
TOL	-1, -2	Time output list
OPHSA	-1, -2	Solution set eigenvectors (modal frequency response only)
LAMA	-1	Eigenvalue summary table
ONRGY2	-1	Element kinetic energy (obsolete)
PSDFH	-1, -2	Power spectral density table for modal coordinates
DSCM2	-1, -2	Design sensitivity coefficient matrix
DSCMCOL	-1, -2	Design sensitivity parameters table
MPOUT	-1,-2	Monitor point outputs

## POSTADF

Default = NO

This parameter is used to export the results of frequency analysis responses and participation factors by PFMODE, PFPANEL, and PFGRID in ADF format file. This exportation is requested by setting this parameter to YES. The unit information of the exported functions should be defined by DTI, UNITS statement for this function (see ADAMSMNF statement). If not defined, SI unit will be assumed with the warning message. The ADF file is defined as input data name with the extension “afu” automatically. Alternatively the name can be defined with ASSIGN ADFFILE= statement in FMS section. For the participation factor output, FILTER=0. option should be applied in PFMODE and PFPANEL.

## POSTU

Default = -1

See POST=0.

## PREFDB

Default = 1.0

See ACOUT.

## PRGPST

Default = NO

PRGPST is replaced by the PRINT keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,PRGPST.

## PRINT

Default = YES

PARAM,PRINT,NO suppresses the automatic printing of the flutter summary in flutter analysis.

## PROUT

Default = -1

PARAM,PROUT,-1 suppresses execution and printout from the ELTPRT module. PARAM,PROUT,-1 prints a list of all elements sorted on EID and summary tables giving the range of element identification numbers for each element type.

ELTPRT will provide an information message when it detects any duplicate element ID. For the case of CONV and CHYBDi elements, duplicate element IDs are by design. The information message provided by ELTPRT can be safely ignored for these elements.

## PRPA

Default = 1.0E37

## PRPJ

PRPA and PRPJ control the printout of intermediate load matrices for diagnostic purposes during superelement assembly. If the value of PRPA (or PRPJ) is positive, all terms larger in magnitude than the value are printed. If the value of PRPA (or PRPJ) is negative, all terms smaller in magnitude than the value are printed. The default value requests no printout. PARAM,IRES,1 must be present for these parameters to be effective. The PA matrix contains the internal loads transmitted to the downstream superelement. The PJ matrix contains external loads applied on the superelement; that is, it has the same content as the data produced by the Case Control command OLOAD. All of this data may be obtained on restart using the SELR Case Control command option. A related parameter is IRES.

## PRPHIVZ

Default = 1.0E37

PRPHIVZ controls the printout of the PHIVZ matrix that contains the component mode eigenvectors of the model. It includes all degrees-of-freedom with motion except the m-set, eliminated for multipoint constraints. The FAPPROX matrix contains the square root of the diagonal terms of the generalized stiffness matrix divided by  $2\pi$ . For fixed-boundary solutions, it is a good approximation for the natural frequencies of the component. For free- or mixed-boundary solutions, it is of a lower value than the natural frequencies. All terms larger than PRPHIVZ in both matrices will be printed (i.e., PARAM,PRPHIVZ,0.0 causes all terms to be printed).

## PRTMAXIM

Default = NO

PRTMAXIM controls the printout of the maximums of applied loads, single-point forces of constraint, multipoint forces of constraint, and displacements. The printouts are titled “MAXIMUM APPLIED LOADS”, “MAXIMUM SPCFORCES”, “MAXIMUM MPCFORCES”, and “MAXIMUM DISPLACEMENTS”.

## PRTRESLT

Default = YES

PRTRESLT controls the printout of the resultants of applied loads, single-point forces of constraint, and multipoint forces of constraint. The printouts are titled “OLOAD RESULTANTS”, “SPCFORCE RESULTANTS”, and “MPCFORCE RESULTANTS”.

## PSENPCH

Default = NO, for SOL 200 job with PART SEs only.

Setting PSENPCH to YES causes updated Bulk Data entries of a PART SE for a design cycle punched to a separate file named as follows

JOBNAME\_psexx\_yy.pch – where xx is the PART SE ID and yy is the design cycle.

## PTHRES

Default = -1

PARAM, PTHRES, TABLED1\_ID requests the PEAKOUT result at which the desired response exceeds a prescribed threshold value. The threshold value for the response is defined on the TABLED1 Bulk Data entry as frequency vs. the threshold of a response value.

## Q

Default = 0.0

Q specifies the dynamic pressure. Q must be specified in aeroelastic response analysis (SOLs 146), and the default value will cause a User Fatal Message.

## RADMOD

Default = YES

This parameter only applies to heat transfer solution sequences for SOLs 153, 159 and 400. The parameter, RADMOD, only affects radiation heat transfer problems. The radiation matrix is modified to avoid the temperature overshoot with a coarse mesh. If the user wants the program to skip this operation (modify the radiation matrix) then insert PARAM,RADMOD,NO.

## RBSETPRT

Default=0

This parameter controls the printout of rigid element IDs that are included in the analysis. Setting PARAM,RBSETPRT to a value not equal to zero activates the printing of rigid element IDs. Note that this function works in conjunction with the rigid element selection via MPC=n in case control and SET3,n,RBxx in bulk data.

## RESLTOPT

Default = 8

RESLTOPT's default value provides component-level force summary output for model checkout (PARAM, CHECKOUT, YES), loads generation (OLOAD output), and SPC and MPC forces. Setting RESLTOPT to a value of 1 produces abbreviated output formats only.

## RESVEC

This parameter and the related parameters RESVINER, RESVSO, RESVSE, and RESVSLI are obsolete or replaced by options on the RESVEC Case Control command.

## RFORSET

Default = 0

A PARAM, RFORSET, n where n is a Bulk Data SET1 entry, allows the user to apply rotational force loading (RFORCE) to just a portion of the structure. RFORSET like RFORCE loading is global and is computed once at the beginning of the analysis. The Bulk Data SET1 n defines the GRIDS to which the rotational force loading is to be applied. RFORSET can appear in either the **Bulk Data** or **ABOVE** or **IN the 1st Subcase**. A RFORSET appearing in any other Subcase will be ignored and may cause incorrect results. A RFORSET appearing in or above the first subcase takes precedence over a RFORSET appearing in Bulk Data. See also the companion GRAVSET entry. If there is no RFORCE loading called out, then PARAM, RFORSET, n needs to be removed from the run.

## RHOCP

Default = 1.0

This parameter represents a scale factor used in the computation of ERP in units of dB. See the ERPREFDB parameter. This parameter may alternatively be set using the ERP Case Control command.

## RIMINTP

RIMINTP can be used to select the interpolation method for Reduced Impedance Matrix(RIM) for PEM and ACTRIM. The input options for RIMINTP are LINEAR and LOG10. The default is LINEAR.

## RKSCHEME

Default = Remark 1., SOL 700 only

Defines the type of time-integration scheme used in the Riemann solution-based Euler solvers.

### Format:

PARAM,RKSCHEME,NUMBER

### Example:

PARAM,RKSCHEME,3

NUMBER      Number of Runge-Kutta stages. (Integer > 0)



**Remarks:**

1. This parameter can be used in combination with the original Roe solver and the improved full 2<sup>nd</sup> order fluid- and gas Euler solver. The default number of stages depends on the spatial accuracy of the solution scheme. One stage is used for first order spatial accuracy, and three stages for second order spatial accuracy.
2. It is recommended that the user leave the setting to the default values as defined depending on the selected spatial accuracy of the solution.
3. It has been found that in some cases with fluid flows, it may be necessary to manually limit the time step to a fixed and lower value than the scheme determines. These occasions are noticeable when you view a contour plot of the pressure. The user may find that the pressure field looks like a checkerboard. This is then caused by a numerical instability due to a time step that is too large for the fluid flow. The face fluxes are always correct but the element values get decoupled due to the time integration instability. Lowering the time step to about half the maximum value the scheme determines solves this problem. Please note that this significantly slows down the computation! Since this problem rarely occurs we have chosen not to automatically limit the time step to a lower value for performance reasons. However, the user needs to be aware that the analysis may exhibit the aforementioned behavior.

**RMSINT**

Default = LINEAR for the trapezoidal approximation.

RMSINT specifies the interpolation method for numerical integration when computing both RMS (Root Mean Square) and N0 (Number of Zero Crossings or Mean Frequency) from PSDF (Power Spectral Density Function). RMSINT = LINEAR requests the trapezoidal approximation, which is the existing Nastran approach. RMSINT = LOGLOG requests the Log-Log interpolation.

**ROHYDRO**

Default = Remark 3., SOL 700 only

Defines the minimum density for hydrodynamic, single-material Eulerian elements.

**Format:**

PARAM,ROHYDRO,VALUE

**Example:**

PARAM,ROHYDRO,1.E-6

VALUE            Density cutoff. (Real > 0.0)

**Remarks:**

1. Hydrodynamic, single-material Eulerian elements with a density less than ROHYDRO are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.

2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than ROHYDRO, then no transport is done.
  - b. If the density of element A after transport is less than ROHYDRO, then all of the mass is transported to element B.
3. By default, the cutoff density for hydrodynamic Eulerian elements is set to 1.E-5 times the material reference density. For the Riemann solution-based solvers, the default is set to 1.E-6 times the reference density.
4. Please note that this parameter has a different effect in the Riemann-solution based Euler solvers. The elements are never viewed as empty, but a small amount of mass (equal to the element's volume times the cut-off density value) remains in the element. All other state variables (velocity, energy and pressure are reset to zero for these types of elements). For fluid flows where you wish to model cavitation, use Tait's equation of state with a critical density at which the pressure remains constant and the fluid cavitates.

## ROMULTI

Default = Remark 3., SOL 700 only

Defines the minimum density for multimaterial Eulerian elements.

### Format:

PARAM,ROMULTI,VALUE

### Example:

PARAM,ROMULTI,1.E-6

VALUE            Density cutoff. (Real > 0.0)

### Remarks:

1. Multimaterial Eulerian elements with a density less than ROMULTI are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of a specific material in element B after transport is less than ROMULTI, no transport is done.
  - b. If the density of a specific material in element A after transport is less than ROMULTI, all of the mass of that material is transported to element B.
3. By default, the cut-off density is set for each material separately as 1.E-5 times the material reference density.

## ROSTR

Default = Remark 3., SOL 700 only

Defines the minimum density for single-material Eulerian elements with shear strength.

**Format:**

PARAM,ROSTR,VALUE

**Example:**

PARAM,ROSTR,1.E-6

VALUE            Density cutoff. (Real &gt; 0.0)

**Remarks:**

1. Single-material Eulerian elements with shear strength with a density less than ROSTR are considered to be empty. All of the variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than ROSTR, then no transport is done.
  - b. If the density of element A after transport is less than ROSTR, then all of the mass is transported to element B.
3. By default the cut-off density for Eulerian elements with shear strength is set to 1.E-5 times the material reference density.

**RSPECTRA**

Default = -1

RSPECTRA = 0 requests that response spectra be calculated for transient analysis. See [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide* for a discussion of this capability. Response spectra will be calculated for any superelements or the residual structure for which other output requests are present in the same run. The requirements for the other output requests are also in [Response Spectrum Analysis](#) (Ch. 9) in *Dynamic Analysis User's Guide*. Any punch data produced is sent to the standard Nastran PUNCH file. Related parameters are TABID and RSPRINT.

**RSPRINT**

Default = 0

RSPRINT controls the printout of tabulated values of response spectra. RSPRINT = -1 suppresses the printout. The related parameter is RSPECTRA.

**RSTTEMP**

Default = NO

In SOL 106, PARAM,RSTTEMP,YES will cause the automatic restart to ignore changes to or additions of TEMPij Bulk Data entries. This is applicable to SOL 106 restart runs in which the temperature changes are only intended to affect the loading and not the material properties. The restart run also requires the use of the DBLOCATE FMS statement instead of the RESTART statement to reference the data base; e.g.,

```
assign run1=plate-run1.MASTER
dbloc logical=run1
```

**RVLDFILT**

Default = NO

This parameter controls a filtering process for residual vector loads. Set `PARAM,RVLDFILT,YES` to check the linear independence of load vectors to be used as a basis for residual vector augmentation. This option is useful if you have many thousands of applied loads, many of which occupy the same vector space (for example, same direction). For this case, linearly dependent load vectors will be discarded and not used for residual vector computation, leading to a more efficient solution. The linear dependence check itself may be very time consuming, hence the default value is NO. If the number of load vectors is more than about 20,000, the linear dependence check may require long running time. When the number of load vectors is large, User Information Messages 10590, 10591, and 10592 may be printed in the F06 file. See also related parameter [MAXAPL](#).

## S1, S1G, S1M

Default = -1

The MAXMIN Case Control command offers more features with much greater efficiency. `PARAM,S1i,+1` requests the sorting and/or filtering of the element stresses selected on the DTI,INDTA entry. Stresses in the element coordinate systems (S1), at grid points (S1G), and/or in material coordinate systems (S1M) based on the parameters BIGER, NUMOUT, SRTOPT, and SRTELTYP may be requested. The S1G and S1M options also require the presence of `PARAM,CURV,1`.

Parameter	Quantity	Coordinate System	Location	Elements
$S1 \geq 0$	Stresses	Element	Element centers	CQUAD4, CQUAD8, CTRIA3, CTRIA6
$S1M \geq 0$	Stresses	Material	Element centers	CQUAD4, CTRIA3
$S1G \geq 0$	Stresses	Material	Grid points to which elements connect	CQUAD4, CTRIA3

NUMOUT, in conjunction with BIGER, controls the amount of stress output.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require `PARAM,S1i,+1`. Similarly, NUMOUT3 and BIGER3 are used for composite strength ratio.

1. NUMOUT = +N requests that N element stresses be printed (or punched) for each element type.
2. NUMOUT=0 outputs all elements in a group when one or more exceeds BIGER. Some of the elements will have stresses small than BIGER. This is conceptually the same as describing an element set in case control, and limiting output in this manner. Stress files obtained with element group filtering may be used for xy plotting and other postprocessor options with DMAP alters. By contrast, the stress file when NUMOUT = -2 is more discontinuous, and may not be used for xy plotting.

3. NUMOUT = 0 does not sort but filters according to BIGER by element group. In static analysis an element group is defined as all case control selected elements for a given load case for SORT1 output. For SORT2 output an element group is defined as the data for a given element type for all load cases. In transient analysis an element group is defined as all case control selected elements at a given time for SORT1 output. For SORT2 output an element group is defined as the data for a given element at all time steps. The element group option applies only to output types described above for PARAM,S1. This option is not available with output types selected by PARAMs S1G and S1M.
4. NUMOUT = -1 requests that stresses be sorted and only those stresses with an absolute value that is greater than BIGER will be output.
5. NUMOUT = -2 (the Default) does not sort but filters according to BIGER. Related parameters include BIGER, NOELOF, NOELOP, and NOGPF.
6. NUMOUT2 = -3 outputs the maximum value of Failure Index for each element.

BIGER controls the elements for which stresses will be printed. Elements with stresses that are smaller in absolute value than BIGER will not be output. The quantity tested is element type dependent. Related parameters include CURV, NUMOUT, S1, S1G, and S1M. SRTOPT controls the scanning option to be performed.

SRTOPT Value	Description
0	Filter/sort on maximum magnitude.
1	Filter/sort on minimum magnitude.
2	Filter/sort on maximum algebraic value.
3	Filter/sort on minimum algebraic value.

SRTELTYP controls the element type to be processed, as described in the following table.

SRTELTYP Value	Description
0	All element types will be processed.
> 0	Only element type SRTELTYP will be processed.

NUMOUT1 and BIGER1 serve the same function as NUMOUT and BIGER except that they apply only to composite element stresses and do not require PARAM,S1i,+1.

NUMOUT2 and BIGER2 serve the same function as NUMOUT and BIGER except that they apply only to composite element failure indices and do not require PARAM,S1i,+1.

### S1AG,S1AM

Default = -1

See CURV.

## SBSPFORM

No Default

This parameter can be used to specify the Subspace Iteration method formulation for the structure-fluid coupled modes computation.

Subspace iteration method selects the formulation automatically based on the ratio of structure and fluid size and the existence of rigid body or constant pressure modes. But using this parameter, user can force one or other of the methods.

MF:     Mobility-Flexibility formulation

SM:     Stiffness-Mass formulation

## SCRSPEC

Default = -1 (SOLs 103 and 115 only)

SCRSPEC=0 requests that structural response be calculated for response spectra input in normal modes analysis. See [Response Spectrum Analysis](#) (Ch. 9) in the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of this capability. The scaled response calculations are made for elements and grid points of the residual structure only. There exist two basic methods which are controlled by PARAM,CQC. The default (CQC=0) selects the traditional method. CQC > 0 selects a more recent method called the complete quadratic method of peak response combination—also called the CQC method. In both methods, the responses are summed with the ABS, SRSS, NRL, or NRLO convention, depending on the value of PARAM,OPTION. If the SRSS, NRL, or NRLO options are used, close natural frequencies will be summed by the ABS convention, where close natural frequencies meet the inequality  $f_{i+1} < \text{CLOSE} \cdot f_i$ . Both PARAM,OPTION and PARAM,CLOSE may be set in any subcase, allowing summation by several conventions in a single run.

In Version 70, the NRL option has been modified slightly to correspond to the NAVSEA-0908-LP-000-3010 specification. NRLO provides the V69 NRL.

PARAM,CQC,1 or 2 selects the complete quadratic method of peak response combination; also called the CQC method. PARAM,CQC,1 selects CQC method of response combination for sum across modes. PARAM,CQC,2 is same PARAM,CQC,1, but outputs the CQC coefficients for each mode ij pair and frequency and damping for each mode. The default is to consider all modes in the calculation but if only a subset of modes are of interest then a DTI,CQC,1,... Bulk Data entry may be used to specify a list of modes (by mode number) to retain for the CQC solution. When multiple excitation directions are specified, PARAM,CQC,1 or 2 specifies the summation across the modes and PARAM,OPTION specifies the summation across excitation directions. As with PARAM,OPTION and PARAM,CLOSE, when PARAM,CQC is used at the subcase level, each subcase can have a different value specified for OPTION, CLOSE and CQC. This allows comparisons to be made in the same run among summation across directions using ABS and SRSS methods while using the CQC method to sum across modes.

For the CQC method, PARAM,DIROUT,YES (Default = NO) outputs the responses combined across the modes for each separate excitation direction as well as the responses combined across modes and directions. This allows results to be assessed per excitation direction and for the total combined response.

PARAM,POST,-1 will write the additional directional responses to the op2 file for post processing with Patran. In this case, the directional responses will be labeled in the Patran Results menu with a TIME value corresponding to the excitation sequence number. For example, if the SUPORT entry specifies the following degrees of freedom to be excited: SUPORT,1,123 then direction 1 will be labeled with TIME value 1, direction 2 with TIME value 2, etc. PARAM,POST,-1 will also write the additional maximum response to the op2 file for post processing. In this case, the maximum response will be labeled in the Patran Results menu with a TIME value 4.

For the CQC method, PARAM,NEWMARK,YES (Default = NO) may be used in the case where 3 simultaneous excitation directions are defined and calculate the Newmark combinations across the excitations directions using the 40% rule. PARAM,PERCENT (Default = 40) may be used to specify a different percentage for Newmark combinations.

## SDCSV

Default = 0

Dictates what stability derivative information is to be stored on a CSV (comma separated values) file in a SOL 144 (static aeroelasticity) task. The unit the CSV file is stored to is specified by PARAM, SDUNIT, n. SDCSV has the following options which can be summed to select multiple options:

0	No output
1	Rigid aero
2	Rigid Splined
4	Elastic restrained
8	Elastic unrestrained
16	Inertial restrained
32	Inertial unrestrained

## SDUNIT,n

n SDUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of stability derivative results in SOL 144.

## SEKD

Default = RSONLY

Controls the calculation of the upstream superelement differential stiffness matrix in SOL 106.

RSONLY	Calculate the differential stiffness for the residual structure only.
ALL	Calculate and reduce the differential stiffness for all superelements and the residual structure.

## SEMAP, SEMAPOPT, SEMAPPRT

SEMAP Default = SEMAP  
 SEMAPOPT Default = 42  
 SEMAPPRT Default = 3

The superelement map (SEMAP table) contains several lists useful for determining how the program has partitioned superelement models. It is printed automatically each time this table is generated. It consists of three major parts:

- GPM        The Grid Point Map contains a list of each grid point, its interior superelement placement, and the SEID of all grid points connected directly to it by elements. Three tables follow that summarize the connectivity between superelements, sorted on grid point sequence, SEID, and the number of connections.
- ISM        The Individual Superelement Map lists the interior grid points, exterior grid and scalar points, elements, and time and storage space estimates for each superelement.
- SDT        The Superelement Definition Table contains the SEID of every superelement in the model, the processing order, and a pictorial representation of the superelement tree.

SEMAP, SEMAPOPT, and SEMAPPRT are used to control the amount of output that is printed and other special features. The possible values for SEMAP are shown in the following table.

SEMAP Value	Output and Application
SEMAP (Default)	ISM, SDT. The lengthy GPM is suppressed. This is the appropriate value for use after the model is stable and only minor changes are to be made.
SEMAPALL	GPM, ISM, SDT. All tables are printed. This value is useful on the initial debug run of a model and when making extensive modeling changes.
SEMAPCON	Only the summary tables of the GPM and the estimation data is output. This is a useful value when iterating to an economic partitioning scheme for large, complex models.
SEMAPEST	Only the estimation data is printed. This is useful when evaluating several alternative partitioning schemes.
SEMAPPUN	No output is printed. The exterior grid points of the superelement with a SEID that is input on SEMAPOPT are placed on a CSUPER entry image on the PUNCH file, allowing the superelement to be used as an external superelement. If SEMAPOPT > 0, the superelement entry is given an SSID of SEMAPOPT. If SEMAPOPT < 0, the exterior points listed are those of the residual structure, but the CSUPER entry is given an SSID of  SEMAPOPT .

Other special features are available with parameters SEMAPOPT and SEMAPPRT. They are fully described under parameters OPT1 and OPT2 in the description of the TABPRT module in the [MSC Nastran DMAP Programmer's Guide](#).



If the default value of SEMAP is used, the other two parameters may be used to further refine this output, as described in [MSC Nastran DMAP Programmer's Guide](#) under the TABPRT module description. The printing of the SEMAP table can be avoided by the use of PARAM,SEMAPPRT,-1.

PARAM,SEMAPPRT,99 requests the printout of the SLIST (superelement processing list) and the DRLIST (superelement data recovery list).

## SENSU00

Default = NO

By default, in dynamic sensitivity analysis in SOL 200, displacements at the o-set due to pseudo-loads do not include any effect due to inertia and damping. If PARAM,SENSU00,YES is specified then these effects will be computed in a quasi-static manner and included in the sensitivity analysis.

## SEP1XOVR

Default = 0

The old and new location of moved shell grid points are printed if SEP1XOVR = 16. When the RSSCON shell-to-solid element connector is used. By default, the moved shell grid points are not printed, SEP1XOVR = 0. See the description of PARAM,TOLRSC for more details.

## SEQOUT

Default = 0

See OLDSEQ.

## SERST

Default = AUTO

By default, all restarts are considered automatic (see [Restart Procedures in the MSC Nastran Reference Guide](#)). If none of the following Case Control commands are entered, then SEALL=ALL is the default action: SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL.

These commands may be used to partition the analysis into several runs. By default, the restart will proceed in automatic fashion for each command, regenerating only that data that is affected by modifications in the Bulk Data and Case Control or changes in upstream superelements. If the user wishes to overwrite the data, even if it is not affected by modifications to the data, then PARAM,SERST,MANUAL must be entered.

With PARAM,SERST,AUTO or MANUAL, all superelements will be processed through Phase 0 (see [Superelement Analysis in the MSC Nastran Reference Guide](#)). This phase includes execution of the sequencer module (SEQP), initial superelement processing (SEP1), and initial geometry processing (GP1 and GP2) modules, which can result in significant CPU overhead. If this overhead is not desired, then PARAM,SERST,SEMI will limit Phase 0 and Phase 1 to only those superelements specified on the SEMG, SELG, SEKR, SELR, SELA, SEMA, SEMR, and SEALL Case Control commands. If none of these commands is entered, then execution will skip Phase 0 and 1.

In the modal solution sequences (SOLs 110, 111, 112, 145, 146, and 200), the modes of the residual structure are automatically computed in Phase 2 if any SE-type command (e.g., SEMG=n) is requested for the residual structure. If PARAM,SERST,SEMI and no SE-type command is specified for the residual structure, then, by default, its modes will not be recomputed. This logic is intended for restarts from SOL

103 into one of the modal solutions. If, however, the modes have not already been computed or need to be recomputed, then PARAM,SERST,RSMDS must be specified to force the calculation of the residual structure modes.

If PARAM,SERST,SEDR is specified, then Phases 0, 1, and 2 will be skipped. This option is intended for data recovery (Phase 3) runs only.

The options of SEMI, RSMDS, and SEDR are intended for models that are defined on more than one database; i.e., superelements are defined on separate databases (multiple MASTER DBsets) and processed in separate runs. Also, with this technique, databases are attached with the DBLOCATE File Management statement rather than the RESTART File Management statement. In general, these options are not recommended because they disable the automatic restart capability, which compromises the database integrity.

## SESDAMP

Default = NO

PARAM,SESDAMP,YES

Modal damping is calculated for superelements if PARAM,SESDAMP,YES is specified. An SDAMPING Case Control command that selects a TABDMP1 Bulk Data entry must also be specified in the desired superelement's subcase. By default, modal damping is added to viscous damping (B). If you insert PARAM,KDAMP,-1 (or PARAM,KDAMPFL,-1 for fluid superelements) then modal damping will be added to structural damping (K4). The SESDAMP parameter needs to be specified as NO for an assembly run.

PARAM,SESDAMP,AUG

By its nature, classical modal damping in superelements is computed only for the component modes. If fixed boundary or mixed boundary (some fixed, some free) methods are used to compute the component modes, no damping is computed for the constraint modes, which can lead to answer differences when comparing the same model with and without superelements. An enhanced modal damping method has been developed to account for damping on the constraint modes. To use the enhanced damping method instead of the classical method, define "PARAM,SESDAMP,AUG". When this enhanced damping method is defined, the user may choose how the damping factor for the constraint modes is determined via the parameter TDAMP.

## SESEF

Default = -1 (SOLs 103 and 115 only)

If SESEF = 0 in superelement normal modes analysis, the fraction of total strain energy for a superelement in each of the system's modes is output in the vector SESEFA for tip superelements and in SESEFG for nontip superelements. If SESEF = 1, strain energy fractions are output, and expansion of the eigenvectors from a-set size to g-set is branched over for tip superelements. If SESEF = -1 (the default value), no strain energy fractions are computed.

Output requests must be present in order for strain energy fractions to be calculated. If SESEF = 1, no other output results for tip superelements.

## SHIFT1

Default = -1.234

The negative shift used when computing massless mechanism constraint modes with PARAM,MMMETH,OLD. For very stiff model (1000. hz for the first flexible mode), consider using a larger value. See also [MECHFIX](#) and [MMMETH](#).

## SHLDAMP

Default = SAME

If SAME, then structural damping is obtained from MID1 material of PSHELL. If DIFF or any value not equal to SAME, each MIDi field of the PSHELL will have its own structural damping. See Remark 5. of the PSHELL.

## SIGMA

Default = 0.0

The radiant heat flux is proportional to

$$\text{SIGMA} \cdot (T_{grid} + T_{ABS})^4$$

where SIGMA is the Stefan-Boltzmann constant,  $T_{grid}$  is the temperature at a grid point, and  $T_{ABS}$  is the scale factor for absolute temperature and may be specified by PARAM,TABS. These parameters must be given in units consistent with the rest of the data in the model. The value for SIGMA in SI units is

$$5.67 \times 10^{-8} \text{ watts/m}^2\text{K}^4$$

The default value causes radiant heat effects to be discarded.

## SKINOUT

Default = NONE

Request that sets of grid and element lists be output for both the fluid and structure at the fluid-structure interface.

NONE	Requests no output of sets.
PUNCH	Requests set output to .pch only.
PRINT	Requests set output to .f06 only.
ALL	Requests set output to both .pch and .f06.

See the Case Control command FLSPOUT as an alternative selection.

## SKPAMP

Default = 0

For SOLs 145, 146, and 200, SKPAMP = -1 suppresses all unsteady aerodynamic calculations. The automatic restart performs a similar function without this parameter. Specifying it ensures suppression of the calculations, regardless of the determination of the automatic restart.

## SLOOPID

Default = 0 (SOL 129 and 159 only)

In a nonlinear transient analysis (SOLs 129 and 159) restart, SLOOPID identifies the initial conditioning previous nonlinear analysis run (SOLs 106 and 153 respectively). Setting SLOOPID greater than 0 will cause SOLs 129 and 159 to start from the static deformed position.

## SMALLQ

Default = 0.0

By default Nastran removes unused superelement q-set degrees-of-freedom from the residual structure solution set. Set this parameter to a small value (e.g., 1.0E-10) if you do not want unused superelement q-set degrees-of-freedom removed.

## SNORM

Default = 20.0

SNORM > 0.0 requests the generation of unique grid point normals for adjacent shell elements (see [Figure 6-2](#)). Unique grid point normals are generated for the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements. The grid point normal is the average of the local normals from all adjacent shell elements including CQUAD8 and CTRIA6 elements. If grid point normals are present, they are used in all element calculations of the CQUAD4, CTRIA3, CQUADR, and CTRIAR elements.

SNORM	Tolerance in Degrees
> 0.0	Unique grid point normals are generated if each angle between the grid point normal and each local normal of the adjacent shell elements is smaller than SNORM. SNORM Bulk Data entries overwrite a generated normal.
= 0.0	The generation of grid point normals is turned off. The user can define normals with the SNORM Bulk Data entry.
< 0.0	Grid point normals are not generated. SNORM Bulk Data entries are ignored.

**Caution:** If the grid shown in [Figure 6-2](#) is located on a symmetric half model boundary and, hence, Shell 2 is not present, you may attain the same result as a full model by specifying the normal direction with the SNORM Bulk Data entry.

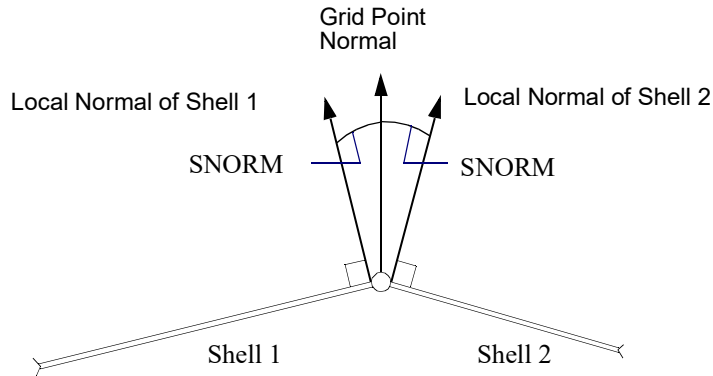


Figure 6-2 Unique Grid Point Normal for Adjacent Shell Elements

## SNORMPRT

Default = -1

PARAM,SNORMPRT,>0 writes the grid point normals of the model in the basic coordinate system to the .f06 and/or .pch files.

SNORMPRT	Switch to Print Out Normals
$\leq 0$	No output
1	Print out to the punch file (.pch)
2	Print out to the print file (.f06)
3	Print out to the punch (.pch) and print file (.f06)

## SOFTEXIT

Default = NO

In SOL 200, if soft convergence is achieved during optimization, before completing the maximum number of design iterations, the user may request an exit with PARAM,SOFTEXIT,YES.

## SOLADJC

Default = 0

PARAM SOLADJC indicates if adjoit solution vectors are to be calculated during the analysis:

- 1 Do not calculate adjoint solution vectors during the analysis. Any required adjoint solution vectors will be computed during sensitivity analysis.
- ≥0 For ANALYSIS=DFREQ, the adjoint vectors will be computed during the solution if:
1. All frequency response DRESP1 entries are grid responses.
  2. Each subcase has the same set of excitation frequencies.
  3. The number of degrees-of-freedom referenced on DRESP1 entries < (number of independent design variables + number of type-2 properties + number of spawned nonlinear beam library properties) x (number of subcases.)
  4. PARAM AUTOADJ=YES (Default)
- 999999 For ANALYSIS=MFREQ, the adjoint vectors will be computed during the solution if the four items above are satisfied and SOLADJC=999999. SOLADJC=999999 cannot be used in conjunction with FREQ3, FREQ4 or FREQ5 entries to specify excitation frequencies.

## SOLID

Default = 0

SOLID also allows several models to be stored in the same graphics database created by PARAM,POST,0.

## SPARSEDM

Default = YES

See SPARSEDR.

## SPARSEDR

Default = YES

SPARSEDR=YES limits the data recovery matrix calculations to recover only those grid points specified on SET commands referenced by grid point output requests (DISP, SPCF, etc.) or those points connected to elements specified on element output requests (STRESS, FORCE, etc.). In SOL 200, the design model and design responses are also taken into consideration for determining which grid points are needed for data recovery.

SPARSEDM=YES is for SOL 200 and takes further advantage of small design models/responses during the adjoint or pseudo-load sensitivity calculations.

These methods take advantage of very small output requests, and/or small design models in SOL 200, for large models resulting in significant CPU and disk space savings.

If, however, the output requests and/or the size of the design model in SOL 200 require the calculation of the solution over a large enough percentage of degrees-of-freedom, then it is more efficient to compute the solution at all grid points. The user PARAMeter SPDRRAT (Default75) and SPDMRAT (Default60) specifies this percentage.

The sparse data recovery method is not supported in Aeroelastic (SOL 146), Flutter (SOLs 145 and 200), Complex Eigenvalue (SOLs 107, 110, 145, and 200), Nonlinear (SOLs 106, 129, 153, 159, and 400) and Cyclic Symmetry Analysis (SOLs 114, 115, and 118).

The sparse data recovery method is deactivated when the following Case Control commands are specified: EKE, ESE, EDE, and CMSENRGY.

PARAM,DDRMM is ignored under PARAM,SPARSEDR,YES. To restore the data recovery solution process to pre-V2004 methods insert PARAM,SPARSEDR,NO.

### **SPARSEPH**

Default = YES

The default selects the very efficient sparse data recovery method during eigenvector data recovery in DOMAINSOLVER ACMS (PARTOPT=DOF). This method uses the same process for eigenvector data recovery as is requested by PARAM,SPARSEDR,YES in other dynamic solution sequences like SOLs 107 through 112.

SPARSEPH will be automatically changed to NO if ACPOWER or INTENSITY calculations are requested. These calculations require all eigenvector DOF to be computed.

PARAM,SPARSEPH,NO forces full data recovery on all degrees-of-freedom and can be significantly less efficient if data recovery is required at only a few degrees-of-freedom. See related user parameters [MDOTM](#) and [SPARSEDR](#).

### **SPDRRAT**

Default = 75

See SPARSEDR.

### **SPCGEN**

Default = NO

SPCGEN is analogous to and described by the PUNCH keyword on the AUTOSPC Case Control command. The specification of the AUTOSPC command overrides the specification of PARAM,SPCGEN.

### **SPDMRAT**

Default = 60

See SPARSEDR.

### **SQSETID**

Default = 99000001

See [AUTOQSET](#).

### **SRCOMPS**

Default = NO

SRCOMPS controls the computation and printout of ply strength ratios. If SRCOMPS=YES, ply strength ratios are output for composite elements that have failure indices requested.

## SRTLTYP

Default = 0

See S1, S1G, S1M.

## SRTOPT

Default = 0

See S1, S1G, S1M.

## START

Default = 0

See OLDSEQ.

## STEPFCT

Real, Default = 0.666, SOL 700 only. The scale factor has an effect on the time step computation of Lagrangian and Euler elements.

## STEPFCTL

Real > 0, Default = 0.9, SOL 700 only.

Defines a scale factor to be used on the internally calculated time step. When both of STEPFCT and STEPFCTL are defined, STEPFCTL will be used for Lagrangian elements and STEPFCT will be used for Euler elements.

## STIME

Default = 0.0 (SOLs 109, 112, 129 and 159 only)

In restarts from previous transient analysis runs, the user provides  $STIME = t_N$  where  $t_N$  is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from  $t_N$  as though the original run contained the new subcase data.

In SOLs 109 and 112 restarts from previous SOLs 109 and 112 runs,  $STIME$  is used to specify the proper starting time of the restart run. If  $STIME$  exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to  $STIME$ . In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any time earlier than that. The program informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run.

In SOLs 109 and 112 restarts, the user must ensure that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run. The user may, however, specify different  $TSTEP$  and  $DLOAD$  requests in Case Control and also different  $TSTEP$  and dynamic loading entries in the Bulk Data compared to the previous run. The loading and the results output from the restart run will start from the new starting time.



**Notes:**

1. The responsibility for ensuring that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run is left to the user; the program does not check for this condition. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.
2. Restarts in SOLs 109 and 112 using  $STIME > 0.0$  are supported **only** for applied loads, **not** for SPC/SPCD enforced motion. If this usage is violated, the program terminates the execution with an appropriate fatal message.

**STRUCTMP**

Replaced by a keyword on the FLSPOUT Case Control command.

**SUBCASID**

Default = 0

PARAM,SUBCASID,n where n is greater than zero, specifies that the restart proceeds from SUBCASE n in nonlinear static analysis, SOL 106. SUBCASID is an alternative to SUBID and is recommended over SUBID which indicates the subcase sequence number.

**SUBID**

Default = 1

In SOL 106 by default, the restart proceeds from the last LOOPID in the last subcase. SUBID may be used to specify an earlier subcase by specifying the sequential number (for SEID = 0) of the subcase. In SOLs 106 and 153, PARAM,LOOPID may also be specified for an earlier LOOPID. SUBCASID is an alternative to SUBID and is recommended over SUBID. See [Restarts for Nonlinear Static Analysis](#) in Appendix C: Nonlinear Analysis in *MSC Nastran Reference Guide* for further discussion.

**SUPAERO**

Default = ZONA

If SUPAERO=ZONA, then the ZONA51 code is used for supersonic aerodynamic calculations. If SUPAERO=CPM, then the CPM method is used. If ZONA51 is not available at a particular installation, PARAM,SUPAERO,CPM must be specified to avoid a fatal error when performing supersonic aerodynamic analyses. Only one supersonic aerodynamics method can be selected in a given run.

**Note:**

CPM is only supported for planar configurations.

**SUPDOF**

Default = 0

Digits 1 through 6 indicate which of the six rigid body degrees of freedom are supported for a Solution 144 run. For example, SUPDOF=35 indicates that the plunge (3) and pitch (5) degrees of freedom are supported. This parameter is optional. If omitted, Nastran will compute these data using a numerical method.

## SUPER

$$\text{Default} = \begin{cases} 0 & \text{(nonsuperelement sequences)} \\ -1 & \text{(superelement sequences)} \end{cases}$$

See OLDSEQ.

## TABID

Default = 2

TABID controls the punch output for response spectra. See [Appendix C](#) in *MSC Nastran Reference Guide*. A related parameter is RSPECTRA.

## TABS

Default = 0.0

TABS is used to convert units of the temperature input (°F or °C) to the absolute temperature (°R or °K). Specify:

PARAM,TABS,273.16      When Celsius is used.

PARAM,TABS,459.69      When Fahrenheit is used.

Refer to the Bulk Data entry, [CREEP](#) for a creep analysis with SOLs 106 or 153. Refer to PARAM,SIGMA for heat transfer analysis.

## TDAMP

PARAM,TDAMP,Real

When using enhanced modal damping for superelements (see PARAM,SESDAMP,AUG ), the user may choose how the damping factor for the constraint modes is determined.

If TDAMP=0.0 (the default), the damping factor is averaged over all the damping coefficients defined on TABDMP1.

If TDAMP is less than 0.0, the damping factor from TABDMP1 associated with the lowest natural frequency is used.

If TDAMP is positive, the damping factor defined by TDAMP is used directly.

## TESTNEG

$$\text{Default} = \begin{cases} -2 & \text{for Newtons method} \\ -1 & \text{for Arc-length method and SOLs 101-112,115,118,187, and} \\ & 200 \end{cases}$$

In nonlinear static analysis (SOLs 106 and 153), and differential stiffness generation with preload (SOLs 101-112, 115, 118, 187, and 200 with STATSUB) this parameter specifies the action to take when negative terms are encountered on the factor diagonal of matrix decomposition. Negative terms indicate that the differential

stiffness has introduced a structural instability. The instability may be real (structural buckling) or mathematical (the current iteration appears unstable, but a stable solution exists).

TESTNEG	Results
-1	Stop if negative terms occur.
1 or 0	Continue if negative terms occur.
-2	If negative terms exist, do not use differential stiffness.
2	Do not use differential stiffness.
3	Use differential stiffness with preload even if negative terms exist (SOLs 101-112, 115, 118, 187, and 200 with STATSUB only)

### TFSYMFAC

Default = 1.D-08

TFSYMFAC specifies the tolerance at which matrices generated via the TFL Case Control command are treated as anti-symmetric in the solution process. The default will not symmetrize the matrix even if the tolerance is not exceeded. Whereas, if TFSYMFAC is negative then the absolute values is used as the tolerance and the matrix will be symmetrized if the tolerance is exceeded. If TFSYMFAC is 0.D0 then the symmetry is not checked.

### TINY

Default = 1.E-3

Any elements with strain energy that is less than a TINY percentage of the total strain energy for any superelement will not be printed or made available for postprocessing by MSC Patran or other programs. TINY may also be used to suppress the printing of small numbers in the constraint check matrix  $[E_{mh}]$  described in [Geometry Processing in SubDMAP PHASE0](#) in the *MSC Nastran Reference Guide*.

### TOLRSC

Default = 0.05

When the RSSCON shell-to-solid element connector is used, the connecting grid points of the shell element are moved on to the solid face if the grid points are close enough. The tolerable distance of the shell grid point to the solid edge or face is  $\varepsilon \cdot h$  where  $h$  is the height of the solid edge; see the sample figure below. The relative tolerance is user modifiable using the parameter.

PARAM,TOLRSC, $\varepsilon$

The default for the relative tolerance is  $\varepsilon = 0.05$ . Rigid body invariance is satisfied with double-precision accuracy if the shell grid points are adjusted.

## TOPOCONV

Default = 80

Parameter TOPOCONV is applicable only to ESLNRO topology optimization tasks. It sets a lower bound for the percentage of the design variables whose maximum relative changes are within the tolerance specified by CONVDV on the DOPTPRM entry.

By default, when more than 80% of the design variables show their maximum relative changes are within CONVDV, the job will be terminated.

## TRMBIM

Default = PHYSICAL

TRMBIM is used to define the formulation used for reduced impedance matrix, RIM, for all trim components requested under TRIMGRP case control commands. It is default to 'PHYSICAL', which will generate RIM in physical coordinates. The other valid option for TRMBIM is 'MODAL', which will produce RIM in modal coordinates.

It is important to note that data recovery for trim components are supported only under 'PARAM, TRMBIM, PHYSICAL' and 'PARAM, PEMNPART, 1 (default)' for all trim components. In addition, SOL 200 supports for trim components is limited to following conditions:

1. Design model on structure and/or fluid model only (not the trim/PEM material).
2. Design responses/constraints on responses of structure and/or fluid only (not the trim/PEM material).

## TSTATIC

Default = -1 (SOLs 129, 159, 400 and 600 only)

If TSTATIC = 1, a static solution may be obtained while ignoring inertial and damping forces. For SOLs 129 and 159, this option is available only with the adaptive time-stepping method (see METHOD = "ADAPT" on the Bulk Data entry, TSTEPNL). For SOL 600, TSTATIC=1 may only be used for the default auto step iteration procedure.

## UGASC

SOL 700 only,

Real. Defines a value for the universal gas constant.

1. Must be used if the molar weight is used on a GAS option of the AIRBAG entry, or if molar gas fractions are given on an INFLFRAC entry.
2. Specify only one universal gas constant per problem.
3. In SI units, R equals  $8.3145 \text{ J mol}^{-1} \text{ K}^{-1}$ .

Using the tonne, mm, system of units R has a value of  $8314.5 \text{ tonne mm}^2 \text{ s}^{-2} \text{ mol}^{-1} \text{ K}^{-1}$ .

In imperial units, R equals  $1.9859 \text{ Btu lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$  or  $1545.3 \text{ ft lbf lbmol}^{-1} \text{ }^{\circ}\text{R}^{-1}$

## UNSYMF

Default = NO

In SOL 106, nonlinear statics, PARAM,UNSYME,YES is required to include damping effects in the calculation of complex eigenvalues. See PARAM,NMLOOP.

## UNSYMKB

Character, default = 'NO'

In modal solutions or real eigenvalue solutions, if the stiffness or damping matrix is not symmetric, the run will normally terminate with a FATAL error message. Setting UNSYMKB to 'YES' will allow the run to continue.

## UPDTBSH

Default = NO

UPDTBSH controls the update of boundary shapes generated by auxiliary boundary model analysis in SOL 200. By default, the auxiliary boundary models and shapes are generated only once at the initial design cycle and will not be updated in subsequent cycles even if the shape of the primary model is changing.

PARAM,UPDTBSH,YES requests that the auxiliary models and shapes are updated and reanalyzed at every cycle.

## USETPRT

Default = -1

USETPRT controls the tabular printout of the structural degree-of-freedom sets. See [Degree-of-Freedom Sets](#). To print aerodynamic degree-of-freedom sets, please see [OPGEOM](#).

Sequence	Print	USETPRT
None	None (Default)	-1
	Row sort only	0
Internal	Column sort only	1
	Row and Column sort	2
	Row sort only	10
External	Column sort only	11
	Row and Column sort	12

The degrees-of-freedom can be listed in ascending order according to their internal or external sequence number, but not both. The external sequence number is the grid, scalar, or extra point identification number. The internal sequence number is the number assigned after resequencing (see PARAM,OLDSEQ).

For a given sequence there are two types of tables that may be printed: row sort and column sort. For row sort, a table is printed for each set selected by USETSEL. Here is an example of row sort (USETPRT = 0 or 10):

USET	DEFINITION	TABLE	(INTERNAL SEQUENCE, ROW SORT)								
	-1-	-2-	-3-	-4-	A	-5-	-6-	-7-	-8-	-9-	-10-
	DISPLACEMENT SET										
1=	2-1	2-2									

For column sort, a single table is printed for the following sets: SB, SG, L, A, F, N, G, R, O, S, M, E. Here is an example of column sort (USETPRT=1 or 11):

USET	DEFINITION	TABLE	(INTERNAL SEQUENCE, COLUMN SORT)													
	EXT GP.	DOF	INT DOF	INT GP.	SB	SG	L	A	F	N	G	R	O	S	M	E
---																
	1	- 1	1-	1 G	1					1	1			1		
	- 2		2-		2					2	2			2		
	- 3		3-			1				3	3			3		
	- 4		4-			2				4	4			4		
	- 5		5-			3				5	5			5		
	- 6		6-			4				6	6			6		

### USETSEL

Default = 0

USETSEL specifies the sets which will be printed in the row sort (USETPRT = 0 or 10). In order to select specific sets to be printed, you must sum their corresponding decimal equivalent numbers. For example, sets A, L, and R are selected with USETSEL=128+256+8=392. To print the aerodynamic degree-of-freedom sets: J, JS, K, KS, and SA, please see [OPGEOM](#).

USETSEL	Sets Printed
0	Sets SB, SG, L, A, F, N, G, R, O, S, M, and E
-1	All sets as defined in <a href="#">Degree-of-Freedom Sets</a> .
-2	Mutually exclusive sets only; i.e., sets MP, MR, SB, SG, SZ, O, Q, R, C, B, LM, and E.

### USETSTR1

Input-character-Default ' '

USETSTR1 through USETSTR4 specifies the sets that will be printed by the specification of parameters USETPRT and USETSEL. Any set in [Degree-of-Freedom Sets](#) may be specified. A ":" is used as a separator. In the following example, the m-set (degrees-of-freedom eliminated by multipoint constraints) and s-set (degrees-of-freedom eliminated by single point constraints) are specified.

#### Example:

PARAM,USETSTR1,M:S

## VARPHI

Default =  $\pi/4$  (0.78539816)

Feature angle for detection of singular geometric features (sharp edges or corners) in the mesh during adaptive mesh refinement.

When adaptive meshing is requested (see Case Control command, [HADAPT \(Case\)](#) and Bulk Data entry, [HADAPTL](#)), singular geometric features such as sharp edges or sharp corners must be detected. To this end the face outward normals  $N_1, N_2$  of each pair of adjacent mesh faces and the edge oriented tangents  $T_1, T_2$  of each pair of adjacent mesh edges are computed (see [Figure 6-3](#)). If the angle between  $N_1$  and  $N_2$  for mesh faces, or between  $T_1$  and  $T_2$  for mesh edges is bigger than the feature angle  $\varphi$ , then the common edge or vertex will be considered a splitting edge or vertex where surfaces or lines are broken and a singular geometric feature is defined.

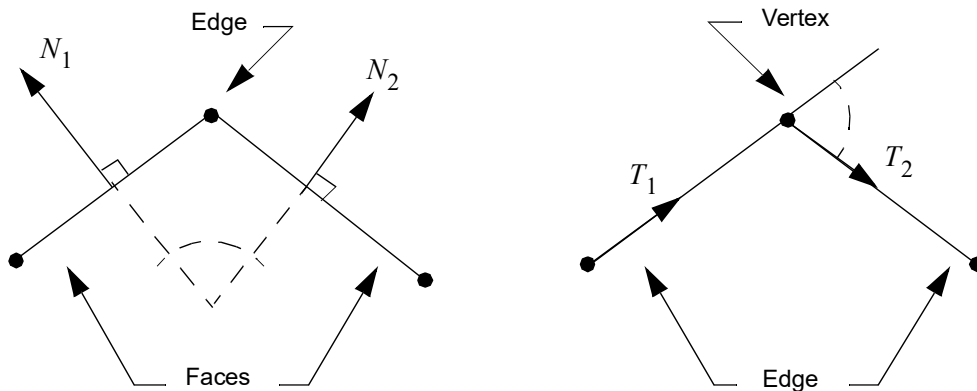


Figure 6-3

Mesh faces and elements are preprocessed to ensure consistent orientation and that the appropriate sign of face normals and edge tangents will be accounted for during the computation of their mutual angle.

## VELCUT

Default = 1.E-6, SOL 700 only

Defines the minimum velocity in Eulerian meshes.

### Format:

PARAM,VELCUT,VALUE

### Example:

PARAM,VELCUT,1.0E-6

VALUE            Minimum velocity. (Real > 0.0)

**Remark:**

Any velocity less than VELCUT is set to zero. It is mainly used to eliminate harmless but annoying small values of velocity caused by round-off error and numerical dispersion.

**VMOPT**

Default = 0

By default (VMOPT=0), virtual mass is included after the component modes are computed. Virtual mass is not taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=1, then the virtual mass will be included in the mass matrix at the same time as all other mass elements. In other words, the component modes will reflect the virtual mass. Virtual mass is taken into consideration when using GRAV, RFORCE, or ACCEL loading.

If VMOPT=2, the modes of the structure or component without the fluid are computed first (“dry” modes). The fluid effects are added to the modal basis during the residual flexibility computation to produce the “wet” modes for the component. Both eigenvalue tables are printed, allowing comparison of the dry and wet modes. The wet modes are used in modal dynamic analysis. Cost savings result from the dense Virtual Mass (VM) matrix being kept out when computing dry modes in the physical basis. Its presence can increase memory and computation times by an order of magnitude. The VM is added only to the smaller generalized basis used in Residual Flexibility Computations. The approximations introduced by this approach are generally small due to the homogeneous nature of the fluid. It is the preferred method when the number of wetted elements exceeds several hundred, for reasons of efficiency. It is to be noted that MAX normalization for eigenvectors with PARAM,VMOPT,2 may produce incorrect results. Therefore, MAX normalization is not recommended when using PARAM,VMOPT,2.

If VMOPT is not equal to 0, 1 or 2, then no virtual mass is computed.

**VREF**

Default = 1.0

In modal flutter analysis, the velocities are divided by VREF to convert units or to compute flutter indices.

**WHIRLOPT**

Default = FWD

Control forward whirl or backward whirl analysis for SYNC option in SOL 107/110 and SOL 108/111 for analysis in rotating reference frame using the ROTOR entry.

Usage:

PARAM, WHIRLOPT, FWD: For forward whirl analysis

PARAM, WHIRLOPT, BWD: For backward analysis

**WMODAL**

Default = NO

In Modal Transient analysis (SOL 112), PARAM, WMODAL, YES will automatically produce accurate conversion at all frequencies using the modal frequencies based on the eigenvalues as shown below. This



relieves the user of trying to determine the “best” value for (PARAM, W3 and W4). Note, use of WMODAL does not suppress user specified values of W3 and W4.

$$\begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix} [\Phi_{dq}^T] (G[K_{dd}^1] + [K_{dd}^4]) [\Phi_{dq}] \begin{bmatrix} 1/\sqrt{\omega_1} & 0 & \dots & 0 \\ 0 & 1/\sqrt{\omega_2} & \dots & 0 \\ \vdots & \vdots & \ddots & \vdots \\ 0 & 0 & \dots & 1/\sqrt{\omega_q} \end{bmatrix}$$

Note the CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery do not currently take into account the WMODAL conversion in SOL112.

### WRBEAMB

Integer, Default = 0

- 0 Write equivalent radius for all beams (see PARAM, BEAMB) whether beam-beam contact is anticipated or not. The equivalent radius is the 7th field of Marc’s GEOMETRY values for beam type elements.
- 1 Do not write equivalent radius (7th field is blank). This might be necessary for versions of Marc earlier than 2003.

### WTMAS600

Integer, Default = 1

Determines whether GRAV entries will be scaled by WTMAS or not for SOL 600 only.

- 0 The value of A (4th field) on the GRAV entry will not be scaled by WTMAS.
- 1 The value of A (4th field) on the GRAV entry will be scaled by WTMAS.

### Remarks:

1. This parameter is available starting with the MD Nastran 2010.
2. Prior to MD Nastran 2010, WTMAS600=1 for all analyses.

### WTMASS

Default = 1.0

The terms of the structural mass matrix are multiplied by the value of WTMASS when they are generated. In coupled fluid-structure analysis WTMASS is applied to the structural portion of the model only. WTMASS applies to MFLUID entries but it is not recommended for use in hydroelastic problems.

### W3, W4, W3FL, W4FL

Default = 0.0

The damping matrix for transient analysis is assembled from the equation:

$$[B_{dd}] = [B_{dd}^1] + [B_{dd}^2] + \frac{G}{W3}[K_{dd}^1] + \frac{1}{W4}[K_{dd}^4]$$

In coupled fluid-structure analysis, W3 and W4 are applied to structural portion of the model and W3FL and W4FL to the fluid portion of the model. The default values of 0.0 for W3, W4, W3FL, and W4FL cause the  $[K_{dd}^1]$  and  $[K_{dd}^4]$  items to be ignored in the damping matrix, regardless of the presence of the PARAM, G or GFL or  $[K_{dd}^4]$ .  $[K_{dd}^1]$  is the stiffness.

$[K_{dd}^4]$  is the structural damping and is created when GE is specified on the MATi entries.

$[K_{dd}^1]$  is the stiffness. The units of W3, W4, W3FL, and W4FL are radians per unit time. (See [Real Eigenvalue Analysis in SubDMAPs SEMR3 and MODERS](#) in the *MSC Nastran Reference Guide* for further discussion.)

In SOLs 129 and 159, W4 may vary between subcases. *However, the linear portion of the model uses only the W4 value from the first subcase and the values in the subsequent subcases are applied to the nonlinear portion of the model.*

Parameters G, W3, W4 are case control selectable by either the PARAM entries in Case Control, or RSDAMP or SEDAMP appearing in Case Control pointing to a bulk data DAMPING entry. If both DAMPING type entries and PARAM entries occur in a subcase, the DAMPING entries take precedence.

When there is no Parameters G, W3, W4 or DMAPING reference in the first Case Control, the Bulk Data specification (if any) will be used. Subsequent Case Control specification will override the Bulk Data specification and Case Control rules apply to subsequent specifications.

The CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL109 and SOL112.

Linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements when computing the element forces for data recovery take into account W3 and W4 in SOL400 with ANALYSIS=NLTRAN. These elements behave in SOL400 with ANALYSIS=NLTRAN in a linear fashion when PARAM, LGDISP, -1 (Default) and there are no PBUSHT referring to the CBUSH or PELAST entries referring to the CELAS1, CELAS2, CELAS3.

For linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements in SOL400 with ANALYSIS=NLTRAN, SOL400 considers Case Control specification of G, W3, or W4 as a data recovery boundary condition change and each STEPi for the linear elements will pick up the appropriate G, W3, and W4 values.

Any of these elements in SOL400 with ANALYSIS=NLTRAN that are nonlinear only use the standard force output data recovery  $F=Ku(t)$  since nonlinear elements compute only the unbalanced stiffness force.

In SOL129 or SOL159 linear CBUSH, CELAS1, CELAS2, CELAS3, and CFAST elements (determined as in SOL400 above) should never have G, W3, or W4 selected in the Case Control as the linear force recovery will be incorrect as only the G, W3, and W4 of the first subcase is used.

In SOL129, the FUSE option of the CBUSH should never be used if GE or B is specified for the fusing CBUSH. This is because, unlike SOL400, SOL129 does not update the B or K4 matrix after the CBUSH fuses. Thus remnant CBUSH forces due to damping will be present.

### WR3, WR4, WRH

Default = 0.0, no rotor damping or circulation terms.

Specifies “average” excitation frequency for calculation of rotor damping and circulation terms. See “Equations Used in Analyses” in the *MSC Nastran 2004 Release Guide* for equations.

### XFACT

Default=0.5 for both STRUCT and FLUID.

Additional factor to determine upper bound for Q-set reorthogonalization subspace, for ACMS VERSION=NEW (module ACMS1). Values should range between 0.1 and 1.0. Note that PARAM,XFACT only affects XFACT for STRUCT. For the FLUID model, see DOMAINSOLVER.

### XFACTX

Default=1.0 for both STRUCT and FLUID.

Multiplier applied to user’s maximum frequency to determine maximum frequency for reorthogonalization for ACMS VERSION=NEW (module ACMS1). Values should range between 0.1 and 1.0. Note that PARAM,XFACTX only affects XFACT for STRUCT. For the FLUID model, see DOMAINSOLVER.

### XFLAG

Default = 0

By default (XFLAG = 0), when temperature loads and element deformations are present, the element strain energy for the linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^T K_e u - u^T P_{et}$$

where  $u$  is the deformation,  $K_e$  is the element stiffness and  $P_{et}$  is the element load vector for temperature differences and element deformations. If XFLAG is set to 2, the element strain energy for linear elements is calculated using the following equation:

$$E = \frac{1}{2}u^T K_e u - \frac{1}{2}u^T P_{et}$$

The latter formula is the same strain energy calculation used for nonlinear elements.

When requesting Case Control commands ESE or EDE and executing SOL 108 or SOL 111, if Nastran detects frequency dependent elements (CBUSH if referred to by PBUSH/PBUSHT; CELAS1, CELAS3 if referred to by PELAS/PELAST; CDAMP1, CDAMP3 if referred to by PDAMP/PDAMPPT), Nastran will automatically remove the entire class of CBUSH, or CELAS1, or CELAS3, or CDAMP1, or CDAMP3 elements, whether or not all of a class are frequency dependent, from the residual (SE0) energy calculations because frequency dependent elements are not fully implemented in the ESE/EDE calculations and may result in wrong answers. USER INFORMATION MESSAGE 5245 will be issued for each class removed. If the user wishes to override this removal, set XFLAG to 32. If the user wants XFLAG=2 and XFLAG=33, then set XFLAG=34.

## XYMPCH

Default = NO

The XYPLOT/XPUNCH commands do not support SOL103 type mode shapes. XYMPCH when set to a value PARAM, XYMPCH, YES allows the XPUNCH command XPUNCH, DISP, RESPONSE / ... to punch without selective selection SOL103 mode shapes as SOL101 type displacements. For example:

```
$
METHOD = 10
SET 100 = 6, 7, 13
PARAM, XYMPCH, YES
DISP=100
MODES=20 $

...
OUTPUT (XYPLOT)
XPUNCH DISP RESPONSE / 6 (T3)    $
XPUNCH DISP RESPONSE / 7 (T3)    $
XPUNCH DISP RESPONSE / 13 (T2)   $
```

A more modern solution is to include in a standard SOL103 run the Nastran entry

```
NASTRAN HDF5=0
```

With for example DISP(PLOT)=ALL

And extract the resulting mode shapes from the resulting HDF5 Data Base.

Then the PARAM, XYMPCH, and MODES=n is not needed nor is any of the OUTPUT(XYPLOT) commands.

## XYUNIT, n

- n XYUNIT is used in conjunction with an FMS ASSIGN statement to specify the unit number for the storage of design optimization results and design sensitivity data in comma separated value format for use in a spreadsheet in SOL 200 or the storage of trim results in SOL 144.

## ZROCMAS

Default = NO

When performing component modal synthesis with free or mixed boundary conditions, the c-set mass is normally included during the calculation of the component modes. If the component has large masses on the c-set degrees-of-freedom, or if the user requests too many modes for the component, the c-set residual

flexibility will become singular. This causes a failure of the component reduction. The singularity may be avoided by setting ZROCMAS to YES, which will exclude the c-set mass when calculating the component modes.

## ZROVEC

Default=1.E-06

ZROVEC specifies the tolerance at which a residual vector is not linearly independent and subsequently removed from the residual vector computation process.

## Parameter Applicability Tables

[Table 6-1](#), [Table 6-2](#), [Table 6-3](#), [Table 6-4](#), and [Table 6-5](#) list parameter applicability to the solution sequences (SOLs 101 through 114).

[Table 6-6](#), [Table 6-7](#), [Table 6-8](#), [Table 6-9](#), [Table 6-10](#), [Table 6-11](#), [Table 6-12](#), [Table 6-13](#), [Table 6-14](#), [Table 6-15](#), and [Table 6-16](#) list parameter applicability to the solution sequences (SOLs 115 through 700).

- B Must be specified in the Bulk Data Section only.
- E May be specified in either the Bulk Data and/or Case Control Section.
- C Must be specified in the Case Control Section only.

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
ACEXTMTD									B		
ACEXTSET									B		
ACOUT	B				E	E	E	E	E	E	
ACOWEAK									B		
ACSYM						B			B		
ADJMETH											
ADMPOST									E	E	
ADPCON				E							
ADSTAT	B						B			B	
AERODOF											
AESDISC											
AESMAXIT											
AESMETH											
AESRNDM											
AESTOL											
ALPHA1					B	B	B	B	B	B	
ALPHA2					B	B	B	B	B	B	
ALPHA1FL											
ALPHA2FL											
ALTRED	B		B								B
ARBMASP											
ARBMFEM											
ARBMPS											
ARBMSS											
ARBMSTYP											
ARF											
ARS											
ASCOUNP	B	B	B	B	B	B	B	B	B	B	B
ASING	E	E	E	E	E	E	E	E	E	E	E
AUNITS											
AUTOADJ											
AUTOGOUT	E	E	E	E	E	E	E	E	E	E	E

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
AUTOMSET	E	E	E		E	E	E	E	E	E	E
AUTOQSET	E	E	E	E	E	E	E	E	E	E	E
AUTOSPC	E	E	E	E	E	E	E	E	E	E	E
AUTOSPCR				E							
AUTOSPRT		E		E	E	E	E	E	E	E	
BAILOUT	E	E	E	E	E	E	E	E	E	E	E
BEAMBEA											
BEIGRED				E	E	E	E	E	E	E	
BETA											
BIGER	E	E	E								E
BIGER1	E	E	E								E
BIGER2	E	E	E								E
BUCKLE				E							
					E						
CASPIV	E										
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E
CDIF											
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E
CLOSE		B									
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E
CONFAC	B	B	B	B	B	B	B	B	B	B	B
COUPMASS	E	E	E	E	E	E	E	E	E	E	E
CP1, CP2	E		E	E		E	E		E	E	E
CQC		B									
CURV	E	E	E								E
CURVPLOT	E	E	E	E							E
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E
DBALL	E	E	E	E	E	E	E	E	E	E	E

Table 6-1 PARAMeter Names in SOLs 101 Through 114 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
DBC CONV	E	E	E	E	E	E	E	E	E	E	E
DBC DIAG	E	E	E	E	E	E	E	E	E	E	E
DBC OVWRT	E	E	E	E	E	E	E	E	E	E	E
DB DICT	B	B	B	B	B	B	B	B	B	B	B
DB DN	E	E	E	E	E	E	E	E	E	E	E

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
DB DRPRJ	B	B	B	B	B	B	B	B	B	B	B
DB DRVER	B	B	B	B	B	B	B	B	B	B	B
DB EXT	E	E	E	E	E	E	E	E	E	E	E
DB RCV	E	E	E	E	E	E	E	E	E	E	E
DB UP	E	E	E	E	E	E	E	E	E	E	E
DD RMM									B	B	
DE SPCH											
DE SPCH1											
DFREQ						B			B		
DOPT	E	E	E								E
DPEPS											
DSNOKD			B								
DSZERO	B	B	B								
DYN SPCF		E			E	E	E	E	E	E	
EIGFILT		B							B		
ENFMETH						E	E		E	E	
ENFMOTN						E	E		E	E	
ERPC											
ERPREFDB											
ERPRHO											
ERPRLF											
ERROR	E	E	E	E	E	E	E	E	E	E	E
ESLFSAV											



Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
ESLMOVE											
ESLRFC											
EST	E	E	E	E	E	E	E	E	E	E	E
EXCLUDE			E								
EXTUNIT	B	B	B	B	B	B	B	B	B	B	B
FACTOR	B	B	B	B	B	B	B	B	B	B	B
FBATOLR						E			E		
FBLEND											
FIXEDB	E	E									
FKSYMFAC				E							
FLUIDSE		B			B	B	B	B	B	B	
FOLLOWK		E	E	E	E	E	E	E	E	E	
FRQDEPO						B			B		
FULLSEDR	E	E	E	E	E	E	E	E	E	E	E
FZERO		E		E	E	E	E	E	E	E	
G					E	E	E	E	E	E	
GEOMU	E	E	E	E	E	E	E	E	E	E	E
GFL					B	B	B	B	B	B	
GPECT	E	E	E	E	E	E	E	E	E	E	E
GRAVSET	E	E	E		E	E	E	E	E	E	E
GRDPNT	E	E	E	E	E	E	E	E	E	E	E
GUSTAERO											
GYROAVG						E			E		
HEATSTAT	B										
HFREQ								B	B	B	
HFREQFL								B	B	B	
HTOCITS				E							
HTOCPRT				E							
HTOCTOL				E							
IFP	E	E	E	E	E	E	E	E	E	E	E
INREL	E		E								
IRES	E		E	E		E	E		E	E	E
ITAPE	B	B	B								

Table 6-2 PARAMeter Names in SOLs 101 Through 114 Part 2 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
IUNIT	B	B	B								
KDAMP								B	B	B	
KDAMPFL								B	B	B	
KDIAG				E							
K6ROT	E	E	E	E	E	E	E	E	E	E	E
LANGLE				B							
LFREQ								B	B	B	
LFREQFL								B	B	B	

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
LGDISP				E							
LMFACT	B	B	B								
LMODES		E						B	B	B	
LMODESFL								B	B	B	
LOADU	E	E	E	E	E	E	E	E	E	E	E
LOOPID				E							
MACH											
MAXLP											
MAXRATIO	E	E	E	E	E	E	E	E	E	E	E
MDK4OPT		B						B	B	B	
MDOPT14		B						B	B	B	
MDOTM		B						B	B	B	
MDOTMFAC		B						B	B	B	
MDREDOPT		B						B	B	B	
MESH	B										
METHCMRS		E			E	E	E	E	E	E	
MHRED		E			E	E	E	E	E	E	
MINIGOA	B	B	B	B	B	B	B	B	B	B	B
MODACC								B	B	B	
MODEL	B	B	B	B	B	B	B	B	B	B	B

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
MPCX	B	B	B	B	B	B	B	B	B	B	B
MPTUNIT											
MXICODE0											
NASPRT											
NDAMP											
NEWSET	B										
NHPLUS											
NINTPTS	E	E	E						E		E
NLAYERS				E							
NLHTLS				E							
NLHTOL											
NLHTWK											
NLMAX		E		E	E	E	E	E	E	E	
NLMIN		E		E	E	E	E	E	E	E	
NMLOOP				E							
NMNLFRQ						E			E		
NOCOMPS	E	E	E	E							E
NOELOF	E	E	E								E
NOELOP	E	E	E								E
NOGPF	E	E	E								E
NONCUP										B	
NQSET		E			E	E	E	E	E	E	
NLTOL				B							
NUMOUT	E	E	E								E
NUMOUT1	E	E	E								E
NUMOUT2	E	E	E								E
OCMP	E	E	E	E							E
OEE	E	E	E	E							E
OEF	E	E	E	E	E	E	E	E	E	E	E
OEFX	E	E	E	E	E	E	E	E	E	E	E
OELMOPT	E	E	E	E	E	E	E	E	E	E	E
OELMSET	E	E	E	E	E	E	E	E	E	E	E
OES	E	E	E	E	E	E	E	E	E	E	E

Table 6-3 PARAMeter Names in SOLs 101 Through 114 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OESE	E	E	E	E	E	E	E	E	E	E	E
OESX	E	E	E	E	E	E	E	E	E	E	E
OG	E	E	E	E							E
OGEOM	E	E	E	E	E	E	E	E	E	E	E
OGPF	E	E	E	E							E
OGPS	E	E	E				E			E	E
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E
OGRDSET	E	E	E	E	E	E	E	E	E	E	E
OLDSEQ	B	B	B	B	B	B	B	B	B	B	B
OMAXR	E	E	E	E	E	E	E	E	E	E	E
OMID	E	E	E			E	E	E	E	E	E
OMSGLVL	E	E	E	E	E	E	E	E	E	E	E
OPCHSET	E	E	E	E	E	E	E	E	E	E	E
OPG	E		E	E		E	E		E	E	E
OPGEO											

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OPGTKG											
OPPHIB											
OPPHIPA											
OPTEXIT											
OPTION		B									
OQG	E	E	E	E	E	E	E	E	E	E	E
OSETELE	E	E	E	E	E	E	E	E	E	E	E
OSETGRD	E	E	E	E	E	E	E	E	E	E	E
OSWELM	B	B	B	B	B	B	B	B	B	B	B
OSWPPT	B	B	B	B	B	B	B	B	B	B	B
OUG	E	E	E	E	E	E	E	E	E	E	E
OUGC	E	E	E	E	E	E	E	E	E	E	E
OUGSP				E		E	E		E	E	

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
OUMU		E									
OUNIT1	E	E	E	E	E	E	E	E	E	E	E
OUNIT2	E	E	E	E	E	E	E	E	E	E	E
OUTOPT	E	E	E	E							E
PACINFID						B			B		
PATPLUS	E	E	E	E	E	E	E	E	E	E	E
PATVER	E	E	E	E	E	E	E	E	E	E	E
PCOMPRM	B					B	B		B	B	
PDRMSG	E	E	E	E	E	E	E	E	E	E	E
PENFN	B	B	B								
PLTMSG	E	E	E	E	E	E	E	E	E	E	E
POST	E	E	E	E	E	E	E	E	E	E	E
POSTEXT	E	E	E	E	E	E	E	E	E	E	E
POSTU	E	E	E	E	E	E	E	E	E	E	E
PREFDB							E	E	E	E	
PRINT											
PROUT	E	E	E	E	E	E	E	E	E	E	E
PRPA	E		E								
PRPHIVZ		E			E	E	E	E	E	E	E
PRPJ	E		E			E	E		E	E	E
PRTBGPDT	E	E	E	E	E	E	E	E	E	E	E
PRTCSTM	E	E	E	E	E	E	E	E	E	E	E
PRTEQXIN	E	E	E	E	E	E	E	E	E	E	E
PRTGPDT	E	E	E	E	E	E	E	E	E	E	E
PRTGPL	E	E	E	E	E	E	E	E	E	E	E
PRTGPTT	E	E	E	E	E	E	E	E	E	E	E
PRTMAXIM	E	E	E								
PRTMGG	E	E	E	E	E	E	E	E	E	E	E
PRTPG	E	E	E	E	E	E	E	E	E	E	E
PRTRESLT	E	E	E	E	E	E	E	E	E	E	E
Q											
RBSETPRT	E	E	E	E	E	E	E	E	E	E	E
RESLTOPT	E	E	E	E	E	E	E	E	E	E	E

Table 6-4 PARAMeter Names in SOLs 101 Through 114 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
RFORSET	E	E	E		E	E	E	E	E	E	E
RHOCP											
RSPECTRA							E			E	
RSPRINT							E			E	
RSTTEMP				E							
S1	E	E	E	E			E			E	E
S1A	E	E	E	E							E
S1AG	E	E	E	E							E
S1AM	E	E	E	E							E
S1G	E	E	E	E							E
S1M	E	E	E	E							E
SCRSPEC		B									
SEMAP	B	B	B	B	B	B	B	B	B	B	B
SEMAPOPT	B	B	B	B	B	B	B	B	B	B	B
SEMAPPRT	B	B	B	B	B	B	B	B	B	B	B
SENSUOO											
SEP1XOVR	B	B	B	B	B	B	B	B	B	B	B
SEQOUT	B	B	B	B	B	B	B	B	B	B	B
SERST	B	B	B	B	B	B	B	B	B	B	B
SESDAMP		E			E	E	E	E	E	E	
SESEF		E									

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
SHLDAMP									E		
SIGMA											
SKINOUT		E			E	E	E	E	E	E	
SKPAMP											
SLOOPID											
SMALLQ	B	B	B	B	B	B	B	B	B	B	B
SNORM	B	B	B	B	B	B	B	B	B	B	B

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)										
	101	103	105	106	107	108	109	110	111	112	114
SNORMPRT	B	B	B	B	B	B	B	B	B	B	B
SOFTEXIT											
SOLADJC											
SOLID	B	B	B	B	B	B	B	B	B	B	B
SPARSEDM	E	E	E			E	E		E	E	
SPARSEDR	E	E	E			E	E		E	E	
SPARSEPH		E						E	E	E	
SPDDMAT											
SPDRRAT	E	E	E			E	E		E	E	
SQSETID	E	E	E	E	E	E	E	E	E	E	E
SRTELTYP	E	E	E								E
SRTOPT	E	E	E								E
START	B	B	B	B	B	B	B	B	B	B	B
STIME							E			E	
SUBCASID				E							
SUBID				E							
SUPAERO											
SUPER	B	B	B	B	B	B	B	B	B	B	B
TABID							E			E	
TABS											
TESTNEG				E							
TFSYMFAC					E	E	E	E	E	E	
TINY	E	E	E								E
TOLRSC	B	B	B	B	B	B	B	B	B	B	B
TSTATIC											
UNSYMF				B							
UNSYMKB		B			B	B	B	B	B	B	
UPDTBSH											
USETPRT	E	E	E	E	E	E	E	E	E	E	E
USETSEL	E	E	E	E	E	E	E	E	E	E	E
VARPHI	C										
VMOPT		E	E	E	E	E	E	E	E	E	E
VREF						B				B	

Table 6-5 PARAMeter Names in SOLs 101 Through 114 Part 5 (continued)

PARAMeter Name	Solution Sequence Numbers (101 through 114)											
	101	103	105	106	107	108	109	110	111	112	114	
WTMASS	E	E	E	E	E	E	E	E	E	E	E	E
W3							E			E		
W3FL							E			E		
W4							E			E		
W4FL							E			E		
XFACT		B							B			
XFACTX		B							B			
XFLAG	E		E									E
XYUNIT												
ZROCMAS		E		E	E	E	E	E	E	E	E	
ZROVEC		E		E	E	E	E	E	E	E	E	

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
ACEXTMTD										B			
ACEXTSET										B			
ACOUT										E			
ACOWEAK										B			
ACSYM										B			
ADJMETH										B			
ADPCON				E				E	E		E		
ADSTAT													
AERODOF					E	E	E			E			
AESDISC					B					B			
AESMAXIT					B					B			
AESMETH					B					B			
AESRNDM					B					B			
AESTOL					B					B			
ALPHA1			B	E		B	B	B	B	B	E		
ALPHA2			B	E		B	B	B	B	B	E		
ALPHA1FL													



Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
ALPHA2FL													
ALTRED													
ARBMASP													
ARBMFEM													
ARBMP5													
ARBMS5													
ARBMSTYP													
ARF													
ARS													
ASC0UP													
ASING													
AUNITS													
AUTOADJ													
AUTOGOUT	E	E	E	E	E	E	E	E	E	E	E		
AUTOMSET	E	E	E		E	E	E			E		E	
AUTOQSET	E	E	E	E	E	E	E	E	E	E			
AUTOSPC													
AUTOSPCR											E		
AUTOSPRT	E		E	E	E	E	E		E		E		
BAILOUT	E	E	E	E	E	E	E	E	E	E	E		
BEAMBEA													
BEIGRED			E	E		E	E			E	E		
BETA				E					E				
BIGER	E	E			E			E		E			
BIGER1	E	E			E			E		E			
BIGER2	E	E			E			E		E			
BUCKLE													
CASIEMA											E		
CASPIV										E	E		
CB1, CB2	E	E	E	E	E	E	E	E	E	E	E		
CDIF										E			
CFDIAGP	E	E	E	E	E	E	E	E	E	E	E		
CFRANDEL	E	E	E	E	E	E	E	E	E	E	E		

Table 6-6 PARAMeter Names in SOLs 115 Through 700 Part 1 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
CHECKOUT	E	E	E	E	E	E	E	E	E	E	E	E	
CK1, CK2, CK3	E	E	E	E	E	E	E	E	E	E	E	E	
CLOSE	B												
CM1, CM2	E	E	E	E	E	E	E	E	E	E	E		
CONFAC	B	B	B	B	B	B	B	B	B	B	B		
COUPMASS	E	E	E	E	E	E	E	E	E	E	E		
CP1, CP2		E	E	E	E		E	E	E	E			
CQC	E												
CURV	E	E			E			E		E			
CURVPLOT	E	E			E					E	E		
CWDIAGP	E	E	E	E	E	E	E	E	E	E	E		
CWRANDEL	E	E	E	E	E	E	E	E	E	E	E		
DBALL	E		E	E	E	E	E	E	E	E	E		
DBC CONV	E		E	E	E	E	E	E	E	E	E		

Table 6-7 PARAMeter Names in SOLs 115 Through 700 Part 2

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DBC DIAG	E		E	E	E	E	E	E	E	E	E		
DBC OVWRT	E		E	E	E	E	E	E	E	E	E		
DB DICT	B		B	B	B	B	B	B	B	B	B		
DB DRPRJ	B		B	B	B	B	B	B	B	B	B		
DB DRVER	B		B	B	B	B	B	B	B	B	B		
DB DN	E		E	E	E	E	E	E	E	E	E		
DB EXT	E	E	E	E	E	E	E	E	E	E	E		
DB RCV	E		E	E	E	E	E	E	E	E	E		
DB UP	E		E	E	E	E	E	E	E	E	E		
DD RMM							B			E			
DE ACTEL											E		
DE CLUMP													B
DE SPCH										E			
DE SPCH1										E			

Table 6-7 PARAMeter Names in SOLs 115 Through 700 Part 2 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DFREQ			B				B			B			
DOPT	E				E			E		E			
DPEPS										B			
DPHFLG													
DSNOKD										B			
DSZERO													
DV3PASS										B			

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
DYNSPCF	E		E			E	E			E			
ENFMETH							E			E			
ENFMOTN							E			E			
EPPRT													
EPSILONT													
EPZERO													
ERPC													
ERPREFDB													
ERPRHO													
ERPRLF													
ERROR	E		E	E	E	E	E	E	E	E	E	E	
ESLFSAV											E		
ESLLCOMP											E		
ESLMOVE											E		
ESLMPC1											E		
ESLOPTEX											E		
ESLPRT											E		
ESLPRT1											E		
ESLRCF											E		
EULBND													B
EULSTRES													B
EXTUNIT	B		B	B	B	B	B	B	B	B	B		
FACTOR	B		B	B	B	B	B	B	B	B	B		
FBATOLR													
FBLEND													B
FIRSTKI						B	B			B			
FIXEDB										E			
FKSYMFAC											E		
FLEXINCR													
FLUIDMP													
FLUIDSE										B			
FMULTI													B

Table 6-8 PARAMeter Names in SOLs 115 Through 700 Part 3 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
FOLLOWK	E	E								E	E	B	
FRQDEPO							B			B			
FULLSEDR	E	E	E	E	E	E	E	E	E	E	E		
FZERO											E		
G			E	E		E	E			E	E		
GEOMU	E		E	E	E	E	E	E	E	E	E		
GPECT	E		E	E	E	E	E	E	E	E	E		
GRADMESH													B
GRAVSET	E	E	E	E	E	E	E	E	E	E	E		
GRDPNT	E		E	E	E	E	E	E	E	E	E	E	

Table 6-9 PARAMeter Names in SOLs 115 Through 700 Part 4

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
GUSTAERO						B	B			B			
GYROAVG													
HEATCMD												B	
HEATSTAT													
HFREQ						B	B			B			
HFREQFL							B			B			
HTOCITS													
HTOCPRT													
HTOCTOL													
IFP	E		E	E	E	E	E	E	E	E			
INREL										E			
INRLM													
IRES			E	E	E	E	E	E	E	E	E		
ITAPE										B			
IUNIT										B			
KDAMP						B				B			
KDAMPFL							B			B			
KDIAG								E			E		
K6ROT	E		E	E	E	E	E	E	E	E	E		
LANGLE				B				B	B		B		
LDSUM					E					E			
LFREQ						B	B			B			
LFREQFL							B			B			
LGDISP				E				E	E		E	B	
LMFACT											B		
LMODES						B	B			B			
LMODESFL							B			B			
LOADU	E		E	E				E	E	E	E		
LOOPID				E				E	E		E		
MACH							B						
MALIAS02												B	
MALIAS03												B	

Table 6-9 PARAMeter Names in SOLs 115 Through 700 Part 4 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARALPHA												B	
MARAUTO												B	
MARBK105												B	
MARBK106												B	
MARC3901												B	
MARC3D												C	
MARC4401												B	
MARCASUM												B	
MARCAUTO												B	
MARCAXEL												B	
MARCBEAM												B	
MARCBUG												C	
MARCBUSK												B	
MARCCBAR												B	
MARCCENT												B	
MARCCON2												B	
MARCCON3												B	
MARCCPY												B	
MARCDEF												B	
MARCDILT												B	
MARCDIS2												B	
MARCDIS3												B	
MARCDIS4												B	
MARCDMIG												B	
MARCEKND												B	
MARCEXIT												B	
MARCFEAT,N												B	
MARCFIL												B	
MARCFRIC												B	

Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCGAPP												B	
MARCGLUE												B	
MARCHOST												B	
MARCIAMN												B	
MARCITER												B	
MARCL001												B	
MARCLOWE												B	
MARCLUMP												B	
MARCMATT												B	
MARCMEM												B	
MARCMNF												B	
MARCMPII												B	
MARCND99												B	
MARCNOER												B	
MARCOFFT												B	
MARCONLY												B	
MARCONTF												B	
MARCOOCC												B	
MARCOPT												B	
MARCOTIM												B	
MARCPARR												B	
MARCPENT												B	
MARCPINN												B	
MARCPOS												B	
MARCPOST												B	
MARCPR99												B	
MARCPRES												B	
MARCPRN												B	
MARCPRNG												B	
MARCPRNR												B	
MARCPROG												B	
MARCPH												B	



Table 6-10 PARAMeter Names in SOLs 115 Through 700 Part 5 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCRBAL												B	
MARCRBAR												B	
MARCRBE2												B	
MARCRBE3												B	
MARCREVR												B	
MARCRIGD												B	
MARCRUN												B	
MARCSAME												C	
MARCSCLR												B	
MARCSETS												B	
MARCSSETT												B	
MARCSINC												B	
MARCSIZ3												B	
MARCSIZ4												B	
MARCSIZ5												B	
MARCSIZ6												B	
MARCSLHT												B	
MARCSOLV												B	
MARCSPPC												B	
MARCSSTIFF												B	
MARCSUMY												B	
MARCT16												B	
MARCT19												B	
MARCTABL												B	
MARCTEDF												B	
MARCTEDN												B	
MARCTEMP												B	
MARCTETT												B	
MARCTIEC												B	
MARCTOL												B	
MARCTOTD												B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MARCTOTL												B	
MARCTOTT												B	
MARCTUBE												B	
MARCTVL												B	
MARCUSUB												B	
MARCVERS												B	
MARCWDIS												B	
MARCWELD												B	
MARCWIND												B	
MARELSTO												B	
MARFACEA												B	
MARFACEB												B	
MARFATAL												B	
MARGPFOR												B	
MARIBOOC												B	
MARIPROJ												B	
MARMPCHK												B	
MARMPCID												C	
MARNOSET												B	
MARNOT16												B	
MARPLANE												B	
MARPROCS													
MARRBAR2												B	
MARROUTT												B	
MARSHRII												B	
MARUPDAT												B	
MAXIREVV												B	
MAXLP				E					E				
MAXRATIO	E		E	E	E	E	E	E	E	E	E	E	
MBENDCAP												B	
MCNLPARM												B	
MCON2D3D												B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MCORDUPD												B	
MCSSHLCK												C	
MCSSHORR												C	
MDAREAMD												B	
MDK4OPT										B			
MDOPT14										B			
MDOTM										B			
MDOTMFAC										B			
MDREDOPT										B			
MDUMLOAD												B	
MESH								B	B	B			
METHCMRS			E	E		E	E		E	E			
MEXTRNOD												B	
MEXTSEE												C	
MFASTCMP												B	
MFORCOR1												B	
MFSKIPPP												B	
MGAPINIT												B	
MGLUETOL												B	
MHEATSHL												B	
MHEATUNT												B	
MHOUBOLT												B	
MHRED	E		E	E	E	E	E			E			
MICRO													B
MINCLDSB												B	
MINIGOA	B		B	B	B	B	B	B	B	B	B	B	
MINRECCC												B	
MINSOUTT												B	
MINVASHF												B	
MINVCITR												B	
MINVCSHF												B	
MINVCTOL												B	
MINFMAX												B	

Table 6-11 PARAMeter Names in SOLs 115 Through 700 Part 6 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MINVNM0D												B	
MLSTRAIN												B	
MLSTRAN2												B	
MMEMDETT												B	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MNASTLDS												B	
MODACC						B	B						
MODEL	B		B	B	B	B	B	B	B	B	B	B	
MOFFCORE												B	
MOP2TITL												B	
MOUTRCOR												B	
MPCX	B		B	B	B	B	B	B	B	B	B	B	
MPERMPRT												B	
MPTUNIT							E						
MRAFFLOR												B	
MRAFFLOT												B	
MRAFFLOW												B	
MRALIAS												B	
MRBE2SNG												B	
MRBEAMB												B	
MRBEPARAM												B	
MRBIGMEM												B	
MRBUKMTH												B	
MRC2DADD												B	
MRCKBODY												B	
MRCKLOAD												B	
MRCONADD												B	
MRCONTAB												B	
MRCONUSE												C	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRCONRES												B	
MRCONVER												B	
MRCOORDS												B	
MRCWANGL												B	
MRCWELD												B	
MRDELTTT												B	
MRDISCMB												B	
MRDYNOLD												B	
MREIGMTH												B	
MREL1103												B	
MRELRB												B	
MRELSPCD												B	
MRENUELE												B	
MRENUGRD												B	
MRENUMBR												B	
MRENUMMT												B	
MRESTALL												B	
MRESULTS												B	
MREVPLST												B	
MRFINITE												B	
MRFOLLO2												B	
MRFOLLOW												B	
MRFOLLOW1												B	
MRFOLLOW3												B	
MRFOLLOW4												B	
MRGAPUSE												B	
MRHYPMID												B	
MRITTYPE												B	
MRMAT8A3												B	
MRMAT8E3												B	
MRMAT8N1												B	
MRMAT8N3												B	
MRMAXISZ												B	

Table 6-12 PARAMeter Names in SOLs 115 Through 700 Part 7 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRMAXNUM												B	
MRMEMSUM												B	
MRMTXKGG												B	
MRNOCOR												B	
MRORINTS												B	
MRPARALL												B	
MRPELAST												B	
MRPBUSHT												B	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRPIDCHK												B	
MRPLOAD4												B	
MRPLOAD4R												B	
MRPLSUPD												B	
MRPREFER												B	
MRPSHCMP												B	
MRPSHELL												B	
MRRELNOD												B	
MRRCFILE												B	
MRSETNAM												B	
MRSPAWN2												B	
MRSPLINE												B	
MRSRING												B	
MRT16STP												B	
MRTABLIM												B	
MRTABLS1												B	
MRTABLS2												B	
MRTABPRC												B	
MRTFINAL												B	
MRTIMING												B	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
MRTHREAD												B	
MRTSHEAR												B	
MRUSDATA												B	
MSOLMEM												B	
MSPCCCHK												B	
MSPCKEEP												B	
MSPEEDCB												B	
MSPEEDCW												B	
MSPEEDOU												B	
MSPEEDP4												B	
MSPEEDSE												B	
MSOLMEM, MBYTE												B	
MSTFBEAM												B	
MTABINGNR												B	
MTABLD1M												B	
MTABLD1T												B	
MTEMPDWN												B	
MTLD1SID												B	
MULRFORC												B	
MUMPSOLV												B	
MUSBKEEP												B	
MWINQUOT												B	
MWLDGSGA												B	
MXLAGM1										B			
NASPRT										B			
NDAMP				E					E		E		
NDAMPM											E		
NEWSET													
NINTPTS	E				E			E		E			
NLAYERS				E								E	
NLHTLS								E				E	
NLMAX												E	

Table 6-13 PARAMeter Names in SOLs 115 Through 700 Part 8 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
NLMIN										E	E		
NMLOOP													
NMNLFRQ										E*	E*		
NOCOMPS	E				E			E		E	E		
NOELOF	E				E			E		E			
NOELOP	E				E			E		E			
NOGPF	E				E			E		E			
NOMSGSTR	E				E			E		E			
NONCUP										B			
NQSET	E		E	E	E	E	E		E				
NLTOL									E		B		
NUMOUT	E				E			E		E			
NUMOUT1	E				E			E		E			

\* ANALYSIS=MFREQ or ANALYSIS=DFREQ only.

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
NUMOUT2	E				E			E		E			
OCMP	E									E	E		
OEE	E									E	E	E	
OEF	E		E	E	E	E	E	E	E	E	E	E	
OEFX	E		E	E	E	E	E	E	E	E	E	E	
OELMOPT	E	E	E	E	E	E	E	E	E	E	E		
OELMSET	E	E	E	E	E	E	E	E	E	E	E		
OES	E		E	E	E	E	E	E	E	E	E	E	
OESE	E		E	E	E	E	E	E	E	E	E	E	
OESX	E		E	E	E	E	E	E	E	E	E	E	
OG	E				E			E		E	E	E	
OGEOM	E		E	E	E	E	E	E	E	E	E	E	
OGPF	E				E					E	E	E	
OGPS	E				E					E			



Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
OGRDOPT	E	E	E	E	E	E	E	E	E	E	E		
OGRDSET	E	E	E	E	E	E	E	E	E	E	E		
OLDSEQ	B		B	B	B	B	B	B	B	B	B	B	
OMAXR	E		E	E	E	E	E	E	E	E	E	E	
OMID	E	E	E	E	E	E	E	E	E	E			
OMSGLVL	E	E	E	E	E	E	E	E	E	E	E		
OPCHSET	E	E	E	E	E	E	E	E	E	E	E		
OPG			E	E		E	E		E	E	E	E	
OPGEOM					E	E	E			E			
OPGTKG					B	B	B			B			
OPPHIB						B	B			B			
OPPHIPA						B	B			B			
OPTEXIT										B			
OPTION	B												
OQG	E		E	E	E	E	E	E	E	E	E	E	
OSETELE	E	E	E	E	E	E	E	E	E	E	E		
OSETGRD	E	E	E	E	E	E	E	E	E	E	E		
OSWELM	B	B	B	B	B	B	B			B	B	B	
OSWPPT	B	B	B	B	B	B	B			B	B	B	
OUG	E		E	E	E	E	E	E	E	E	E	E	
OUGCORD	E		E	E	E	E	E	E	E	E	E		
OUGSPLIT			E	E			E		E	E	E		
OUMU	E									E			
OUNIT1	E		E	E	E	E	E	E	E	E	E		
OUNIT2	E		E	E	E	E	E	E	E	E	E		
OUTOPT	E				E			E		E	E		
PARTMEM													
PATPLUS	E		E	E	E	E	E	E	E	E	E		
PATVER	E		E	E	E	E	E	E	E	E	E		
PDRMSG	E		E	E	E	E	E	E	E	E	E		
PEDGEPE													
PENFN												B	
PH2OUT												B	

Table 6-14 PARAMeter Names in SOLs 115 Through 700 Part 9 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
PLTMSG	E		E	E	E	E	E	E	E	E	E		
POST	E		E	E	E	E	E	E	E	E	E	E*	
POSTEXT	E	E	E	E	E	E	E	E	E	E	E		
POSTU	E		E	E	E	E	E	E	E	E	E		
PREFDB										E			
PRINT						B				B			
PROUT	E		E	E	E	E	E	E	E	E	E		
PRPA					E			E		E			
PRPHIVZ	E		E	E	E	E	E		E	E			
PRPJ			E		E		E	E	E	E			
PRTCSTM	E		E	E	E	E	E	E	E	E	E		
PRTEQXIN	E		E	E	E	E	E	E	E	E	E		
PRTGPDT	E		E	E	E	E	E	E	E	E	E		
PRTGPL	E		E	E	E	E	E	E	E	E	E		
PRTGPTT	E		E	E	E	E	E	E	E	E	E		
PRTMAXIM	E		E	E	E	E	E	E	E	E			
PRTMGG	E		E	E	E	E	E	E	E	E	E		
PRTPG	E		E	E	E	E	E	E	E	E	E		
PSENPCH										B			

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
PVALINIT										B			
Q							B						
RADMOD								B	B		B		
RBSETPRT	E	E	E										
RCONTACT												B	
RESLTOPT	E	E	E	E	E	E	E	E	E	E	E		
RFORSET	E	E	E	E	E	E	E	E	E	E	E		
RHOCP													
RKSCHEME													B

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
RMSINT													
ROHYDRO													B
ROMULTI													B
ROSTR													B
RSPECTRA													
RSPRINT													
RSTTEMP													
S1	E				E			E		E	E		
S1A	E				E			E		E	E		
S1AG	E				E			E		E	E		
S1AM	E				E			E		E	E		
S1G	E				E			E		E	E		
S1M	E				E			E		E	E		
SCRSPEC	B												
SEMAP	B		B	B	B	B	B	B	B	B	B		
SEMAPOPT	B		B	B	B	B	B	B	B	B	B		
SEMAPPRT	B		B	B	B	B	B	B	B	B	B		
SENSUOO										B			
SEP1XOVR	B		B	B	B	B	B	B	B	B	B		
SEQOUT	B		B	B	B	B	B	B	B	B	B		
SERST	B		B	B	B	B	B	B	B	B	B		
SESDAMP	E	E	E	E		E							
SESEF	E												
SHLDAMP										E			
SIGMA									B				
SKINOUT										E			
SKPAMP						B	B			B			
SLOOPID				E									
SMALLQ	B		B	B	B	B	B	B	B	B	B		
SNORM	B		B	B	B	B	B	B	B	B	B		
SNORMPRT	B		B	B	B	B	B	B	B	B	B		
SOFTEXIT										B			
SOLADJC										B			

Table 6-15 PARAMeter Names in SOLs 115 Through 700 Part 10 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
SOLID	B		B	B	B	B	B	B	B	B	B		
SPARSEDM					E					E			
SPARSEDR					E					E			
SPARSEPH						E	E			E			
SPDMRAT										E			
SPDRRAT					E					E			
SQSETID	E	E	E	E	E	E	E	E	E	E	E		
STEPFCT													B
STEPFCTL													B
SRTETYP	E				E			E		E			
SRTOPT	E				E			E		E			
START	B		B		B	B	B	B	B	B	B		
STIME				E					E				
STRUCTMP										E			
SUBCASID											E		
SUBID								E			E		
SUPAERO					E	E	E			E			
SUPDOF					B								
SUPER	B		B	B	B	B	B	B	B	B	B		
TABID													
TABS								E	E	E			
TESTNEG								E			E		
TFSYMFAC			E	E		E	E	E	E	E	E		

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
TINY	E				E			E		E			
TOLRSC	B		B	B	B	B	B	B	B	B	B		
TOPOCONV											E		
TSTATIC				E					E		E	E	
UNSYMF											B		

Table 6-16 PARAMeter Names in SOLs 115 Through 700 Part 11 (continued)

PARAMeter Name	Solution Sequence Numbers (115 through 700)												
	115	116	118	129	144	145	146	153	159	200	400	600	700
UNSYMKB	B		B	B		B	B			B	B		
UPDTBSH										E			
USETPRT	E		E	E	E	E	E	E	E	E	E		
VARPHI											C		
VELCUT													B
VMOPT	E		E	E	E	E	E	E	E	E	E		
VREF						B				B			
WTMASS	E		E	E	E	E	E	E	E	E	E	B	
W3				B					E	B	E	B	
W3FL										B			
W4				B					E	B	E	B	
W4FL										B			
XFLAG					E					E			
XYUNIT					E					E			
ZROCMAS	E		E	E	E	E	E			E	E		
ZROVEC	E		E	E	E	E	E			E	E		



# 7

## Item Codes

- Item Code Description
- Element Stress (or Strain) Item Codes
- Element Force Item Codes
- Fluid Virtual Mass Pressure Item Codes
- 2D Slideline and 3D Surface Contact Item Codes
- Element Strain Energy Item Codes
- Fatigue Item Codes for LOC=ELEM on FTGPARM
- Fatigue Item Codes for LOC=NODE or NODA on FTGPARM
- Item Codes for Fatigue Analysis of Spot Welds
- Item Codes for Fatigue Analysis of Seam Welds
- Equivalent Radiated Power (ERP) Item Codes

## Item Code Description

Item codes are integer numbers assigned to specific output quantities; such as, the third translational component of displacement, the major principal stress at Z1 in a CQUAD4 element, or the torque in a CBAR element. Item codes are specified on the following input statements:

- DRESP1 entry for Design Sensitivity and Optimization (SOL 200).
- X-Y Plotting commands. See [Plotting](#) in the *MSC Nastran Reference Guide*.
- DTI,INDTA entry for stress sorting.

The following tables provide item codes for:

- [Table 7-1](#). Element Stress or Strain.
- [Table 7-5](#). Element Force.
- [Table 7-7](#). Fluid Virtual Mass Pressure.
- [Table 7-8](#). Heat Transfer Flux.
- [Table 7-9](#). Slideline Contact Output.
- [Table 7-12](#). Element Strain Energy Item Codes.
- [Table 7-13](#). Fatigue Item Codes (LOC=ELEM)
- [Table 7-14](#) Fatigue Item Codes (LOC=NODE or NODA)
- [Table 7-15](#). Item Codes for Fatigue Analysis of Spot Welds
- [Table 7-16](#). Item Codes for Fatigue Analysis of Seam Welds
- [Table 7-17](#) Equivalent Radiated Power (ERP) Item Codes

The following superscripts appear in the tables and indicate:

1. Data for components marked with the symbol (1) are included in the data block MES and MEF outputs from modules DRMH1/DRMH3 and DRMS1. DRMH1/DRMH3 are used in EXTSEOUT Case Control processing. (See [MSC Nastran DMAP Programmer's Guide](#).)
2. Composite Element Stresses and Failure Indices.

## Element Stress (or Strain) Item Codes

All item codes refer to stresses (or strains) unless otherwise denoted.

If output is magnitude/phase, the magnitude replaces the real part, and the phase replaces the imaginary part. Strain item codes are equivalent to stress item codes. However, strain is computed for only some elements. See [Table 3-1](#) in the *MSC Nastran Reference Guide*.



Table 7-1 Element Stress-Strain Item Codes

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CAXIF2 (47)	2 <sup>1</sup>	Radial axis	2 <sup>1</sup>	Radial axis	RM
	3 <sup>1</sup>	Axial axis	3 <sup>1</sup>	Axial axis	RM
	4 <sup>1</sup>	Tangential edge	4 <sup>1</sup>	Tangential edge	RM
	5 <sup>1</sup>	Circumferential edge	5 <sup>1</sup>	Circumferential edge	RM
			6 <sup>1</sup>	Radial axis	IP
			7 <sup>1</sup>	Axial axis	IP
			8 <sup>1</sup>	Tangential edge	IP
			9 <sup>1</sup>	Circumferential edge	IP
CAXIF3 (48)	2 <sup>1</sup>	Radial centroid	2 <sup>1</sup>	Radial centroid	RM
	3 <sup>1</sup>	Circumferential centroid	3 <sup>1</sup>	Circumferential centroid	RM
	4 <sup>1</sup>	Axial centroid	4 <sup>1</sup>	Axial centroid	RM
	5 <sup>1</sup>	Tangential edge 1	5 <sup>1</sup>	Tangential edge 1	RM
	6 <sup>1</sup>	Circumferential edge 1	6 <sup>1</sup>	Circumferential edge 1	RM
	7 <sup>1</sup>	Tangential edge 2	7 <sup>1</sup>	Tangential edge 2	RM
	8 <sup>1</sup>	Circumferential edge 2	8 <sup>1</sup>	Circumferential edge 2	RM
	9 <sup>1</sup>	Tangential edge 3	9 <sup>1</sup>	Tangential edge 3	RM
	10 <sup>1</sup>	Circumferential edge 3	10 <sup>1</sup>	Circumferential edge 3	RM
			11 <sup>1</sup>	Radial centroid	IP
			12 <sup>1</sup>	Circumferential centroid	IP
			13 <sup>1</sup>	Axial centroid	IP
			14 <sup>1</sup>	Tangential edge 1	IP
			15 <sup>1</sup>	Circumferential edge 1	IP
			16 <sup>1</sup>	Tangential edge 2	IP
			17 <sup>1</sup>	Circumferential edge 2	IP
			18 <sup>1</sup>	Tangential edge 3	IP
			19 <sup>1</sup>	Circumferential edge 3	IP
CAXIF4 (49)	2 <sup>1</sup>	Radial centroid	2 <sup>1</sup>	Radial centroid	RM
	3 <sup>1</sup>	Circumferential centroid	3 <sup>1</sup>	Circumferential centroid	RM
	4 <sup>1</sup>	Axial centroid	4 <sup>1</sup>	Axial centroid	RM
	5 <sup>1</sup>	Tangential edge 1	5 <sup>1</sup>	Tangential edge 1	RM
	6 <sup>1</sup>	Circumferential edge 1	6 <sup>1</sup>	Circumferential edge 1	RM

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7 <sup>1</sup>	Tangential edge 2	7 <sup>1</sup>	Tangential edge 2	RM
	8 <sup>1</sup>	Circumferential edge 2	8 <sup>1</sup>	Circumferential edge 2	RM
	9 <sup>1</sup>	Tangential edge 3	9 <sup>1</sup>	Tangential edge 3	RM
	10 <sup>1</sup>	Circumferential edge 3	10 <sup>1</sup>	Circumferential edge 3	RM
	11 <sup>1</sup>	Tangential edge 4	11 <sup>1</sup>	Tangential edge 4	RM
	12 <sup>1</sup>	Circumferential edge 4	12 <sup>1</sup>	Circumferential edge 4	RM
			13	Radial centroid	IP
			14	Circumferential centroid	IP
			15	Axialcentroid	IP
			16	Tangential edge 1	IP
			17	Circumferential edge 1	IP
			18	Tangential edge 2	IP
			19	Circumferential edge 2	IP
			20	Tangential edge 3	IP
			21	Circumferential edge 3	IP
			22	Tangential edge 4	IP
			23	Circumferential edge 4	IP
<b>CAXISYM (241)</b>	2	Z1-Fiber distance 1			
Nonlinear	3	Stress-X			
	4	Stress-Y			
	5	Stress-XY			
	6	Strain-X			
	7	Strain-Y			
	8	Strain XY			
	9-15	Item 9 through 15 repeated for fiber distance Z2			
<b>CBAR (34)</b>	2 <sup>1</sup>	End A-Point C	2 <sup>1</sup>	End A-Point C	RM
Linear	3 <sup>1</sup>	End A-Point D	3 <sup>1</sup>	End A-Point D	RM
	4 <sup>1</sup>	End A-Point E	4 <sup>1</sup>	End A-Point E	RM
	5 <sup>1</sup>	End A-Point F	5 <sup>1</sup>	End A-Point F	RM
	6 <sup>1</sup>	Axial	6 <sup>1</sup>	Axial	RM
	7	End A maximum	7 <sup>1</sup>	End A-Point C	IP
	8	End A minimum	8 <sup>1</sup>	End A-Point D	IP

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	9	Safety margin in tension	9 <sup>1</sup>	End A-Point E	IP
	10 <sup>1</sup>	End B-Point C	10 <sup>1</sup>	End A-Point F	IP
	11 <sup>1</sup>	End B-Point D	11 <sup>1</sup>	Axial	IP
	12 <sup>1</sup>	End B-Point E	12 <sup>1</sup>	End B-Point C	RM
	13 <sup>1</sup>	End B-Point F	13 <sup>1</sup>	End B-Point D	RM
	14	End B maximum	14 <sup>1</sup>	End B-Point E	RM
	15	End B minimum	15 <sup>1</sup>	End B-Point F	RM
	16	Safety margin in compression	16 <sup>1</sup>	End B-Point C	IP
			17 <sup>1</sup>	End B-Point D	IP
			18 <sup>1</sup>	End B-Point E	IP
			19 <sup>1</sup>	End B-Point F	IP
<b>CBAR (100)</b>	2	Station Distance/Length	2	Station Distance/Length	RM
Intermediate Stations	3	Point C	3	Point C	RM
	4	Point D	4	Point D	RM
	5	Point E	5	Point E	RM
	6	Point F	6	Point F	RM
	7	Axial	7	Axial	RM
	8	Maximum	8	Maximum	RM
	9	Minimum	9	Minimum	RM
	10	Margin of Safety	10	Point C	IP
		(Item codes above are given for End A. For codes 2 through 10 at intermediate stations add (K-1)*9 where K is the station number, and for codes at End B, K=number of stations plus 1.)	11	Point D	IP
			12	Point E	IP
			13	Point F	IP
			14	Axial	IP
			15	Maximum	IP
			16	Minimum	
					(Item codes above are given for End A. For codes 2 through 16 at intermediate stations add (K-1)*15 where K is the station number, and for codes at End B, K=number of stations plus 1.)

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBAR (238) Arbitrary Cross Section	2	Station Distance/Length			
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in zx			
	8	Min Shear Stress in zx			
	9	Max vonMises Stress		(Not Supported)	
	CBEAM (2) Linear	2	External grid point ID	2	External grid point ID
3		Station distance/length	3	Station distance/length	
4 <sup>1</sup>		Long. Stress at Point C	4 <sup>1</sup>	Long. Stress at Point C	RM
5 <sup>1</sup>		Long. Stress at Point D	5 <sup>1</sup>	Long. Stress at Point D	RM
6 <sup>1</sup>		Long. Stress at Point E	6 <sup>1</sup>	Long. Stress at Point E	RM
7 <sup>1</sup>		Long. Stress at Point F	7 <sup>1</sup>	Long. Stress at Point F	RM
8		Maximum stress	8 <sup>1</sup>	Long. Stress at Point C	IP
9		Minimum stress	9 <sup>1</sup>	Long. Stress at Point D	IP
10		Safety margin in tension	10 <sup>1</sup>	Long. Stress at Point E	IP
11		Safety margin in compression	11 <sup>1</sup>	Long. Stress at Point F	IP
		(Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)		(Item codes are given for end A. Addition of the quantity (K-1)10 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	
CBEAM (94) Nonlinear	2	External grid point ID		Not applicable	
	3	C (Character)			
	4	Long. Stress at point C			
	5	Equivalent stress			
	6	Total strain			
	7	Effective plastic strain			
	8	Effective creep strain			
		(Item codes 3 through 8 are repeated for points D, E, and F. Then the entire record (from 2 through N) is repeated for end B of the element.)			

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBEAM (239) Arbitrary Cross Section	2	Station Distance/Length			
	3	Max Axial Stress			
	4	Min Axial Stress			
	5	Max Shear Stress in xy			
	6	Min Shear Stress in xy			
	7	Max Shear Stress in xz			
	8	Min Shear Stress in xz			
	9	Max von Mises Stress		(Not supported)	
	CBEAM3 (184) Linear	2	External grid point ID	2	External grid point ID
3		Normal stress at point C	3	Normal stress at point C	RM
4		Normal stress at point D	4	Normal stress at point D	RM
5		Normal stress at point E	5	Normal stress at point E	RM
6		Normal stress at point F	6	Normal stress at point F	RM
7		Maximum stress	7	Normal stress at point C	IP
8		Minimum stress	8	Normal stress at point D	IP
9		Safety margin in tension	9	Normal stress at point E	IP
10		Safety margin in compression	10	Normal stress at point F	IP
11		Shear stress in y-direction at point C	11	Shear stress in y-direction at point C	RM
12		Shear stress in y-direction at point D	12	Shear stress in y-direction at point D	RM
13		Shear stress in y-direction at point E	13	Shear stress in y-direction at point E	RM
14		Shear stress in y-direction at point F	14	Shear stress in y-direction at point F	RM
15		Maximum shear stress in y-direction	15	Shear stress in y-direction at point C	IP
16		Minimum shear stress in y-direction	16	Shear stress in y-direction at point D	IP
17		(Not applicable)	17	Shear stress in y-direction at point E	IP
18		(Not applicable)	18	Shear stress in y-direction at point F	IP
19		Shear stress in z-direction at point C	19	Shear stress in z-direction at point C	RM

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	20	Shear stress in z-direction at point D	20	Shear stress in z-direction at point D	RM
	21	Shear stress in z-direction at point E	21	Shear stress in z-direction at point E	RM
	22	Shear stress in z-direction at point F	22	Shear stress in z-direction at point F	RM
	23	Maximum shear stress in z-direction	23	Shear stress in z-direction at point C	IP
	24	Minimum shear stress in z-direction	24	Shear stress in z-direction at point D	IP
	25	(Not applicable)	25	Shear stress in z-direction at point Z	IP
	26	(Not applicable)  (Item codes are given for end A. they are repeated for end B and mid-node C, respectively)	26	Shear stress in z-direction at point F  (Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	IP
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3	Circumferential angle	3	Circumferential angle	
	4 <sup>1</sup>	Long. Stress at Point C	4 <sup>1</sup>	Long. Stress at Point C	RM
	5 <sup>1</sup>	Long. Stress at Point D	5 <sup>1</sup>	Long. Stress at Point D	RM
	6 <sup>1</sup>	Long. Stress at Point E	6 <sup>1</sup>	Long. Stress at Point E	RM
	7 <sup>1</sup>	Long. Stress at Point F	7 <sup>1</sup>	Long. Stress at Point F	RM
	8	Maximum stress	8 <sup>1</sup>	Long. Stress at Point C	IP
	9	Minimum stress	9 <sup>1</sup>	Long. Stress at Point D	IP
	10	Safety margin in tension	10 <sup>1</sup>	Long. Stress at Point E	IP
	11	Safety margin in compression  (Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)	11 <sup>1</sup>	Long. Stress at Point F  (Item codes are given for end A. Item codes 12 through 21 point to the same information for end B.)	IP
CBUSH (102)	2 <sup>1</sup>	Translation-x	2 <sup>1</sup>	Translation-x	R
	3 <sup>1</sup>	Translation-y	3 <sup>1</sup>	Translation-y	R
	4 <sup>1</sup>	Translation-z	4 <sup>1</sup>	Translation-z	R
	5 <sup>1</sup>	Rotation-x	5 <sup>1</sup>	Rotation-x	R
	6 <sup>1</sup>	Rotation-y	6 <sup>1</sup>	Rotation-y	R
	7 <sup>1</sup>	Rotation-z	7 <sup>1</sup>	Rotation-z	R
			8 <sup>1</sup>	Translation-x	I

Table 7-1 Element Stress-Strain Item Codes (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
			9 <sup>1</sup>	Translation-y	I
			10 <sup>1</sup>	Translation-z	I
			11 <sup>1</sup>	Rotation-x	I
			12 <sup>1</sup>	Rotation-y	I
			13 <sup>1</sup>	Rotation-z	I
CBUSH1D (40)	1	Element ID			
	2	Axial force			
	3	Axial displacement			
	4	Axial velocity		Not applicable	
	5	Axial stress			
	6	Axial strain			
	7				
	8				

Table 7-2 Element Stress-Strain Item Codes Part 2

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CCONEAX (35)	2	Harmonic or point angle		Not applicable	
	3	Z1=Fiber Distance 1			
	4 <sup>1</sup>	Normal v at Z1			
	5 <sup>1</sup>	Normal u at Z1			
	6 <sup>1</sup>	Shear uv at Z1			
	7	Shear angle at Z1			
	8	Major principal at Z1			
	9	Minor principal at Z1			
	10	Maximum shear at Z1			
	11	Z2= Fiber Distance 2			
	12 <sup>1</sup>	Normal v at Z2			
	13 <sup>1</sup>	Normal u at Z2			
	14 <sup>1</sup>	Shear uv at Z2			
	15	Shear angle at Z2			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	16	Major principal at Z2			
	17	Minor principal at Z2			
	18	Maximum shear at Z2			
CDUM3 thru CDUM9 (55-61)	2 <sup>1</sup>	S1	2 <sup>1</sup>	S1	RM
	3 <sup>1</sup>	S2	3 <sup>1</sup>	S2	RM
	4 <sup>1</sup>	S3	4 <sup>1</sup>	S3	RM
	5 <sup>1</sup>	S4	5 <sup>1</sup>	S4	RM
	6 <sup>1</sup>	S5	6 <sup>1</sup>	S5	RM
	7 <sup>1</sup>	S6	7 <sup>1</sup>	S6	RM
	8 <sup>1</sup>	S7	8 <sup>1</sup>	S7	RM
CELAS1 (11)	2 <sup>1</sup>	Stress	2 <sup>1</sup>	Stress	RM
			3 <sup>1</sup>	Stress	IP
CELAS2 (12)	2 <sup>1</sup>	Stress	2 <sup>1</sup>	Stress	RM
			3 <sup>1</sup>	Stress	IP
CELAS3 (13)	2 <sup>1</sup>	Stress	2 <sup>1</sup>	Stress	RM
			3 <sup>1</sup>	Stress	IP
CGAP (86)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			
CHEXA (67) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6 <sup>1</sup>	Normal x	6 <sup>1</sup>	Normal x	RM
	7 <sup>1</sup>	Shear xy	7 <sup>1</sup>	Normal y	RM
	8	First principal	8 <sup>1</sup>	Normal z	RM
	9	First principal x cosine	9 <sup>1</sup>	Shear xy	RM



Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	10	Second principal x cosine	10 <sup>1</sup>	Shear yz	RM
	11	Third principal x cosine	11 <sup>1</sup>	Shear zx	RM
	12	Mean pressure	12 <sup>1</sup>	Normal x	IP
	13	von Mises or octahedral shear stress	13 <sup>1</sup>	Normal y	IP
	14 <sup>1</sup>	Normal y	14 <sup>1</sup>	Normal z	IP
	15 <sup>1</sup>	Shear yz	15 <sup>1</sup>	Shear xy	IP
	16	Second principal	16 <sup>1</sup>	Shear yz	IP
	17	First principal y cosine	17 <sup>1</sup>	Shear zx	IP
	18	Second principal y cosine	18-121	Items 5 through 17 repeated for 8 corners	
	19	Third principal y cosine			
	20 <sup>1</sup>	Normal z			
	21 <sup>1</sup>	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-193	Items 5 through 25 repeated for 8 corners			
CHEXA (93) Nonlinear	2	Stress coordinate system			
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Stress-X			
	7	Stress-Y			
	8	Stress-Z			
	9	Stress-XY			
	10	Stress-YZ			
	11	Stress-ZX			
	12	Equivalent stress			
	13	Effective plastic strain		Not applicable	
	14	Effective creep strain			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	15	Strain-X			
	16	Strain-Y			
	17	Strain-Z			
	18	Strain-XY			
	19	Strain-YZ			
	20	Strain-ZX			
	21-148	Items 5 through 20 Repeated for 8 corners			
CHEXAFD (202) Nonlinear Finite Deformation with 8 grid points	2	Grid/Gauss			
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Cauchy stress-YZ			
	9	Cauchy stress-ZX			
	10	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	11	Volumetric strain J-1			
	12	Logarithmic strain-X			
	13	Logarithmic strain-Y			
	14	Logarithmic strain-Z			
	15	Logarithmic strain-XY			
	16	Logarithmic strain-YZ			
	17	Logarithmic strain-ZX			
	18-122	Items 3 through 17 repeated for 7 Gauss/grid points			
CHEXAFD (207) Nonlinear Finite Deformation with 20 grid points	2-17	Same as CHEXAFD (202)			
	18-407	Items 3 through 17 repeated for 26 Gauss points		Not applicable	
CIFHEX (65) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	5	Shear Stress 1			
	6	Shear Stress 2			
	7	Normal Strain			
	8	Shear Strain 1			
	9	Shear Strain 2			
	10	Damage Value			
	11-74	Items 3 through 10 repeated for 6 Corners			
CIFPENT (66) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress 1			
	6	Shear Stress 2			
	7	Normal Strain			
	8	Shear Strain 1			
	9	Shear Strain 2			
	10	Damage Value			
		11-58	Items 3 through 10 repeated for 6 Corners		
CIFQDX (73) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress			
	6	Normal Strain			
	7	Shear Strain			
	8	Damage Value			
		9-32	Items 3 through 8 repeated for 4 Corners		
CIFQUAD (63) Nonlinear	2	Grid/Gauss			
	3	External Grid ID (0 = Center)			
	4	Normal Stress			
	5	Shear Stress			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	6	Normal Strain			
	7	Shear Strain			
	8	Damage Value			
	9-32	Items 3 through 8 repeated for 4 Corners			
CONROD (10) Linear	2 <sup>1</sup>	Axial stress	2 <sup>1</sup>	Axial stress	RM
	3	Axial safety margin	3 <sup>1</sup>	Axial stress	IP
	4 <sup>1</sup>	Torsional stress	4 <sup>1</sup>	Torsional stress	RM
	5	Torsional safety margin	5 <sup>1</sup>	Torsional stress	IP
CONROD (92) Nonlinear	2	Axial stress			
	3	Equivalent stress			
	4	Total strain			
	5	Effective plastic strain		Not applicable	
	6	Effective creep strain			
	7	Linear torsional stress			
CPENTA (68) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=center)	5	External grid ID (0=center)	
	6 <sup>1</sup>	Normal x	6 <sup>1</sup>	Normal x	RM
	7 <sup>1</sup>	Shear xy	7 <sup>1</sup>	Normal y	RM
	8	First principal	8 <sup>1</sup>	Normal z	RM
	9	First principal x cosine	91	Shear xy	RM
	10	Second principal x cosine	10 <sup>1</sup>	Shear yz	RM
	11	Third principal x cosine	11 <sup>1</sup>	Shear zx	RM
	12	Mean pressure	12 <sup>1</sup>	Normal x	IP
	13	von Mises or Octahedral shear stress	13 <sup>1</sup>	Normal y	IP
	14 <sup>1</sup>	Normal y	14 <sup>1</sup>	Normal z	IP
	15 <sup>1</sup>	Shear yz	15 <sup>1</sup>	Shear xy	IP
	16	Second principal	16 <sup>1</sup>	Shear yz	IP
	17	First principal y cosine	17 <sup>1</sup>	Shear zx	IP

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	18	Second principal y cosine	18-95	Items 5 through 17 repeated for 6 corners	
	19	Third principal y cosine			
	20 <sup>1</sup>	Normal z			
	21 <sup>1</sup>	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-151	Items 5 through 25 repeated for 6 corners			
CPENTA (91)	2	Stress coordinate system			
Nonlinear	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Normal x stress			
	7	Normal y stress			
	8	Normal z stress			
	9	Shear xy stress			
	10	Shear yz stress			
	11	Shear zx stress			
	12	Equivalent stress		Not applicable	
	13	Effective plastic strain			
	14	Effective creep strain			
	15	Normal x strain			
	16	Normal y strain			
	17	Normal z strain			
	18	Shear xy strain			
	19	Shear yz strain			
	20	Shear zx strain			
	21-116	Items 5 through 20 Repeated for 6 corners			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CPENTAFD (204)	2-17	Same as CHEXAFD (201)			
Nonlinear Finite Deformation with 6 grid points	18-92	Items 3 through 17 repeated for 5 Gauss points		Not applicable	
CPENTAFD (209)	2-17	Same as CHEXAFD (201)			
Nonlinear Finite Deformation with 15 grid points	18-317	Items 3 through 17 repeated for 20 Gauss points		Not applicable	

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CPYRAM (242) Linear	2	Stress coordinate system	2	Stress coordinate system	
	3	Coordinate Type (Character)	3	Coordinate Type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID (0=centre)	5	External grid ID (0=centre)	
	6	Normal x	6	Normal x	RM
	7	Shear xy	7	Normal y	RM
	8	First principal	8	Normal z	RM
	9	First principal x cosine	9	Shear xy	RM
	10	Second principal x cosine	10	Shear yz	RM
	11	Third principal x cosine	11	Shear zx	RM
	12	Mean pressure	12	Normal x	IP
	13	von Mises or octahedral shear stress	13	Normal y	IP
	14	Normal y	14	Normal z	IP
	15	Shear yz	15	Shear xy	IP
	16	Second principal	16	Shear yz	IP
	17	First principal y cosine	17	Shear zx	IP
	18	Second principal y cosine	18-82	Items 5 through 17 repeated for 5 corners	
	19	Third principal y cosine			
	20	Normal z			
	21	Shear zx			
	22	Third Principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-130	Items 5 through 25 repeated for 5 corners			

Table 7-2 Element Stress-Strain Item Codes Part 2 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CPYRAM (242) Non Linear	2	Stress coordinate system		Not Applicable	
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=centre)			
	6	Stress X			
	7	Stress Y			
	8	Stress Z			
	9	Stress XY			
	10	Stress YZ			
	11	Stress ZX			
	12	Equivalent Stress			
	13	Effective plastic strain			
	14	Effective creep strain			
	15	Strain X			
	16	Strain Y			
	17	Strain Z			
	18	Strain XY			
	19	Strain YZ			
	20	Strain ZX			
	21-100	Items 5 through 20 repeated for 5 corners			

Table 7-3 Element Stress-Strain Item Codes Part 3

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CQUAD4 (33) Linear	2	Z1=Fiber distance 1	2	Z1=Fiber distance 1	
	3 <sup>1</sup>	Normal x at Z1	3 <sup>1</sup>	Normal x at Z1	RM
	4 <sup>1</sup>	Normal y at Z1	4 <sup>1</sup>	Normal x at Z1	IP
	5 <sup>1</sup>	Shear xy at Z1	5 <sup>1</sup>	Normal y at Z1	RM
	6	Shear angle at Z1	6 <sup>1</sup>	Normal y at Z1	IP



Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	Major principal at Z1	7 <sup>1</sup>	Shear xy at Z1	RM
	8	Minor principal at Z1	8 <sup>1</sup>	Shear xy at Z1	IP
	9	von Mises or maximum shear at Z1	9	Z2=Fiber distance 2	
	10	Z2=Fiber distance 2	10 <sup>1</sup>	Normal x at Z2	RM
	11 <sup>1</sup>	Normal x at Z2	11 <sup>1</sup>	Normal x at Z2	IP
	12 <sup>1</sup>	Normal y at Z2	12 <sup>1</sup>	Normal y at Z2	RM
	13 <sup>1</sup>	Shear xy at Z2	13 <sup>1</sup>	Normal y at Z2	IP
	14	Shear angle at Z2	14	Shear xy at Z2	RM
	15	Major principal at Z2	15	Shear xy at Z2	IP
	16	Minor principal at Z2			
	17	von Mises or maximum shear at Z2			
CQUAD4 (90) Nonlinear	2	Z1=Fiber distance 1 (plane stress only)			
	3	Stress-X (at Z1, if plane stress)			
	4	Stress-Y (at Z1, if plane stress)			
	5	Stress-Z (plane strain only)			
	6	Stress-XY (at Z1, if plane stress)			
	7	Equivalent stress (at Z1, if plane stress)			
	8	Plastic strain (at Z1, if plane stress)			
	9	Creep strain (at Z1, if plane stress)		Not applicable	
	10	Strain-X (at Z1, if plane stress)			
	11	Strain-Y (at Z1, if plane stress)			
	12	Strain-Z (plane strain only)			
	13	Strain-XY (at Z1, if plane stress)			
	14-25	Items 2 through 13 repeated for fiber distance Z2 (plane stress only)			
	CQUAD4 <sup>2</sup> (95) Composite	2	Lamina Number	2	Lamina Number
3		Normal-1	3	Normal-1	RM
4		Normal-2	4	Normal-2	RM

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	5	Shear-12	5	Shear-12	RM
	6	Shear-1Z	6	Shear-1Z	RM
	7	Shear-2Z	7	Shear-2Z	RM
	8	Shear angle	8	Normal-1	IP
	9	Major principal	9	Normal-2	IP
	10	Minor principal	10	Shear-12	IP
	11	Maximum shear	11	Shear-1Z	IP
	12		12	Shear-2Z	IP
CQUAD4 (144) CORNER Output	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Z1-Fiber distance	4	Z1-Fiber distance	
	5	Normal x at Z1	5	Normal x at Z1	RM
	6	Normal y at Z1	6	Normal x at Z1	IP
	7	Shear xy at Z1	7	Normal y at Z1	RM
	8	Shear angle at Z1	8	Normal y at Z1	IP
	9	Major principal at Z1	9	Shear xy at Z1	RM
	10	Minor principal at Z1	10	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	11	Z2-Fiber distance	
	12	Z2-Fiber distance	12	Normal x at Z2	RM
	13	Normal x at Z2	13	Normal x at Z2	IP
	14	Normal y at Z2	14	Normal y at Z2	RM
	15	Shear xy at Z2	15	Normal y at Z2	IP
	16	Shear angle at Z2	16	Shear xy at Z2	RM
	17	Major principal at Z2	17	Shear xy at Z2	IP
	18	Minor principal at Z2	18	Grid 1	
	19	von Mises or maximum shear at Z2	19-32	Same as 4 through 17 for corner 1	
	20	Grid 1	33	Grid 2	
	21-36	Same as 4 through 19 for corner 1	34-47	Same as 4 through 17 for corner 2	
	37	Grid 2	48	Grid 3	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	38-53	Same as 4 through 19 for corner 2	49-62	Same as 4 through 17 for corner 3	
	54	Grid 3	63	Grid 4	
	55-70	Same as 4 through 19 for corner 3	64-77	Same as 4 through 17 for corner 4	
	71	Grid 4			
	72-87	Same as 4 through 19 for corner 4			
CQUAD8 (64)	5 <sup>1</sup>	Normal x at Z1	5 <sup>1</sup>	Normal x at Z1	RM
	6 <sup>1</sup>	Normal y at Z1	6 <sup>1</sup>	Normal x at Z1	IP
	7 <sup>1</sup>	Shear xy at Z1	7 <sup>1</sup>	Normal y at Z1	RM
	8	O Shear angle at Z1	8 <sup>1</sup>	Normal y at Z1	IP
	9	Major principal at Z1	9 <sup>1</sup>	Shear xy at Z1	RM
	10	Minor principal at Z1	10 <sup>1</sup>	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	12 <sup>1</sup>	Normal x at Z2	RM
	13 <sup>1</sup>	Normal x at Z2	13 <sup>1</sup>	Normal x at Z2	IP
	14 <sup>1</sup>	Normal y at Z2	14 <sup>1</sup>	Normal y at Z2	RM
	15 <sup>1</sup>	Shear xy at Z2	15 <sup>1</sup>	Normal y at Z2	IP
	16	Q Shear angle at Z2	16 <sup>1</sup>	Shear xy at Z2	RM
	17	Major principal at Z2	17 <sup>1</sup>	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	
	19	von Mises or maximum shear at Z2	35-47	Same as items 5 through 17 for corner 2	
	22-36	Same as items 5 through 19 for corner 1	50-62	Same as items 5 through 17 for corner 3	
	39-53	Same as items 5 through 19 for corner 2	65-77	Same as items 5 through 17 for corner 4	
	56-70	Same as items 5 through 19 for corner 3			
	73-87	Same as items 5 through 19 for corner 4			
CQUAD8 <sup>2</sup> (96) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CQUADFD (201) Nonlinear Finite Deformation with 4 grid points	2	Grid/Gauss			
	3	Grid/Gauss ID (0=center)			
	4	Cauchy stress-X			
	5	Cauchy stress-Y			
	6	Cauchy stress-Z			
	7	Cauchy stress-XY			
	8	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X			
	11	Logarithmic strain-Y			
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for 3 Gauss points			
CQUADFD (208) Nonlinear Finite Deformation with 8 or 9 grid points	2-13	Same as CQUADFD (201)			
	14-101	Items 3 through 13 repeated for 8 Gauss points		Not applicable	
CQUADR (82) Linear		Same as CQUAD4(144) CORNER output		Same as CQUAD4(144)	
CQUADR (172) Nonlinear		Same as CQUAD4(90)		Not applicable	
CQUADR (232) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CQUADX(18) Harmonic	1	(Elem ID)	1	(Elem ID)	
	2	Harmonic	2	Harmonic	
	3	CEN/grid	3	CEN/grid	
	4	Stress_xx	4	Stress_xx	RM
	5	Stress_yy	5	Stress_yy	RM
	6	Stress_tt	6	Stress_tt	RM
	7	Stress_xy	7	Stress_xy	RM
	8	Stress_yt	8	Stress_yt	RM
	9	Stress_tx	9	Stress_tx	RM
	10	VON MISES*	10	Stress_xx	IP
	11-18	1st corner	11	Stress_yy	IP
	19-26	2nd corner	12	Stress_tt	IP
	27-34	3rd corner	13	Stress_xy	IP
	35-42	4th corner	14	Stress_yt	IP
			15	Stress_tx	IP
			16-28	1st corner	
			29-41	2nd corner	
			42-54	3rd corner	
		55-67	4th corner		
CQUADXFD (214) Nonlinear Finite Deformation with 4 grid points	2	Grid/Gauss			
	3	Gauss ID			
	4	Cauchy stress-X (radial)			
	5	Cauchy stress-Y (axial)			
	6	Cauchy stress-Z (circumferential)			
	7	Cauchy stress-XY			
	8	Pressure $p = \frac{1}{3}(\sigma_x + \sigma_y + \sigma_z)$		Not applicable	
	9	Volumetric strain J-1			
	10	Logarithmic strain-X (radial)			
	11	Logarithmic strain-Y (axial)			
	12	Logarithmic strain-Z (circumferential)			

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	13	Logarithmic strain-XY			
	14-46	Items 3 through 13 repeated for remaining 3 Gauss points			
CQUADXFD (215)	2-13	Same as QUADXFD (214)		Not applicable	
Nonlinear Finite Deformation with 8 or 9 grid points	14-101	Items 3 through 13 repeated for remaining 8 Gauss points			
CROD (1) Linear		Same as CONROD(10)		Same as CONROD(10)	
CROD (89) Nonlinear		Same as CONROD(92)		Not applicable	
CSHEAR (4)	2	Maximum shear	2	Maximum shear	RM
	3 <sup>1</sup>	Average shear	3	Maximum shear	IP
	4	Safety margin	4 <sup>1</sup>	Average shear	RM
			5 <sup>1</sup>	Average shear	IP
CSLOT3 (50)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
			7	Radial centroid	IP
		8	Axialcentroid	IP	
		9	Tangential edge 1	IP	
		10	Tangential edge 2	IP	
		11	Tangential edge 3	IP	
CSLOT4 (51)	2	Radial centroid	2	Radial centroid	RM
	3	Axial centroid	3	Axial centroid	RM
	4	Tangential edge 1	4	Tangential edge 1	RM
	5	Tangential edge 2	5	Tangential edge 2	RM
	6	Tangential edge 3	6	Tangential edge 3	RM
	7	Tangential edge 4	7	Tangential edge 4	RM
			8	Radial centroid	IP
		9	Axial centroid	IP	
		10	Tangential edge 1	IP	

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
			11	Tangential edge 2	IP
			12	Tangential edge 3	IP
			13	Tangential edge 4	IP
CTETRA (39)	2	Stress coordinate system	2	Stress coordinate system	
Linear	3	Coordinate type (Character)	3	Coordinate type (Character)	
	4	Number of active points	4	Number of active points	
	5	External grid ID	5	External grid ID	
		(0=center)		(0=center)	
	6 <sup>1</sup>	Normal x	6 <sup>1</sup>	Normal x	RM
	7 <sup>1</sup>	Shear xy	7 <sup>1</sup>	Normal y	RM
	8	First principal	8 <sup>1</sup>	Normal z	RM
	9	First principal x cosine	9 <sup>1</sup>	Shear xy	RM
	10	Second principal x cosine	10 <sup>1</sup>	Shear yz	RM
	11	Third principal x cosine	11 <sup>1</sup>	Shear zx	RM
	12	Mean pressure	12 <sup>1</sup>	Normal x	IP
	13	von Mises or octahedral shear stress	13 <sup>1</sup>	Normal y	IP
	14 <sup>1</sup>	Normal y	14 <sup>1</sup>	Normal z	IP
	15 <sup>1</sup>	Shear yz	15 <sup>1</sup>	Shear xy	IP
	16	Second principal	16 <sup>1</sup>	Shear yz	IP
	17	First principal y cosine	17 <sup>1</sup>	Shear zx	IP
	18	Second principal y cosine	18-69	Items 5 through 17 repeated for four corners	
	19	Third principal y cosine			
	20 <sup>1</sup>	Normal z			
	21 <sup>1</sup>	Shear zx			
	22	Third principal			
	23	First principal z cosine			
	24	Second principal z cosine			
	25	Third principal z cosine			
	26-109	Items 5 through 25 repeated for four corners			

Table 7-3 Element Stress-Strain Item Codes Part 3 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTETRA (85) Nonlinear	2	Stress coordinate system			
	3	Grid/Gauss			
	4	Number of active points			
	5	External grid ID (0=center)			
	6	Stress-X			
	7	Stress-Y			
	8	Stress-Z			
	9	Stress-XY			
	10	Stress-YZ			
	11	Stress-ZX			
	12	Equivalent stress			
	13	Effective plastic strain		Not applicable	
	14	Effective creep strain			
	15	Strain-X			
	16	Strain-Y			
	17	Strain-Z			
	18	Strain-XY			
	19	Strain-YZ			
	20	Strain-ZX			
		21-84	Items 5 through 20 Repeated for four corners		
CTETRAFD (205) Nonlinear Finite Deformation with 4 grid points	2-17	Same as CHEXAFD (202)		Not applicable	
CTETRAFD (210) Nonlinear Finite Deformation with 10 grid points	2-17	Same as CHEXAFD (202)			
	18-77	Items 3 through 17 repeated for 4 Gauss points		Not applicable	



Table 7-4 Element Stress-Strain Item Codes Part 4

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTRIA3 (74)		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTRIA3 <sup>2</sup> (97) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIA3 (88) Nonlinear		Same as CQUAD4(90)		Not applicable	
CTRIA6 (75) Linear	5 <sup>1</sup>	Normal x at Z1	5 <sup>1</sup>	Normal x at Z1	RM
	6 <sup>1</sup>	Normal y at Z1	6 <sup>1</sup>	Normal x at Z1	IP
	7 <sup>1</sup>	Shear xy at Z1	7 <sup>1</sup>	Normal y at Z1	RM
	8	Q shear angle at Z1	8 <sup>1</sup>	Normal y at Z1	IP
	9	Major principal at Z1	9 <sup>1</sup>	Shear xy at Z1	RM
	10	Minor principal at Z1	10 <sup>1</sup>	Shear xy at Z1	IP
	11	von Mises or maximum shear at Z1	12 <sup>1</sup>	Normal x at Z2	RM
	13 <sup>1</sup>	Normal x at Z2	13 <sup>1</sup>	Normal x at Z2	IP
	14 <sup>1</sup>	Normal y at Z2	14 <sup>1</sup>	Normal y at Z2	RM
	15 <sup>1</sup>	Shear xy at Z2	15 <sup>1</sup>	Normal y at Z2	IP
	16	Q shear angle at Z2	16 <sup>1</sup>	Shear xy at Z2	RM
	17	Major principal at Z2	17 <sup>1</sup>	Shear xy at Z2	IP
	18	Minor principal at Z2	20-32	Same as items 5 through 17 for corner 1	
	19	von Mises or maximum shear at Z2	35-47	Same as items 5 through 17 for corner 2	
	22-36	Same as items 5 through 19 for corner 1	50-62	Same as items 5 through 17 for corner 3	
	39-53	Same as items 5 through 19 for corner 2			
56-70	Same as items 5 through 19 for corner 3				
CTRIA6 <sup>2</sup> (98)		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAFD (206) Nonlinear Deformation with 3 grid points	2-13	Same as CQUADFD(201)		Not applicable	

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CTRIAFD (211)	2-13	Same as CQUADFD (201)			
Nonlinear Finite Deformation with 6 grid points	14-35	Items 3 through 12 repeated for 2 Gauss points		Not applicable	
CTRIAR (70)		Same as CTRIA6(75)		Same as CTRIA6(75)	
Linear					
CTRIAR (173)		Same as CQUAD4(90)		Not applicable	
Nonlinear					
CTRIAR (233)		Same as CQUAD4(95)		Same as CQUAD4(95)	
Composite					
CTRIAX(17)	1	(Elem ID)	1	(Elem ID)	
Harmonic	2	Harmonic	2	Harmonic	
	3	CEN/grid	3	CEN/grid	
	4	Stress_xx	4	Stress_xx	RM
	5	Stress_yy	5	Stress_yy	RM
	6	Stress_tt	6	Stress_tt	RM
	7	Stress_xy	7	Stress_xy	RM
	8	Stress_yt	8	Stress_yt	RM
	9	Stress_tx	9	Stress_tx	RM
	10	VON MISES*	10	Stress_xx	IP
	11-18	1st corner	11	Stress_yy	IP
	19-26	2nd corner	12	Stress_tt	IP
	27-34	3rd corner	13	Stress_xy	IP
			14	Stress_yt	IP
			15	Stress_tx	IP
		16-28	1st corner		
		29-41	2nd corner		
		42-54	3rd corner		
CTRIAX6 (53)	3 <sup>1</sup>	Radial	3 <sup>1</sup>	Radial	RM
	4 <sup>1</sup>	Azimuthal	4 <sup>1</sup>	Radial	IP
	5 <sup>1</sup>	Axial	5 <sup>1</sup>	Azimuthal	RM
	6 <sup>1</sup>	Shear stress	6 <sup>1</sup>	Azimuthal	IP

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
	7	Maximum principal	7	Axial	RM
	8	Maximum shear	8	Axial	IP
	9	von Mises or octahedral	9	Shear	RM
			10	Shear	IP
	11-17	Same as items 3 through 9 for corner 1	12-19	Same as items 3 through 10 for corner 1	
	19-25	Same as Items 3 through 9 for corner 2	21-28	Same as items 3 through 10 for corner 2	
	27-33	Same as items 3 through 9 for corner 3	30-37	Same as items 3 through 10 for corner 3	
<b>CTRIAXFD (212)</b> Nonlinear Finite Deformation with 3 grid points	2-13	Same as CQUADXFD (214)		Not applicable	
<b>CTRIAXFD (213)</b> Nonlinear Finite Deformation with 6 grid points	2-13	Same as CQUADXFD (214)			
	14-35	Items 3 through 13 repeated for 2 Gauss points		Not applicable	
<b>CTUBE (3)</b> Linear		Same as CONROD (10)		Same as CONROD(10)	
<b>CTUBE (87)</b> Nonlinear		Same as CONROD(92)		Not applicable	

Table 7-4 Element Stress-Strain Item Codes Part 4 (continued)

Element Name (Code)	Real Stresses or Strains		Complex Stresses or Strains		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CWELDP (118) if PARTPAT or ELPAT	2	Axial	2	Axial	RM
	3	Maximum End A	3	Maximum End A	RM
	4	Minimum End A	4	Minimum End A	RM
	5	Maximum End B	5	Maximum End B	RM
CWELDC (117) if MSET=OFF and oldweld=yes	6	Minimum End B	6	Minimum End B	RM
	7	Maximum Shear	7	Maximum Shear	RM
	8	Bearing Stress (Not applicable to Strain)	8	Bearing Stress (Not applicable to Strain)	RM
CWELD (200) if MSET=ON or olddweld=no			9	Axial	IP
			10	Maximum End A	IP
			11	Minimum End A	IP
			12	Maximum End B	IP
			13	Minimum End B	IP
			14	Maximum Shear	IP
			15	Bearing Stress (Not applicable to Strain)	IP

## Element Force Item Codes

All items are element forces (or moments) unless otherwise indicated.

Table 7-5 Element Force Item Codes Part 1

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag./Phase
CBAR (34) Linear	2 <sup>1</sup>	Bending End A plane 1	2 <sup>1</sup>	Bending End A plane 1	RM
	3 <sup>1</sup>	Bending End A plane 2	3 <sup>1</sup>	Bending End A plane 2	RM
	4 <sup>1</sup>	Bending End B plane 1	4 <sup>1</sup>	Bending End B plane 1	RM
	5 <sup>1</sup>	Bending End B plane 2	5 <sup>1</sup>	Bending End B plane 2	RM
	6 <sup>1</sup>	Shear plane 1	6 <sup>1</sup>	Shear plane 1	RM
	7 <sup>1</sup>	Shear plane 2	7 <sup>1</sup>	Shear plane 2	RM
	8 <sup>1</sup>	Axial force	8 <sup>1</sup>	Axial force	RM
	9 <sup>1</sup>	Torque	9 <sup>1</sup>	Torque	RM
			10 <sup>1</sup>	Bending End A plane 1	IP

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			11 <sup>1</sup>	Bending End A plane 2	IP
			12 <sup>1</sup>	Bending End B plane 1	IP
			13 <sup>1</sup>	Bending End B plane 2	IP
			14 <sup>1</sup>	Shear plane 1	IP
			15 <sup>1</sup>	Shear plane 2	IP
			16 <sup>1</sup>	Axial force	IP
			17 <sup>1</sup>	Torque	IP
<b>CBAR (100)</b>	2	Station Distance/Length	2	Station Distance/Length	
Intermediate Stations	3	Bending Moment Plane 1	3	Bending Moment Plane 1	RM
	4	Bending Moment Plane 2	4	Bending Moment Plane 2	RM
	5	Shear Force Plane 1	5	Shear Force Plane 1	RM
	6	Shear Force Plane 2	6	Shear Force Plane 2	RM
	7	Axial	7	Axial	RM
	8	Torque	8	Torque	RM
		Item codes are given for end A. Addition of the quantity (K-1) * 8 to the item code points to the same information for other stations, where K is the station number. K=8 for end B and 2 through 7 for intermediate stations.	9	Bending Moment Plane 1	IP
			10	Bending Moment Plane 2	IP
			11	Shear Force Plane 1	IP
			12	Shear Force Plane 2	IP
			13	Axial	IP
			14	Torque	IP
					(Item codes above are given for End A. For codes 2 through 14 at intermediate stations add (K-1) * 13 where K is the station number, and for codes at End B, K+number of stations plus 1.)
<b>CBEAM (2)</b>	2	External grid point ID	2	External grid point ID	
Linear	3	Station distance/length	3	Station distance/length	
	4 <sup>1</sup>	Bending moment plane 1	4 <sup>1</sup>	Bending moment plane 1	RM
	5 <sup>1</sup>	Bending moment plane 2	5 <sup>1</sup>	Bending moment plane 2	RM
	6 <sup>1</sup>	Web shear plane 1	6 <sup>1</sup>	Web shear plane 1	RM
	7 <sup>1</sup>	Web shear plane 2	7 <sup>1</sup>	Web shear plane 2	RM
	8 <sup>1</sup>	Axial force	8 <sup>1</sup>	Axial force	RM
	9 <sup>1</sup>	Total torque	9 <sup>1</sup>	Total torque	RM

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	10 <sup>1</sup>	Warping torque	10 <sup>1</sup>	Warping torque	RM
		(Item codes are given for end A. Addition of the quantity (K-1) 9 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)	11 <sup>1</sup>	Bending moment plane 1	IP
			12 <sup>1</sup>	Bending moment plane 2	IP
			13 <sup>1</sup>	Web shear plane 1	IP
			14 <sup>1</sup>	Web shear plane 2	IP
			15 <sup>1</sup>	Axial force	IP
			16 <sup>1</sup>	Total torque	IP
			17 <sup>1</sup>	Warping torque	IP
					(Item codes are given for end A. Addition of the quantity (K-1) 16 to the item code points to the same information for other stations, where K is the station number. K=11 for end B and 2 through 10 for intermediate stations.)
CBEAM3 (184)	2	External grid point ID	2	External grid point ID	
Linear	3	Bending moment in y-direction	3	Bending moment in y-direction	RM
	4	Bending moment in z-direction	4	Bending moment in z-direction	RM
	5	Shear force in y-direction	5	Shear force in y-direction	RM
	6	Shear force in z-direction	6	Shear force in z-direction	RM
	7	Axial force	7	Axial force	RM
	8	Total torque	8	Total torque	RM
	9	Bi-shear force	9	Bi-shear force	RM
	10	Bi-moment	10	Bi-moment	RM
		(Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	11	Bending moment in y-direction	IP
			12	Bending moment in z-direction	IP
			13	Shear force in y-direction	IP
			14	Shear force in z-direction	IP
			15	Axial force	IP
			16	Total torque	IP
			17	Bi-shear force	IP

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			18	Bi-moment  (Item codes are given for end A. They are repeated for end B and mid-node C, respectively)	IP
CBEND (69)	2	External grid point ID	2	External grid point ID	
	3 <sup>1</sup>	Bending moment plane 1	3 <sup>1</sup>	Bending moment plane 1	RM
	4 <sup>1</sup>	Bending moment plane 2	4 <sup>1</sup>	Bending moment plane 2	RM
	5 <sup>1</sup>	Shear plane 1	5 <sup>1</sup>	Shear plane 1	RM
	6 <sup>1</sup>	Shear plane 2	6 <sup>1</sup>	Shear plane 2	RM
	7 <sup>1</sup>	Axial force	7 <sup>1</sup>	Axial force	RM
	8 <sup>1</sup>	Torque	8 <sup>1</sup>	Torque	RM
		(Item codes are given for end A. Item codes 9 through 15 point to the same information for end B.)	9 <sup>1</sup>	Bending moment plane 1	IP
		10 <sup>1</sup>	Bending moment plane 2	IP	
		11 <sup>1</sup>	Shear plane 1	IP	
		12 <sup>1</sup>	Shear plane 2	IP	
		13 <sup>1</sup>	Axial force	IP	
		14 <sup>1</sup>	Torque  (Item codes are given for end A. Item codes 15 through 27 point to the same information for end B.)	IP	
CBUSH (102)	2 <sup>1</sup>	Force-x	2 <sup>1</sup>	Force-x	RM
	3 <sup>1</sup>	Force-y	3 <sup>1</sup>	Force-y	RM
	4 <sup>1</sup>	Force-z	4 <sup>1</sup>	Force-z	RM
	5 <sup>1</sup>	Moment-x	5 <sup>1</sup>	Moment-x	RM
	6 <sup>1</sup>	Moment-y	6 <sup>1</sup>	Moment-y	RM
	7 <sup>1</sup>	Moment-z	7 <sup>1</sup>	Moment-z	RM
			8 <sup>1</sup>	Force-x	IP
		9 <sup>1</sup>	Force-y	IP	
		10 <sup>1</sup>	Force-z	IP	
		11 <sup>1</sup>	Moment-x	IP	
		12 <sup>1</sup>	Moment-y	IP	
		13 <sup>1</sup>	Moment-z	IP	
CCONEAX (35)	2	Harmonic or point angle		Not applicable	
	3	Bending moment v			

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	4	Bending moment u			
	5	Twist moment			
	6	Shear v			
	7	Shear u			
CDAMP1 (20)		Same as CELAS1		Same as CELAS1	
CDAMP2 (21)		Same as CELAS1		Same as CELAS1	
CDAMP3 (22)		Same as CELAS1		Same as CELAS1	
CDAMP4 (23)		Same as CELAS1		Same as CELAS1	
CDUM3 thru CDUM9 (55 - 61)	2 <sup>1</sup>	F1	2 <sup>1</sup>	F1	RM
	3 <sup>1</sup>	F2	3 <sup>1</sup>	F2	RM
	4 <sup>1</sup>	F3	4 <sup>1</sup>	F3	RM
	5 <sup>1</sup>	F4	5 <sup>1</sup>	F4	RM
	6 <sup>1</sup>	F5	6 <sup>1</sup>	F5	RM
	7 <sup>1</sup>	F6	7 <sup>1</sup>	F6	RM
	8 <sup>1</sup>	F7	8 <sup>1</sup>	F7	RM
	9 <sup>1</sup>	F8	9 <sup>1</sup>	F8	RM
	10 <sup>1</sup>	F9	10 <sup>1</sup>	F9	RM
				11 <sup>1</sup>	F1
			12 <sup>1</sup>	F2	IP
			13 <sup>1</sup>	F3	IP
			14 <sup>1</sup>	F4	IP
			15 <sup>1</sup>	F5	IP
			16 <sup>1</sup>	F6	IP
			17 <sup>1</sup>	F7	IP
			18 <sup>1</sup>	F8	IP
			19 <sup>1</sup>	F9	IP
CELAS1 (11)	2 <sup>1</sup>	Force	2 <sup>1</sup>	Force	RM
	3	Force	3	Force	IP
CELAS2 (12)		Same as CELAS1		Same as CELAS1	
CELAS3 (13)		Same as CELAS1		Same as CELAS1	
CELAS4 (14)		Same as CELAS1		Same as CELAS1	



Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CGAP (38)	2	Normal x		Not applicable	
	3	Shear y			
	4	Shear z			
	5	Axial u			
	6	Shear v			
	7	Shear w			
	8	Slip v			
	9	Slip w			
	CONROD (10)	2 <sup>1</sup>	Axial force	2 <sup>1</sup>	Axial force
3 <sup>1</sup>		Torque	3 <sup>1</sup>	Axial force	IP
			4 <sup>1</sup>	Torque	RM
			5 <sup>1</sup>	Torque	IP
CQUAD4 (33) Linear	2 <sup>1</sup>	Membrane force x	2 <sup>1</sup>	Membrane force x	RM
	3 <sup>1</sup>	Membrane force y	3 <sup>1</sup>	Membrane force y	RM
	4 <sup>1</sup>	Membrane force xy	4 <sup>1</sup>	Membrane force xy	RM
	5 <sup>1</sup>	Bending moment x	5 <sup>1</sup>	Bending moment x	RM
	6 <sup>1</sup>	Bending moment y	6 <sup>1</sup>	Bending moment y	RM
	7 <sup>1</sup>	Bending moment xy	7 <sup>1</sup>	Bending moment xy	RM
	8 <sup>1</sup>	Shear x	8 <sup>1</sup>	Shear x	RM
	9 <sup>1</sup>	Shear y	9 <sup>1</sup>	Shear y	RM
			10 <sup>1</sup>	Membrane force x	IP
			11 <sup>1</sup>	Membrane force y	IP
			12 <sup>1</sup>	Membrane force xy	IP
			13 <sup>1</sup>	Bending moment x	IP
			14 <sup>1</sup>	Bending moment y	IP
			15 <sup>1</sup>	Bending moment xy	IP
			16 <sup>1</sup>	Shear x	IP
			17 <sup>1</sup>	Shear y	IP
	CQUAD4 (95) Composite	2-3	Theory or blank		Not applicable
4		Lamina number			
5		FP (failure index) /SP (strength ratio) for direct stresses			

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	6	Failure mode for Maximum strain theory			
	7	FB (failure index) /SB (strength ratio) or -1 for interlaminar shear-stress			
	8	MAX of FP, FB or -1 or MIN of SP, SB or -1			
	9	Failure flag			
CQUAD4 (144) Corner Output	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4	Membrane x	4	Membrane x	RM
	5	Membrane y	5	Membrane y	RM
	6	Membrane xy	6	Membrane xy	RM
	7	Bending x	7	Bending x	RM
	8	Bending y	8	Bending y	RM
	9	Bending xy	9	Bending xy	RM
	10	Shear x	10	Shear x	RM
	11	Shear y	11	Shear y	RM
	12	Grid 1	12	Membrane x	IP
	13-20	Same as 4 through 11 for corner 1	13	Membrane y	IP
	21	Grid 2	14	Membrane xy	IP
	22-29	Same as 4 through 11 for corner 2	15	Bending x	IP
	30	Grid 3	16	Bending y	IP
	31-38	Same as 4 through 11 for corner 3	17	Bending xy	IP
	39	Grid 4	18	Shear x	IP
	40-47	Same as 4 through 11 for corner 4	19	Shear y	IP
			20	Grid 1	
			21-36	Same as 4 through 19 for corner 1	
			37	Grid 2	

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			38-53	Same as 4 through 19 for corner 2	
			54	Grid 3	
			55-70	Same as 4 through 19 for corner 3	
			71	Grid 4	
			71-87	Same as 4 through 19 for corner 4	
CQUAD8 (64) Linear	1	EID	1	EID	
	2	CEN/	2	CEN/	
	3	4	3	4	
	4 <sup>1</sup>	Membrane force x	4 <sup>1</sup>	Membrane force x	RM
	5 <sup>1</sup>	Membrane force y	5 <sup>1</sup>	Membrane force y	RM
	6 <sup>1</sup>	Membrane force xy	6 <sup>1</sup>	Membrane force xy	RM
	7 <sup>1</sup>	Bending moment x	7 <sup>1</sup>	Bending moment x	RM
	8 <sup>1</sup>	Bending moment y	8 <sup>1</sup>	Bending moment y	RM
	9 <sup>1</sup>	Bending moment xy	9 <sup>1</sup>	Bending moment xy	RM
	10 <sup>1</sup>	Shear x	10 <sup>1</sup>	Shear x	RM
	11 <sup>1</sup>	Shear y	11 <sup>1</sup>	Shear y	RM
	12	Grid1	12 <sup>1</sup>	Membrane force x	IP
	13-20	Same as items 4 through 11 for corner 1	13 <sup>1</sup>	Membrane force y	IP
	21	Grid2	14 <sup>1</sup>	Membrane force xy	IP
	22-29	Same as items 4 through 11 for corner 2	15 <sup>1</sup>	Bending moment x	IP
	30	Grid3	16 <sup>1</sup>	Bending moment y	IP
	31-38	Same as items 4 through 11 for corner 3	17 <sup>1</sup>	Bending moment xy	IP
39	Grid4	18 <sup>1</sup>	Shear x	IP	
40-47	Same as items 4 through 11 for corner 4	19 <sup>1</sup>	Shear y	IP	
			20	Grid1	
			21-36	Same as items 4 through 19 for corner 1	
37			37	Grid2	

Table 7-5 Element Force Item Codes Part 1 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
			38-53	Same as items 4 through 19 for corner 2	
			54	Grid3	
			55-70	Same as items 4 through 19 for corner 3	
			71	Grid4	
			72-87	Same as items 4 through 19 for corner 4	
CQUAD8 <sup>2</sup> (96) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CQUADR (82)		Same as CQUAD8(64)		Same as CQUAD8(64)	
CQUADR (235) Center		Same as CQUAD4(33)		Same as CQUAD4(33)	

Table 7-6 Element Force Item Codes Part 2

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CROD (1)		Same as CONROD(10)		Same as CONROD(10)	
CSHEAR (4)	2 <sup>1</sup>	Force 4 to 1	2 <sup>1</sup>	Force 4 to 1	RM
	3 <sup>1</sup>	Force 2 to 1	3 <sup>1</sup>	Force 2 to 1	RM
	4 <sup>1</sup>	Force 1 to 2	4 <sup>1</sup>	Force 1 to 2	RM
	5 <sup>1</sup>	Force 3 to 2	5 <sup>1</sup>	Force 3 to 2	RM
	6 <sup>1</sup>	Force 2 to 3	6 <sup>1</sup>	Force 2 to 3	RM
	7 <sup>1</sup>	Force 4 to 3	7 <sup>1</sup>	Force 4 to 3	RM
	8 <sup>1</sup>	Force 3 to 4	8 <sup>1</sup>	Force 3 to 4	RM
	9 <sup>1</sup>	Force 1 to 4	9 <sup>1</sup>	Force 1 to 4	RM
	10 <sup>1</sup>	Kick force on 1	10 <sup>1</sup>	Force 4 to 1	IP
	11 <sup>1</sup>	Shear 12	11 <sup>1</sup>	Force 2 to 1	IP
	12 <sup>1</sup>	Kick force on 2	12 <sup>1</sup>	Force 1 to 2	IP
	13 <sup>1</sup>	Shear 23	13 <sup>1</sup>	Force 3 to 2	IP
	14 <sup>1</sup>	Kick force on 3	14 <sup>1</sup>	Force 2 to 3	IP

Table 7-6 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	15 <sup>1</sup>	Shear 34	15 <sup>1</sup>	Force 4 to 3	IP
	16 <sup>1</sup>	Kick force on 4	16 <sup>1</sup>	Force 3 to 4	IP
	17 <sup>1</sup>	Shear 41	17 <sup>1</sup>	Force 1 to 4	IP
			18 <sup>1</sup>	Kick force on 1	RM
			19 <sup>1</sup>	Shear 12	RM
			20 <sup>1</sup>	Kick force on 2	RM
			21 <sup>1</sup>	Shear 23	RM
			22 <sup>1</sup>	Kick force on 3	RM
			23 <sup>1</sup>	Shear 34	RM
			24 <sup>1</sup>	Kick force on 4	RM
			25 <sup>1</sup>	Shear 41	RM
			26 <sup>1</sup>	Kick force on 1	IP
			27 <sup>1</sup>	Shear 12	IP
			28 <sup>1</sup>	Kick force on 2	IP
			29 <sup>1</sup>	Shear 23	IP
			30 <sup>1</sup>	Kick force on 3	IP
			31 <sup>1</sup>	Shear 34	IP
			32 <sup>1</sup>	Kick force on 4	IP
			33 <sup>1</sup>	Shear 41	IP
<b>CTRIA3 (74)</b> Linear		Same as CQUAD4(33)		Same as CQUAD4(33)	
<b>CTRIA3<sup>2</sup> (97)</b> Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
<b>CTRIA6 (75)</b> Linear	4 <sup>1</sup>	Membrane force x	4 <sup>1</sup>	Membrane force x	RM
	5 <sup>1</sup>	Membrane force y	5 <sup>1</sup>	Membrane force y	RM
	6 <sup>1</sup>	Membrane force xy	6 <sup>1</sup>	Membrane force xy	RM
	7 <sup>1</sup>	Bending moment x	7 <sup>1</sup>	Bending moment x	RM
	8 <sup>1</sup>	Bending moment y	8 <sup>1</sup>	Bending moment y	RM
	9 <sup>1</sup>	Bending moment xy	9 <sup>1</sup>	Bending moment xy	RM
	10 <sup>1</sup>	Shear x	10 <sup>1</sup>	Shear x	RM
	11 <sup>1</sup>	Shear y	11 <sup>1</sup>	Shear y	RM
	13-20	Same as items 4 through 11 for corner 1	12 <sup>1</sup>	Membrane force x	IP

Table 7-6 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
	22-29	Same as items 4 through 11 for corner 2	13 <sup>1</sup>	Membrane force y	IP
	31-38	Same as items 4 through 11 for corner 3	14 <sup>1</sup>	Membrane force xy	IP
			15 <sup>1</sup>	Bending moment x	IP
			16 <sup>1</sup>	Bending moment y	IP
			17 <sup>1</sup>	Bending moment xy	IP
			18 <sup>1</sup>	Shear x	IP
			19 <sup>1</sup>	Shear y	IP
			21-36	Same as items 4 through 19 for corner 1	
			38-53	Same as items 4 through 19 for corner 2	
			55-70	Same as items 4 through 19 for corner 3	
CTRIA6 <sup>2</sup> (98) Composite		Same as CQUAD4(95)		Same as CQUAD4(95)	
CTRIAR (70)		Same as CTRIA6(75)		Same as CTRIA6(75)	
CTRIAR (236) Center		Same as CQUAD4(33)		Same as CQUAD4(33)	
CTUBE (3)		Same as CONROD(10)		Same as CONROD(10)	
CVISC (24)		Not applicable		Same as CONROD(10)	
CWELDP (118) if PARTPAT or ELPAT	2	mz bending end A plane 1	2	mz bending end A plane 1	RM
	3	my bending end A plane 2	3	my bending end A plane 2	RM
	4	mz bending end B plane 1	4	mz bending end B plane 1	RM
	5	my bending end B plane 2	5	my bending end B plane 2	RM
	6	fy shear force plane 1	6	fy shear force plane 1	RM
	7	fz shear force plane 2	7	fz shear force plane 2	RM
CWELDC (117) if MSET = OFF	8	fx axial force	8	fx axial force	RM
	9	mx torque	9	mx torque	RM
			10	mz bending end A plane 1	IP
		11	my bending end A plane 2	IP	

Table 7-6 Element Force Item Codes Part 2 (continued)

Element Name Code	Real Element Forces		Complex Element Forces		
	Item Code	Item	Item Code	Item	Real/Mag. or Imag/Phase
CWELD (200) if MSET = ON			12	mz bending end B plane 1	IP
			13	my bending end B plane 2	IP
			14	fy shear force plane 1	IP
			15	fz shear force plane 2	IP
			16	fx axial force	IP
			17	mx torque	IP

## Fluid Virtual Mass Pressure Item Codes

Table 7-7 Fluid Virtual Mass Pressure Item Codes

Element Name	Real Fluid Pressure		Complex Fluid Pressure		
	Code	Item	Code	Item	Real/Mag. or Imag./Phase
Plate	2	Fluid pressure	2	Pressure	RM
Family			3	Pressure	IP

Table 7-8 Heat Transfer Item Codes (Curve type is FLUX.)

Element Name (Code)	Code	Item
Heat Transfer Elements	2	Element type
	3 <sup>1</sup>	
	4 <sup>1</sup>	x gradient
	5 <sup>1</sup>	y gradient
	6 <sup>1</sup>	z gradient
	7	x flux
	8	y flux
	9	z flux
	CHBDYE	4
(107)	5	Free convection
	6	Forced convection
	7	Radiation

Table 7-8 Heat Transfer Item Codes (Curve type is FLUX.)

Element Name (Code)	Code	Item
	8	Total
CHBDYG	Same as	Same as CHBDYE
(108)	CHBDYE	
CHBDYP	Same as	Same as CHBDYE
(109)	CHBDYE	

## 2D Slideline and 3D Surface Contact Item Codes

Table 7-9 Contact Item Codes

Element Name (Code)	Real Element Data		
	Item Code	Slideline Item	3D Surface Item (SOL 600)
CSLIFID (116)	1	SECNDRY grid point	GRID ID
	2	Contact region identification number	(39) Contact Touched Body
	3	Primary grid 1	
	4	Primary grid 2	
	5	Surface coordinate	(38) Contact Status
	6	Normal force	(35) Normal Force
	7	Shear force	(37) Friction Force
	8	Normal stress	(34) Normal Stress
	9	Shear stress	(36) Friction Stress
	10	Normal gap	
	11	Slip	
	12	Slip ratio (Shear force/u*normal force)	
13-14	Slip code (Character)		

1. Numbers in parenthesis refer to MARCOUT nodal post codes.
2. 3D Surface contact is available in SOL 600 only.



Table 7-10 Contact Item Code for Nodal Data (SOL 400 only)

Element Code	Item	Real Nodal Data 3D Surface Item (SOL 400 only)
203	1	Grid point ID
	2	First touched body
	3	Second touched body
	4	Third touched body
	5	Contact status (Remark 7)
	6	Normal contact force magnitude
	7	Friction contact force magnitude
	8	Normal stress
	9	Friction stress 1
	10	Friction stress 2
	11	Contact normal force component $F_{nx}$
	12	Contact normal force component $F_{ny}$
	13	Contact normal force component $F_{nz}$
	14	Contact friction force component $F_{fx}$
	15	Contact friction force component $F_{fy}$
	16	Contact friction force component $F_{fz}$

Table 7-11 Contact Item Code for Body Data (SOL 400 only)

Element Code	Item	Rigid Body Data 3D Surface Item (SOL 400 Only)
203	1	BCBODY ID
	2-17	$[XMAT]_{4 \times 4}$
	18	Contact force component $RF_x$
	19	Contact force component $RF_y$
	20	Contact force component $RF_z$
	21	Contact moment component $RM_x$
	22	Contact moment component $RM_y$
	23	Contact moment component $RM_z$

**Remarks:**

1.  $F_{nx}$ ,  $F_{ny}$ ,  $F_{nz}$  are normal force components in the global coordinate system.
2.  $F_{fx}$ ,  $F_{fy}$ ,  $F_{fz}$  are friction force components in the global coordinate system.
3. Normal force magnitude is square root of sum of squares of  $F_{nx}$ ,  $F_{ny}$ ,  $F_{nz}$ .
4. Friction force magnitude is square root of sum of squares of  $F_{fx}$ ,  $F_{fy}$ ,  $F_{fz}$  with sign of largest absolute of these values.
5. Contact touched body applies to SECNDRY grids (touching node) only.
6. A secondary grid may not contact more than 3 bodies.
7. Only secondary grids have non-zero contact status or touched bodies.
8.  $RF_x$ ,  $RF_y$ ,  $RF_z$ ,  $RM_x$ ,  $RM_y$ ,  $RM_z$  are the contact force and moment in the global coordinate system that applied on the centroid of the corresponding rigid body.
9. [XMAT] is the translation matrix to compute the new location of the rigid body.
10. These contact outputs, both [Table 7-10](#) and [Table 7-11](#) are saved in datablock OFCON3D.

## Element Strain Energy Item Codes

Table 7-12 Element Strain Energy Item Codes

Element Name	Real Element Data	
	Item Code	Item
Element groups A and B	2	Element strain energy
Element groups A and B	3	Percent of total energy
Element group A	4	Element strain energy density

**Remark:**

1. Element group A includes elements of CBAR, CBEAM, CBEND, CONROD, CHEXA, CPENTA, CPYRAM, CQUAD4, CQUADR, CROD, CSHEAR, CTETRA, CTRIA3, CTRIA6, CTRIAR, and CTUBE. Element group B includes elements of CELAS1, CELAS2, CELAS3, AND CGAP.

## Fatigue Item Codes

Note that the codes of the [Table 7-13](#) through [Table 7-16](#) are limited to those that can be referenced on a DRESP1 entry with RTYPE= FATGUE.

Table 7-13 Fatigue Item Codes for LOC=ELEM on FTGPARAM

Element Name	Item Code	Real Element Data Item
Element Group A and Element Group B (for Z1 layer - bottom)	4 (-4)	Fatigue life in <i>Repeats</i> of the loading sequence See remark 4.
	5 (-5)	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6 (-6)	Fatigue life in user defined fatigue equivalent units
	7 (-7)	Log of fatigue life in user defined fatigue equivalent units
	8 (-8)	Fatigue damage
	9 (-9)	Log of fatigue damage
	12 (-12)	Scale factor from Factor of Safety analysis
Element Group B (for Z2 layer - top)	18	Fatigue life in <i>Repeats</i> of the loading sequence
	19	Log of fatigue life in <i>Repeats</i> of the loading sequence
	20	Fatigue life in user defined fatigue equivalent units
	21	Log of fatigue life in user defined fatigue equivalent units
	22	Fatigue damage
	23	Log of fatigue damage
	26	Scale factor from Factor of Safety analysis

### Remark:

1. Element Group A consists of elements: CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CSHEAR
2. Element Group B consists of elements: CQUAD4, CQUAD8, CQUADR, CTRIA6, CTRIAR
3. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARAM use the same item codes in [Table 7-13](#).
4. For Element Group B (shell elements), if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular set of elements or property sets.

Note: SOL 200 only supports NODE and not NODA on the FTGPARM entry

Table 7-14 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM

Element Name	Item Codes	Real Element Data Item
CTETRA	4,16,28,40	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42	Life in user defined fatigue equivalent units
	7,19,31,43	Log of life in user defined equivalent units
	8,20,32,44	Fatigue damage
	9,21,33,45	Log of fatigue damage
	12,24,36,48	Scale factor from Factor of Safety analysis
CQUAD4 CQUAD8 CQUADR (for Z1 layer - bottom)	4,33,62,91 (-4,-33,-62,-91)	Life in <i>Repeats</i> of the loading sequence See remark 8.
	5,34,63,92 (-5,-34,-63,-92)	Log of life in <i>Repeats</i> of the loading sequence
	6,35,64,93 (-6,-35,-64,-93)	Life in user defined fatigue equivalent units
	7, 36, 65, 94 (-7,-36,-65, -94)	Log of life in user defined equivalent units
	8, 37, 66, 95 (-8,-37,-66,-95)	Fatigue damage
	9, 38, 67, 96 (-9,-38,-67-96)	Log of fatigue damage
	12, 41, 70, 99 (-12,-41,-70,-99)	Scale factor from Factor of Safety analysis
	(for Z2 layer - top)	
	18, 47, 76, 105	Life in <i>Repeats</i> of the loading sequence
	19, 48, 77, 106	Log of life in <i>Repeats</i> of the loading sequence
	20, 49, 78, 107	Life in user defined fatigue equivalent units
	21, 50, 79, 108	Log of life in user defined equivalent units
	22, 51, 80, 109	Fatigue damage
	23, 52, 81, 110	Log of fatigue damage
	26, 55, 84, 113	Scale factor from Factor of Safety analysis

Table 7-14 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

Element Name	Item Codes	Real Element Data Item
CTRIA6 CTRIAR (for Z1 layer - bottom)	4, 33, 62 (-4, -33, -62)	Life in <i>Repeats</i> of the loading sequence See remark 8.
	5, 34, 63 (-5, -34, -63)	Log of life in <i>Repeats</i> of the loading sequence
	6, 35, 64 (-6, -35, -64)	Life in user defined fatigue equivalent units
	7, 36, 6 (-7, -36, -6)	Log of life in user defined equivalent units
	8, 37, 66 (-8, -37, -66)	Fatigue damage
	9, 38, 67 (-9, -38, -67)	Log of fatigue damage
	12, 41, 70 (-12, -41, -70)	Scale factor from Factor of Safety analysis
	(for Z2 layer - top)	18, 47, 76
19, 48, 77		Log of life in <i>Repeats</i> of the loading sequence
20, 50, 79		Life in user defined fatigue equivalent units
22, 51, 80		Log of life in user defined equivalent units
23, 52, 81		Fatigue damage
26, 55, 84		Log of fatigue damage
		Scale factor from Factor of Safety analysis
CPENTA	4,16,28,40,52,64	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53,65	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54,66	Life in user defined fatigue equivalent units
	7,19,31,43,55,67	Log of life in user defined equivalent units
	8,20,32,44,56,68	Fatigue damage
	9,21,33,45,57,69	Log of fatigue damage
	12,24,36,48,60,72	Scale factor from Factor of Safety analysis

Table 7-14 Fatigue Item Codes for LOC=NODE or NODA on FTGPARM (continued)

Element Name	Item Codes	Real Element Data Item
CPYRAM	4,16,28,40,52	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54	Life in user defined fatigue equivalent units
	7,19,31,43,55	Log of life in user defined equivalent units
	8,20,32,44,56	Fatigue damage
	9,21,33,45,57	Log of fatigue damage
	12,24,36,48,60	Scale factor from Factor of Safety analysis
CHEXA	4,16,28,40,52,64,76,88	Life in <i>Repeats</i> of the loading sequence
	5,17,29,41,53,65,77,89	Log of life in <i>Repeats</i> of the loading sequence
	6,18,30,42,54,66,78,90	Life in user defined fatigue equivalent units
	7,19,31,43,55,67,79,91	Log of life in user defined equivalent units
	8,20,32,44,56,68,80,92	Fatigue damage
	9,21,33,45,57,69,81,93	Log of fatigue damage
	12,24,36,48,60,72,84,96	Scale factor from Factor of Safety analysis

**Remarks:**

1. CTRIA6 and CTRIAR item codes are listed for Grids 1-3, respectively.
2. CQUAD4, CQUAD8, CQUADR, CTETRA item codes are listed for Grids 1-4, respectively.
3. CPYRAM item codes are listed for Grids 1-5, respectively.
4. CPENTA item codes are listed for Grids 1-6, respectively.
5. CHEXA item codes are listed for Grids 1-8, respectively.
6. For CTRIA3 and CSHEAR elements, LOC=ELEM and LOC=NODE or NODA options on FTGPARM use the same item codes in [Table 7-13](#).
7. If it is desired to use a life response (or other item) for all nodes (using LOC=NODE or NODA on FTGPARM), all item codes for the particular item are required, e.g., 4, 16, 27, and 38 for CTETRA (one DRESP1 entry for each item code).
8. For shell elements with bottom (Z1) and top (Z2) layers, if the item code is negative (e.g., -4) for layer Z1, this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for layer Z2. This is a convenient method to include responses from both top and bottom by defining only a single DRESP1 entry for any particular response for a set of elements or property sets. So as an example, to specify the bottom and top response (Life in *Repeats*) for all the grids of the specified element or property set for a CQUAD4 element, use items codes: -4,-27,-50,-73
9. These item codes are used for LOC=NODE or NODA and all other element nodal values of RECOVER on the FTGPARM entry, namely, "SGAGE", "CORNER", "BILIN", and "CUBIC" also.

Table 7-15 Item Codes for Fatigue Analysis of Spot Welds

Element Name	Item Code	Real Element Data Item
CBAR CBEAM CWELD CHEXA (For Top Sheet at Angle = 0.0 degrees)	4 (-4) (-1004)	Fatigue life in <i>Repeats</i> of the loading sequence See remarks 2. and 3.
	5 (-5)(-1005)	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6 (-6)(-1006)	Fatigue life in user defined fatigue equivalent units
	7 (-7)(-1007)	Log of fatigue life in user defined fatigue equivalent units
	8 (-8)(-1008)	Fatigue damage
	9 (-9)(-1009)	Log of fatigue damage
	13 (-13)(-1013)	Scale factor from Factor of Safety analysis
CBAR CBEAM CWELD CHEXA (For Weld Nugget at Angle = 0.0 degrees)	4+12*NANGLE	Fatigue life in <i>Repeats</i> of the loading sequence
	5+12*NANGLE	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6+12*NANGLE	Fatigue life in user defined fatigue equivalent units
	7+12*NANGLE	Log of fatigue life in user defined fatigue equivalent units
	8+12*NANGLE	Fatigue damage
	9+12*NANGLE	Log of fatigue damage
	13+12*NANGLE	Scale factor from Factor of Safety analysis
CBAR CBEAM CWELD CHEXA (For Bottom Sheet at Angle = 0.0 degrees)	4+24*NANGLE	Fatigue life in <i>Repeats</i> of the loading sequence
	5+24*NANGLE	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6+24*NANGLE	Fatigue life in user defined fatigue equivalent units
	7+24*NANGLE	Log of fatigue life in user defined fatigue equivalent units
	8+24*NANGLE	Fatigue damage
	9+24*NANGLE	Log of fatigue damage
	13+24*NANGLE	Scale factor from Factor of Safety analysis

**Remarks:**

- Spot weld fatigue item code is related to NANGLE, the number of calculation angles around the weld circumference as defined on the FTGPARM entry. Item codes listed above are for the first angle at zero (0) degrees. For each subsequent angle, the item code should be increased by 12. For example, item code =16 is for “Fatigue life in *Repeats* of the loading sequence” at the Top Sheet for the second angle.

2. If item codes 4-9 or 13 are negative (e.g., -4), this flags Nastran to automatically (and internally) create a DRESP1 with the corresponding item code for the nugget and bottom sheet as well and for all specified angles. This is a convenient method to include responses from all locations and angles by defining only a single DRESP1 entry for any particular response for a set of spot weld elements or property sets. So as an example, to specify the response (Life in *Repeats*) for all the locations and angles, use a single DRESP1 entry with item code: -4.
3. In conjunction with the above remark, if the item code is negative and multiplied by 1000 (e.g., -1004), then the nugget responses are not generated. Only the top and bottom sheet responses are internally generated.

Table 7-16 Item Codes for Fatigue Analysis of Seam Welds

Element Name	Item Code	Real Element Data Item
CQUAD4 CQUAD8 CQUADR	4, 19, 34, 45	Fatigue life in <i>Repeats</i> of the loading sequence
	5, 20, 35, 46	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6, 21, 36, 47	Fatigue life in user defined fatigue equivalent units
	7, 22, 37, 48	Log of fatigue life in user defined fatigue equivalent units
	8, 23, 38, 49	Fatigue damage
	9, 24, 39, 50	Log of fatigue damage
	12, 27, 42, 53	Scale factor from Factor of Safety analysis
CTRIA6 CTRIAR	4, 19, 34	Fatigue life in <i>Repeats</i> of the loading sequence
	5, 20, 35	Log of fatigue life in <i>Repeats</i> of the loading sequence
	6, 21, 36	Fatigue life in user defined fatigue equivalent units
	7, 22, 37	Log of fatigue life in user defined fatigue equivalent units
	8, 23, 38	Fatigue damage
	9, 24, 39	Log of fatigue damage
	12, 27, 42	Scale factor from Factor of Safety analysis

**Remarks:**

1. CQUAD4, CQUAD8, CQUADR item codes are listed for Grids 1-4, respectively.
2. CTRIA6 and CTRIA6 item codes are listed for Grids 1-3, respectively.
3. Note that FTGDEF has a keyword “SEAMW” that may have the field NDSIDi, which lists seam line grids. If this is used, it means that some grids of the elements will not have fatigue life associated with them and are excluded from the analysis.



## Equivalent Radiated Power (ERP) Item Codes

Table 7-17 Equivalent Radiated Power (ERP) Item Codes

Element Name	Item Code	Item
QUAD4, CQUADR, CTRIA3,CTRIAR	2	ERP value
QUAD4, CQUADR, CTRIA3,CTRIAR	3	ERP Fraction
QUAD4, CQUADR, CTRIA3,CTRIAR	4	ERP(DB)



# 8

## Degree-of-Freedom Sets

- Degree-of-Freedom Set Definitions
- Degree-of-Freedom Set Bulk Data Entries

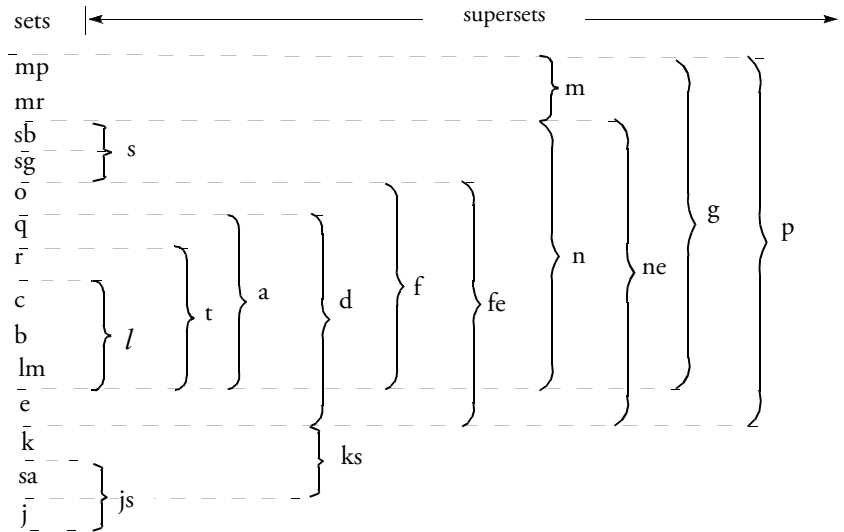
## Degree-of-Freedom Set Definitions

Each degree-of-freedom is a member of one mutually exclusive set. Set names have the following definitions:

Set Name	Definition
mp	Degrees-of-freedom eliminated by <u>m</u> ultipoint constraints.
mr	Degrees-of-freedom eliminated by <u>m</u> ultipoint constraints created by the <u>r</u> igid elements using the LGELIM method on the Case Control command RIGID.
sb*	Degrees-of-freedom eliminated by <u>s</u> ingle-point constraints that are included in <u>b</u> oundary condition changes and by the AUTOSPC feature. (See the sz set)
sg*	Degrees-of-freedom eliminated by <u>s</u> ingle-point constraints that are specified on the PS field on <u>G</u> RID Bulk Data entries.
sz	Degrees-of-freedom eliminated by the AUTOSPC feature.
o	Degrees-of-freedom <u>o</u> mitted by structural matrix partitioning.
q	Generalized degrees-of-freedom assigned to component modes and residual vectors.
r	<u>r</u> eference degrees-of-freedom used to determine free body motion.
c	Degrees-of-freedom that are free during component mode synthesis or dynamic reduction.
b	Degrees-of-freedom fixed during component mode analysis or dynamic reduction.
lm	<u>L</u> agrange <u>m</u> ultiplier degrees-of-freedom created by the rigid elements using the LAGR method on the Case Control command, RIGID.
e	<u>e</u> xtra degrees-of-freedom introduced in dynamic analysis.
sa	Permanently constrained aerodynamic degrees-of-freedom.
k	Aerodynamic mesh point set for forces and displacements on the aero mesh.
j	Aerodynamic mesh collocation point set (exact physical interpretation is dependent on the aerodynamic theory).

\*Strictly speaking, sb and sg are not exclusive with respect to one another. Degrees-of-freedom may exist in both sets simultaneously. Since these sets are not used explicitly in the solution sequences, this need not concern the user. However, those who use these sets in their own DMAPs should avoid redundant specifications when using these sets for partitioning or merging operations. That is, a degree-of-freedom should not be specified on both a PS field of a GRID entry (sg set) and on a selected SPC entry (sb set). Redundant specifications will cause UFM 2120 in the VEC module and behavior listed in [MSC Nastran DMAP Programmer's Guide](#) for the UPARTN module. These sets are exclusive, however, from the other mutually exclusive sets.

Each degree-of-freedom is also a member of one or more combined sets called “supersets.” Supersets have the following definitions:



Set Name	Meaning (+ indicates union of two sets)
$s = sb + sg$	all degrees-of-freedom eliminated by single point constraints
$l = b + c + lm$	the degrees-of-freedom remaining after the reference degrees-of-freedom are removed (degrees-of-freedom left over)
$t = l + r$	the total set of physical boundary degrees-of-freedom for superelements
$a = t + q$	the analysis set used in eigensolution
$d = a + e$	the set used in dynamic analysis by the direct method
$f = a + o$	unconstrained (free) structural degrees-of-freedom
$fe = f + e$	free degrees-of-freedom plus extra degrees-of-freedom
$n = f + s$	all degrees-of-freedom not constrained by multipoint constraints
$ne = n + e$	all degrees-of-freedom not constrained by multipoint constraints plus extra degrees-of-freedom
$m = mp + mr$	all degrees-of-freedom eliminated by multipoint constraints
$g = n + m$	all degrees-of-freedom including scalar degrees-of-freedom
$p = g + e$	all physical degrees-of-freedom including extra point degree-of-freedom
$ks = k + sa$	the union of k and the re-used s-set (6 dof per grid)
$js = j + sa$	the union of j and the re-used s-set (6 dof per grid)

table continued on the next page

Set Name	Meaning (+ indicates union of two sets)
$fr = o + l$	statically independent set minus the statically determinate supports ( $fr = \underline{f} - q - \underline{r}$ )
$v = o + c + r$	the set free to vibrate in dynamic reduction and component mode synthesis
$al = a - lm$	a-set without Lagrange multiplier degree-of-freedoms.
$dl = d - lm$	d-set without Lagrange multiplier degree-of-freedoms.
$gl = g - lm$	g-set without Lagrange multiplier degree-of-freedoms.
$ll = l - lm$	l-set without Lagrange multiplier degree-of-freedoms.
$nf = ne - lm$	ne-set without Lagrange multiplier degree-of-freedoms.
$pl = p - lm$	p-set without Lagrange multiplier degree-of-freedoms.
$tl = t - lm$	t-set without Lagrange multiplier degree-of-freedoms.
$nl = n - lm$	n-set without Lagrange multiplier degree-of-freedoms.
$fl = f - lm$	f-set without Lagrange multiplier degree-of-freedoms.
$ff = fe - lm$	fe-set without Lagrange multiplier degree-of-freedoms.

The a-set and o-set are created in the following ways:

1. If only OMIT<sub>i</sub> entries are present, then the o-set consists of degrees-of-freedom listed explicitly on OMIT<sub>i</sub> entries. The remaining f-set degrees-of-freedom are placed in the b-set, which is a subset of the a-set.
2. If ASET<sub>i</sub> or QSET<sub>i</sub> entries are present, then the a-set consists of all degrees-of-freedom listed on ASET<sub>i</sub> entries and any entries listing its subsets, such as QSET<sub>i</sub>, SUPORT<sub>i</sub>, CSET<sub>i</sub>, and BSET<sub>i</sub> entries. Any OMIT<sub>i</sub> entries are redundant. The remaining f-set degrees-of-freedom are placed in the o-set.
3. If there are no ASET<sub>i</sub>, QSET<sub>i</sub>, or OMIT<sub>i</sub> entries present but there are SUPORT<sub>i</sub> entries present, then the entire f-set is placed in the a-set and the o-set is not created.
4. There must be at least one explicitly ASET<sub>i</sub>, QSET<sub>i</sub>, or OMIT<sub>i</sub> entry for the o-set to exist, even if the ASET<sub>i</sub>, QSET<sub>i</sub>, or OMIT<sub>i</sub> entry is redundant.

In dynamic analysis, additional vector sets are obtained by a modal transformation derived from real eigenvalue analysis of the a-set. These sets are as follows:

$\xi_o$  = rigid body (zero frequency) modal degrees-of-freedom

$\xi_f$  = finite frequency modal degrees-of-freedom

$\xi_i = \xi_o + \xi_f$ , the set of all modal degrees-of-freedom

One vector set is defined that combines physical and modal degrees-of-freedom:

$u_h = \xi_i + u_e$ , the set of all modal degrees-of-freedom

The membership of each degree-of-freedom can be printed by use of the Bulk Data entries  
PARAM,USETPRT and PARAM,USETSEL.

## Degree-of-Freedom Set Bulk Data Entries

Degrees-of-freedom are placed in sets as specified by the user on the following Bulk Data entries:

Name	Bulk Data Entry Name
m	MPC, MPCADD, MPCAX, POINTAX, RBAR, RBAR1, RBE1, RBE2, RBE3, RJOINT, RROD, RSPLINE, RSSCON, RTRPLT, RTRPLT1
sb	SPC, SPC1, SPCADD, SPCAX, FLSYM, GMSPC*, BNDGRID, PARAM,AUTOSPC,YES
sz	PARAM,AUTOSPC,YES
sg	GRID, GRIDB, GRDSET (PS field)
o	OMIT, OMIT1, OMITAX, GRID (SEID field), SESET
q	QSET, QSET1
r	SUPPORT, SUPPORT1, SUPAX
c	CSET, CSET1, BNDFREE, BNDFRE1
b	BSET, BSET1, BNDFIX, BNDFIX1
e	EPOINT
sa	CAEROi
k	CAEROi
a	ASET, ASET1, Superelement exterior degrees-of-freedom, CSUPEXT
ap	ACCSPT
rv	RVDOF and RVDOF1
u1 - u6	USET, USET1, SEUSET, and SEUSET1

In superelement analysis, the appropriate entry names are preceded by the letters SE, and have a field reserved for the superelement identification number. This identification is used because a boundary (exterior) grid point may be in one mutually exclusive set in one superelement and in a different set in the adjoining superelement. The SE-type entries are internally translated to the following types of entry for the referenced superelement:

Entry Type	Equivalent Type
SEQSETi	QSETi
SESUP	SUPPORT
SECSETi	CSETi, BNDFREE, DNBDRE1
SEBSETi	BSETi, BNDFIXi



# 9

## Bulk Data Entries

- Key to Descriptions
- Bulk Data Entry Descriptions

## Key to Descriptions

The name of the entry. Must be entered as shown.

The field names in fields 2 through 9 are for reference only. Names enclosed in quotation marks represent character constants; e.g., "THRU" on ASET1 entry.

A brief sentence about the function of the entry is

**ACMODL** Fluid-Structure Interface Modeling

Defines modeling parameters for the interface between the fluid and the

**Format:**

1	2	3	4	5	6	7	8	9	10
ACMODL	INTER	INFOR	FSET	SSET	FSTO				

**Example:**

ACMODL					0.002				
--------	--	--	--	--	-------	--	--	--	--

If a box in fields 2 through 9 is shaded, then the field must be left blank.

If the box in field 10 is shaded, then no continuation entries may follow.

**FieldContents**

INTERType of interface between the fluid and the structure. See Remark 1. (Character = "IDENT" or "DIFF". Default = "DIFF").

INFORIndicates whether FSET and SSET are to be checked at only those grid points referenced by FSET and SSET that contains a list of fluid grid points on the interface. See Remark 3.

FSETIdentification number of a SET1 entry that contains a list of fluid grid points on the interface. See Remark 2. (Integer > 0 or blank).

SSETIdentification number of a SET1 entry that contains a list of fluid grid points on the interface. See Remark 2. (Integer > 0 or blank).

FSTOLTolerance, in units of length, used in determining the fluid-structure interface. (Integer > 0 or blank). (Default = 0.001).

Each of the fields is briefly described. Further details may be discussed under Remarks.

Character constants are enclosed in quotation marks to distinguish them from field names. Do not input these quotation marks.

Under contents, "blank" usually means that this feature can be optionally selected by the user. It may also mean the default action or value is described under Remarks. When content item is listed as, for example, (Integer>0 or blank) a value of 0 entered in the field is equivalent to "blank".

The field's type (Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, then a value must be specified by the user.

**Remarks:**

- Only one ACMODL entry is allowed to assume INTER = "DIFF" and FSTO = 0.
- If INFOR = "ALL", then both FSET and SSET are checked at only those grid points referenced by FSET and SSET that contains a list of fluid grid points on the interface. See Remark 3.
- See the *MSC Nastran Reference Guide*, 15 for a description of the methods used in determining the fluid-structure interface.
- The points referenced by FSET and SSET must lie exactly on the fluid-structure interface. The remarks are generally arranged in order of importance and indicate such things as the entry's relationship to other entries or commands, restrictions, and recommendations on usage, or further details regarding the fields.

The remarks are generally arranged in order of importance and indicate such things as the entry's relationship to other entries or commands, restrictions, and recommendations on usage, or further details regarding the fields.

The field's type (Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, then a value must be specified by the user.

Under contents, "blank" usually means that this feature can be optionally selected by the user. It may also mean the default action or value is described under Remarks. When content item is listed as, for example, (Integer>0 or blank) a value of 0 entered in the field is equivalent to "blank".

Each of the fields is briefly described. Further details may be discussed under Remarks.

Character constants are enclosed in quotation marks to distinguish them from field names. Do not input these quotation marks.

Under contents, "blank" usually means that this feature can be optionally selected by the user. It may also mean the default action or value is described under Remarks. When content item is listed as, for example, (Integer>0 or blank) a value of 0 entered in the field is equivalent to "blank".

The field's type (Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. If no default value is given, then a value must be specified by the user.

## The Bulk Data Section

The Bulk Data Section contains entries that specify model geometry, element connectivity, element and material properties, constraints (boundary conditions), and loads. Some entries, such as loads and constraints, are selected by an appropriate Case Control command.

Entries are prepared in either fixed or free field format. The descriptions in this section show only the fixed format. For a description of the various format options, see the [Use of Parameters](#) in the *MSC Nastran Reference Guide*.

New Bulk Data Entries added after the MSC Nastran 2011 version are only valid with the default option IFPSTAR=YES.

## Bulk Data Entry Descriptions

Each Bulk Data entry is described as follows:

### Description

A brief sentence about the function of the entry is given.

### Format

The name of the entry is given in the first field. The subsequent fields are described under the Field and Contents Section. Shaded fields must be left blank. If field 10 is shaded, then no continuation entries are permitted. Character strings enclosed in quotation marks must be specified without the quotation marks as shown in the example.

### Example

A typical example is given.

### Field and Contents

Each of the fields 2 through 9 that are named in the Format section is briefly described under Contents. The field's type (e.g., Integer, Real, or Character), allowable range, and default value are enclosed in parentheses. The field must be specified by the user if no default value is given.

### Remarks

The remarks in the Remarks Section are generally arranged in order of importance and indicate such things as how the Bulk Data entry is selected in the Case Control Section, its relationship to other entries, restrictions and recommendations on its use, and further descriptions of the fields.

## Format of Bulk Data Entries

### Real, Integer, and Character Input Data

Nastran is quite particular about the input requirements for data entry. The three possible types of data entries are Integer, Real, and Character (sometimes called literal, or BCD-binary coded decimal). The three types of data are described as follows:

Integer	Cannot contain a decimal point.
Real	Must contain a decimal point. If however, system cell 444 is set to 9 and the only possible value format is real, then an integer representation (less decimal point) will be converted to real. A single decimal point is not be considered a real zero value. Either 0 or 0. (a digit) must be supplied.
Character	Can be alphanumeric, and with exceptions (listed on specific bulk data entries) should always start with an alpha character. Legal alpha characters are the English: A-Z and a-z. Legal numeric characters are 0-9, and in special circumstances in fields that allow user naming the dash (-) and the underscore (_). In user naming fields use of \$, &, *, =, +, period (.), comma (,) should be avoided as they often have special meaning to the nastran bulk data interpreter. Misspelled required words such as the name of an entry will result in a fatal message. Substitution of non English alphabetic characters will cause a fatal message to be issued.

Real numbers may be entered in a variety of ways. For example, the following are all acceptable versions of the real number seven:

```
7.0          .7E1          0.7+1
.70+1       7.E+0         70.-1
```

### Free, Small, and Large Field Formats

Nastran has three different field formats for input data:

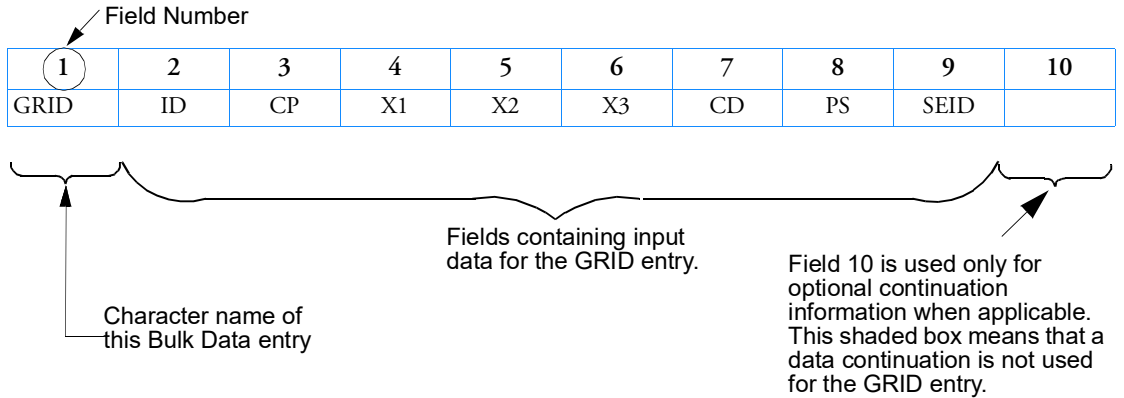
Free Field Format	Input data fields are separated by commas.
Small Field Format	Ten fields of eight characters each.
Large Field Format	Ten fields-eight fields containing actual data are sixteen characters each (fields 2-5 and 6-9). Large fields are used when greater numerical accuracy is required. See the <a href="#">Large Field Format, 1118</a> for additional information.

The NASTRAN statement, File Management Section, Executive Control Section, and Case Control Section use free field format. The Bulk Data Section allows the use of any of the three formats.

Nastran Bulk Data contains ten fields per input data entry. The first field contains the character name of the Bulk Data item (e.g., GRID, CBAR, MAT1, etc.). Fields two through nine contain data input information

for the Bulk Data entry. The tenth field never contains data—it is reserved for entry continuation information, if applicable.

Consider the format of a typical MSC Nastran Bulk Data entry, the GRID entry, which is used in Nastran to describe the geometry of the structural model.



Example:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

We will now represent this example in free field, small field, and large field formats.

### Free Field Format

In free field format, data fields are separated by commas or blanks (commas are strongly recommended). The following shows the GRID Bulk Data entry example in free field format:

GRID, 2,, 1.0, -2.0, 3.0,, 136

Two consecutive commas indicate an empty field

The rules for free field format are as follows:

- Free field data entries must start in column 1.
- To skip one field, use two commas in succession. To skip two fields, use three commas in succession (and so on).
- Integer or character fields with more than eight characters cause a fatal error.
- Real numbers with more than eight characters are rounded off and lose some precision. For example, an entry of 1.2345678+2 becomes 123.4568. If more significant digits are needed, use the large field format. If however, NASTRAN IFPSTAR=YES is used, then the entry will automatically be converted to large field format.

- Free field data cannot contain embedded blanks to skip fields. An example of a free field embedded blank is shown:

GRID,2,,1 0,-2.0,3.0,,136

Embedded blank  
not allowed

In a continuation line, if a comma is the first character and in the first 12 columns, this line is free format.

- A dollar sign \$ can be used within a line. All data following the \$ on the line will be for comments. An example of a comment is shown:

DISP(PUNCH)=ALL \$ request punched displacement output for all grids

The free field data entry capability in Nastran have been enhanced to support easy to use data input formats. The following examples illustrate the possible forms of the free field data input and the resulting translation to the fixed-field format.

Entry with or without user continuation mnemonics.

MATT9,1101,2 , 3 , 4 , , , , 8 , +P101  
+P101, 9 , , , , 13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1101	2	3	4				8	+P101
+P101	9				13				

GRID,100,,1.0,0.0,0.0,,456

Translates to:

GRID	100		1.0	0.0	0.0		456		
------	-----	--	-----	-----	-----	--	-----	--	--

The continuation mnemonics are not included because they are not required. This is illustrated by the entry with automatic continuation:

SPC1,100,12456,1,2,3,4,5,6,7,8,9,10

Translates to:

1	2	3	4	5	6	7	8	9	10
SPC1	100	12456	1	2	3	4	5	6	
	7	8	9	10					

If more than 80 characters of data are required, the free field may be continued in the next line provided that the next entry starts with a comma in the first column. The next entry will be a logical continuation of the first. For example, the free-field entry:

MATT9,1151,2,3,4,,,,8  
,9,,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	
	9				13				

Which is equivalent to:

MATT9,1151,2,3,4,,,,8,+  
,9,,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1151	2	3	4				8	+
+	9				13				

The free field data entry can be used to input mixed Small Field, Large Field continuations. Note that the plus (+) and asterisk (\*) characters are used to indicate Small Field and Large Field input form respectively when free field data entry is used. For example, the entries:

MATT9\*,1302,2,,4,+  
,,,,,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9*	1302		2					4	+
+					13				

MATT9,1303,2,3,4,,,,8,+  
\*,9,,,,+  
\*,13

Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1303	2	3	4				8	+
*	9								
*	13								

MATT9,1355,2,3,,5,,,,8,+  
\*,10,,,,+  
,17



Translates to:

1	2	3	4	5	6	7	8	9	10
MATT9	1355	2	3		5			8	+
*			10						+
+	17								

System cell 363 must be set to 1 (i.e., system(363)=1, or STRICTUAI=1) if more than 80 characters of data are required, then the free-field entry is continued by terminating the parent with a comma. The next entry will be a logical continuation of the first. It is not required to end the first entry at any specific point. This is illustrated by the entry:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,  
17, 18, 19, 20

Translates to:

1	2	3	4	5	6	7	8	9	10
CHEXA	200	200	1	2	3	4	5	6	
	7	8	9	10	11	12	13	14	
	15	16	17	18	19	20			

Because of the feature allowing more than 10 fields of data to be entered on one free field entry, IT IS NOT ALLOWED to terminate a single free field entry with a comma. For example:

CHEXA,200, 200, 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14, 15, 16,

### Small Field Format

Small field format separates a Bulk Data entry into ten equal fields of eight characters each:

←→ 8 character field

1	2	3	4	5	6	7	8	9	10
11	12	13	14	15	16	17	18	19	20

← 80 characters →

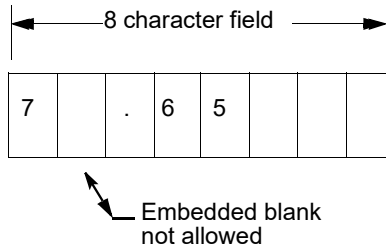
The following is an example of the GRID entry in small field format:

1	2	3	4	5	6	7	8	9	10
GRID	2		1.0	-2.0	3.0		136		

The rules for small field format are as follows:

- Fields 1 and 10 must be left justified.
- Fields 2 through 9 do not need to be either right or left justified, although aligning the data fields is good practice.

- Small field input data cannot contain any embedded blanks. An example of a small field embedded blank is shown:



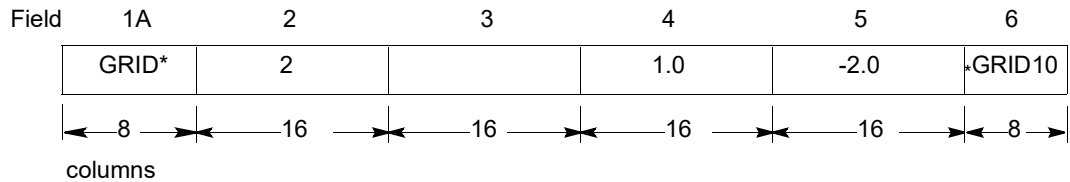
### Large Field Format

A high degree of numerical accuracy is required in some Nastran applications. Large field format is used when small field format does not provide enough significant digits (recall that a minus sign, decimal point, and the “E” in scientific notation count as characters).

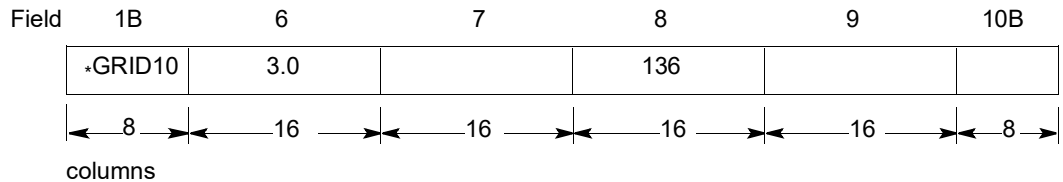
Large field format requires (at least) two lines for each entry: the first and last field of each line contains eight columns, and the fields in between contain 16 columns. Short field becomes two lines. Large field entries are denoted by an asterisk (\*) immediately following the character string in field 1A of the first line and immediately preceding the character string in field 1B of the second line.

The following is an example of the GRID Bulk Data entry example in large field format:

First Line: (Left half of single field)



Second Line: (Right half of single field)



### Continuations

Some Bulk Data entries require more than eight fields (72 columns) of data. Continuations are required in such cases. To do this, a parent entry (the first line) is followed by one or more continuation entries on subsequent lines. For example, consider the following PBAR simple beam property entry (do not worry about what each field represents-this will be explained later):

Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

Continuation Example:

PBAR	39	6	2.9	1.86	2.92	.48			+PB1
+PB1	0.	0.	0.	1.	1.	1.	1.	0.	+PB2
+PB2	.86	.86							

+PB1 in field 10 of the parent entry is an arbitrary (and unique) user-defined pointer to field 1 of the second line. +PB2 in the second line points the third line, and so on.

Continuation fields can also be generated automatically by Nastran (this approach is the recommended practice). To automatically generate a continuation, the continuation line (or lines) must immediately follow the parent Bulk Data entry. In addition, fields 1 and 10 of the continuation line (or lines) must be left blank. In the case of double-width generated continuations are not blank in field 1, but have an "\*" in column 1. Nastran will then generate unique continuations for you. This process is illustrated in the following example:

Input (.DAT) file:

```
CHEXA,      1,      10,      3,      5,      7,      1,      15,      17,
           ,      19,      13,      4,      6,      8,      2,      10,      11,
           ,      12,      9,      16,      18,      20,      14
```

Output (.F06) file:

```
GEAR TOOTH EXAMPLESEPTEMBER 29, 1993 MSC.Nastran 9/ 4/91 PAGE 3

S O R T E D B U L K D A T A E C H O
CARD
COUNT. 1 .. 2 .. 3 .. 4 .. 5 .. 6 .. 7 .. 8 .. 9 .. 10 .
1- CHEXA 1 10 3 5 7 1 15 17 +000001
2-++000001 19 13 4 6 8 2 10 11 +000002
3-++000002 12 9 16 18 20 14 +000003
```

Blank lines are allowed if they are in between entries but not allowed if they are in an entry and further continuations are followed if ifpstar is yes, for example, the blank line in below entry is not allowed. To ignore those blank lines, set system(767) to 1.

```
CHEXA, 1, 10, 3, 5, 7, 1, 15, 17,
, 19, 13, 4, 6, 8, 2, 10, 11,
, 12, 9, 16, 18, 20, 14
```

Nastran Continuation fields (fields one and ten) are replicated using the following conventions:

1. Only letters of the alphabet and integers may be used. They are coded into a base 36 number. That is, the sequence of numbers is 0,1, 2, ..., 8, 9, A, B, ...
2. The first character in field one or ten is not incremented.
3. The continuation fields are incremented by +1 regardless of the value specified by the user.

4. The number of characters in an incremented field will not be increased. For example, if the first field is "0", the thirty-seventh field will also be "0", resulting in an illegal entry. A method to solve this problem would be to start with a first field of "00". This will provide thirty-six squared unique fields.
5. At least one field in fields 2 through 8 of continuation entries must be non-blank.

### Replication

Replication is a limited data generation capability which may be used in a fixed or free-field format and is not currently fully supported with NASTRAN SYSTEM(444)=1 and its use, therefore, is not recommended. Also replication is not at all currently supported with SimX reading a user-generated input file and strange models may be displayed.

1. Duplication of fields from the preceding entry is accomplished by coding the symbol =.
2. Duplication of all trailing fields from the preceding entry is accomplished by coding the symbol ==.
3. Incrementing a value from the previous entry is indicated by coding \*x or \*(x), where x is the value of the increment. "x" should be a real number for real fields or an integer for integer fields.
4. Repeated replication is indicated by coding =n or =(n), where n is the number of images to be generated using the values of the increments on the preceding entry.
5. Data items may be enclosed with parentheses or the parentheses may be deleted.
6. Replication capabilities are described here by example:
  - Continuation entry fields may be incremented or decremented.
  - Repeated replication is indicated by coding =(n) in field 1, where n is number of entry images to be generated using the values of increments from the current or preceding replication entry.

Entered entries:

```
GRID, 101, 17, 1.0, 10.5,,17,3456
= , *1, =, *0.2, ==
=3
```

Generated entries:

GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.5		17	3456		
GRID	103	17	1.4	10.5		17	3456		
GRID	104	17	1.6	10.5		17	3456		
GRID	105	17	1.8	10.5		17	3456		

- A blank in field 1 indicates immediate continuation entry replication. The default continuation entry increment is 1. Example:

```
BSET1,123,1,2,3,4,5,6,7
,,*7,*7,*7,*7,*7,*7,*7
=(3)
```

Generated entries:

BSET1	123	1	2	3	4	5	6	7	+00001
++00001		8	9	10	11	12	13	14	+00002
++00002		15	16	17	18	19	20	21	+00003
++00003		22	23	24	25	26	27	28	+00004
++00004		29	30	31	32	33	34	35	+00005

- A “=(D)” in field 1 indicates delayed continuation entry replication. A maximum of 9 entries may be replicated as a group. The default continuation entry increment is 10. Example:

Entered entries:

```
CTRIA3,10,1,1,10,11/+C1
=(D),*(1),=,*(1),*(1)/(20)
+C1,,,2.0,1.0,1.0
=(2),==
```

Generated entries:

CTRIA3	10	1	1	10	11				+C1
+C1			2.0	1.0	1.0				
CTRIA3	11	1	1	11	12				+C21
+C21			2.0	1.0	1.0				
CTRIA3	12	1	1	12	13				+C41
+C41			2.0	1.0	1.0				

- Parentheses are optional on replication entries and an equal sign may replace an asterisk.

The following is an example of the use of replication, automatic continuation field generation, and the free field format:

```
GRID, 101, 17, 1.0, 10.5,,17,3456
=,*1,=,*0.2, *(0.1), == $ COMMENTS MAY APPEAR AFTER $
=3
EIGR,13,GIV,,30.
,MASS
CBAR,1 ,1 ,101 ,102,0.,0.,1.,,+0
=,*1,=,*1,*1====*1
+0,56
*1,=$
```

The above free-field entries will generate the following Bulk Data in the 8-column format, as seen in the SORTED BULK DATA ECHO:

Note: A “,” should always be used after the “\*1” for the continuation increment even if fixed field format is being used.

CBAR	1	1	101	102	0.	0.	1.		+0
+0	56								

CBAR	2	1	102	103	0.	0.	1.		+1
+1	56								
EIGR	13	GIV		30.					+000001
++000001	MASS								
GRID	101	17	1.0	10.5		17	3456		
GRID	102	17	1.2	10.6		17	3456		
GRID	103	17	1.4	10.7		17	3456		
GRID	104	17	1.6	10.8		17	3456		
GRID	105	17	1.8	10.9		17	3456		

The automatically generated continuation entries start with the number 1, are incremented by 1, and are padded with zeros and plus signs as shown above. If this feature is used, it is the user's responsibility not to enter continuation entries that also use this convention. In particular, data generated on another run and then written to the PUNCH file with the ECHO=PUNCH, will cause problems when introduced into other data with blank continuation fields.

## Identifiers

MSC Nastran is designed to use Identifiers for many quantities, the most common include:

ID	Grid Identification number
EID	Element Identification number
PID	Property Identification number
MID	Material Identification number
CID	Coordinate System number

When these identifications are given they must be unique in their class. All IDs defined on GRID, GRIDB, GRIDF, GRIDS, SPOINT and EPOINT must be unique.

In the case of Element Identification (EID), elements are generally given by "C" entries, such as CBAR, CBEAM, CQUAD, CHEXA, but rigid elements such as RBAR, RBE1, RBE2 are also treated as elements with respect to the EID, and should also be part of this class. All EID must be unique.

All Identification numbers must be greater than zero ( example EID > 0)

Furthermore Element Identification and Grid Identification must be less than 100,000,000 to insure data integrity.

There are two types of Property entries, primary ones and secondary ones. The property id on the primary property entries must be unique.

Referenced by	Primary Property Names
CHBDYP	BDYOR
CHBDYG	BDYOR
CHBDYE	BDYOR
CAABSF	PAABSF
CACINF3	PACINF
CACINF4	PACINF
CAERO1	PAERO1
CAERO2	PAERO2
CAERO3	PAERO3
CAERO4	PAERO4
CAERO5	PAERO5
CAXISYM	PAXISYM
CBAR	PBAR,PBARL, PBRSECT
CBEAM	PBEAM, PBCOMP, PBEAML, PBMSECT,PDISCR, PBMARB6,PMMNUM6
CBEAM3	PBEAM3, PBMSECT
CBELT	PBELTD
CBEND	PBEND
CBUSH	PBUSH, PBUSHT
CBUSH1D	PBUSH1D
CBUSH2D	PBUSH2D
CCONEAX	PCONEAX
CDAMP1	PDAMP
CDAMP1D	PDAMP
CDAMP3	PDAMP
CDAMP5	PDAMP5
CDUMi	PDUMi
CELAS1	PELAS1
CELAS1D	PELAS1
CELAS3	PELAS
CFAST	PFAST
CGAP	PGAP

Referenced by	Primary Property Names
CHACAB	PACABS
CHACBR	PACBAR
CHBDYP	PHBDY
CHEXA	PSOLID, PLSOLID, PCOMPLS
CIFHEX	PCOHE
CIFPENT	PCOHE
CIFQDT	PCOHE
CIFQUAD	PCOHE
CMARKB2	PMARKER
CMARKN1	PMARKER
CMASS1	PMASS
CMASS3	PMASS
CMREBAI	PMREBAI
CMREBAR	PMREBAR
CONSPOT	PBSPOT
CPENTA	PSOLID, PLSOLID
CQUAD	PLPLANE,PLCOMP
CQUAD4	PSHELL,PCOMP,PCOMPG, PLPLANE,PSHELL1,PSHELLD,PLCOMP
CQUAD8	PSHELL,PCOMP,PCOMPG,PLPLANE,PLCOMP
CQUADR	PSHELL,PCOMP,PCOMPG,PLPLANE
CQUADX	PAXSYMH, PLPLANE,PLCOMP
CRAC2D	PRAC2D
CRAC3D	PRAC3D
CROD	PROD,PBELTD
CSEAM	PSEAM
CSHEAR	PSHEAR
CSPH	PSPH
CSPR	PSPRMAT
CSSHL	PSSHL
CSSHLH	PSSHL
CSSHLP	PSSHL
CTETRA	PSOLID, PLSOLID
CTQUAD	PTSHELL



Referenced by	Primary Property Names
CTRIA3	PSHELL,PCOMP,PCOMPG, PLPLANE, PSHELL1
CTRIA6	PSHELL,PCOMP,PCOMPG,PLPLANE
CTRIAR	PSHELL,PCOMP,PCOMPG, PLPLANE
CTRIAX	PAXSYMH
CTTRIA	PTSHELL
CTUBE	PTUBE
CVISC	PVISC
CWELD	PWELD

The secondary Property entries use the same property id as the primary property id. The relationship between these two is shown below.

Primary	Secondary
PSOLID	PSLDN1
PLPLANE	PSHLN2
PCOMP	PCOMPA
PBAR	PBARN1
PBEAM	PBEAM71
PBEAM	PBEMN1
PBEAML	PBEMN1
PCOMP	PCOMPF
PCOMPG	PCOMPF
PDAMP	PDAMPT
PELAS	PELAST
PROD	PROND1
PSHEAR	PSHEARN
PSHELL	PSHLN1
PCOMP	PSHLN1

There are two types of Material entries, primary ones and secondary ones. The material id on the primary material entries must be unique. The material id on the secondary Material entry must agree with the material id on the associated primary Material entry. The correlation between primary and secondary Material entries is given below.

Material Model	Primary	Secondary										
Isotropic	MAT1	MATT1	MATS1	MATVP	MATEP	MATTEP	MATF	MATTF	MATF1	MATFTG	MATVE	MATVE
Shell Anisotropic	MAT2	MATT2		MATVP	MATEP	MATTEP	MATF	MATTF				
Planar Orthotropic	MAT3	MATT3	MATS3	MATVP	MATEP	MATTEP	MATF	MATTF				
Hypoelastic User Sub.	MATUSR	MATTUSR										
General Orthotropic	MATORT	MATTORT	MATSORT				MATF	MATTF				
Shell Orthotropic	MAT8	MATT8	MATS8				MATF	MATTF				
General Anisotropic	MAT9	MATT9										
Geneal Hyperelastic	MATHE	MATTHE									MATVE	MATVE
Gasket	MATG	MATTG										
5th order Mooney-Rivlin	MATHP											
Advanced NLELAST	MATNLE											
Shape Memory	MATSMA											
Isotropic Poroelastic	MATPE1											
CZM Material SOL 400	MCOHE											
CZM Material SOL 600	MDELAM											
Composite Mixture	MIXTURE											
NLELAST	MATNLE											
Digimat Composite	MATDIGI											
Isotropic Heat Transfer	MAT4	MATT4										
Anisotropic Heat Transfer	MAT5	MATT5										

## Bulk Data Entry Summary

This section contains a summary of all Bulk Data entries. The entries are categorized as Geometry, Elements, Material Properties, Constraints, Loads, Solution Control, and Miscellaneous. Entries that are exclusive to SOL 600 have been grouped together at the end of the summary. Entries that are exclusive to Explicit Nonlinear analysis (SOL 700) have also been grouped together at the end of the summary.



## Constraints and Partitioning

### Component Mode Boundary Conditions

<b>BNDFIX</b>	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
<b>BNDFIX1</b>	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
<b>BNDFREE</b>	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.
<b>BNDFREE1</b>	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.
<b>BSET</b>	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
<b>BSET1</b>	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
<b>CSET</b>	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
<b>CSET1</b>	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during component modes calculations.
<b>QSET</b>	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
<b>QSET1</b>	Defines generalized degrees-of-freedom (q-set) to be used for component mode synthesis.
<b>SEBSET</b>	Defines boundary degrees-of-freedom to be fixed (b-set) during component mode synthesis calculations.
<b>SEBSET1</b>	Defines fixed boundary points for superelement.
<b>SECSET</b>	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
<b>SECSET1</b>	Defines boundary degrees-of-freedom to be free (c-set) during component mode synthesis calculations.
<b>SENQSET</b>	Defines number of internally generated scalar points for superelement dynamic reduction.
<b>SEQSET</b>	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
<b>SEQSET1</b>	Defines the generalized degrees-of-freedom of the superelement to be used in component mode synthesis.
<b>SESUP</b>	Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

## Free Body Supports

CYSUP	Defines fictitious supports for cyclic symmetry analysis.
SUPAX	Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.
SUPPORTi	Defines degrees-of-freedom for determinate reactions.

## Multipoint Constraints

MONSUM	Defines a new monitor result that is the weighted sum of existing monitor results.
MPC	Defines a linear relationship for two or more degrees-of-freedom.
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.
MPCD	Defines a load selectable value for nonhomogeneous multi-point constraint.
MPCY	Defines a linear nonhomogeneous relationship for two or more degrees-of-freedom.
MPCAX	Defines multipoint constraints for conical shell problems.
POINTAX	Defines multipoint constraints for point on conical shell.
RBAR	Defines multipoint constraints for rigid bar.
RBEi	Defines multipoint constraints for RBE1, RBE2, RBE3.
RROD	Defines multipoint constraints for rigid rod.
RSPLINE	Defines multipoint constraints for spline element.
RTRPLT	Defines multipoint constraints for rigid triangular plate.

## Partitioning

ASET	Defines degrees-of-freedom in the analysis set (a-set).
ASET1	Defines degrees-of-freedom in the analysis set (a-set).
CSUPEXT	Assigns exterior points to a superelement.
GRID	Defines interior points for a superelement.
OMIT	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMIT1	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
OMITAX	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
RELEASE	Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SESET	Defines interior grid points for a superelement.

## Single Point Constraints

FLSYM	Symmetry control for boundary in axisymmetric fluid problem.
GRID	Includes single point constraint definition.
GRIDB	Includes single point constraint definition.
GRDSET	Includes default for single point constraints.
SPC	Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).
SPC1	Defines a set of single point constraints.
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.
SPCAX	Defines a set of single-point constraints or enforced displacements for conical shell coordinates.
SPCOFF	Defines degrees-of-freedom to be excluded from the AUTOSPC operation.
SPCOFF1	Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation.

## User Sets

DEFUSET	Defines new names for degree-of-freedom sets.
SEUSET	Defines a degree-of-freedom set for a superelement.
SEUSET1	Defines a degree-of-freedom set for a superelement.
USET	Defines a degree-of-freedom set.
USET1	Defines a degrees-of-freedom set.

## Elements

A summary of the capabilities and characteristics of the small strain elements is available in [Element Summary](#) -- [Conventional MSC Nastran Elements](#) in the *MSC Nastran Reference Guide*.

## Aerodynamic Elements

AEFACT	Defines real numbers for aeroelastic analysis.
AELINK	Defines relationships between or among AESTAT and AESURF entries.
AELIST	Defines a list of aerodynamic elements to undergo the motion prescribed with the AESURF Bulk Data entry for static aeroelasticity. Also defines server specific integer data for external spline methods.
AEQUAD4	Defines the connectivity of a quadrilateral aerodynamic element.
AESTAT	Specifies rigid body motions to be used as trim variables in static aeroelasticity.
AESURF	Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.

<b>AESURFS</b>	Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry.
<b>AETRIA3</b>	Defines the connectivity of a triangular aerodynamic element.
<b>CAERO1</b>	Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords.
<b>CAERO2</b>	Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.
<b>CAERO3</b>	Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.
<b>CAERO4</b>	Defines an aerodynamic macro element for Strip theory.
<b>CAERO5</b>	Defines an aerodynamic macro element for Piston theory.
<b>CSSCHD</b>	Defines a scheduled control surface deflection as a function of Mach number and angle of attack.
<b>PAERO1</b>	Defines associated bodies for the panels in the Doublet-Lattice method.
<b>PAERO2</b>	Defines the cross-sectional properties of aerodynamic bodies.
<b>PAERO3</b>	Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.
<b>PAERO4</b>	Defines properties of each strip element for Strip theory.
<b>PAERO5</b>	Defines properties of each strip element for Piston theory.

### **Aerodynamic to Structure Interconnection**

<b>AELISTC</b>	Defines a list of 8-character strings.
<b>SET1</b>	Defines a list of structural grid points.
<b>SET2</b>	Defines a list of structural grid points in terms of aerodynamic macro elements.
<b>SET3</b>	Defines a list of grids, elements or points.
<b>SPBLND1</b>	Defines a strip based blending of two splines.
<b>SPBLND2</b>	Defines a curve based blending of two splines.
<b>SPLINE1</b>	Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
<b>SPLINE2</b>	Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
<b>SPLINE3</b>	Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.
<b>SPLINE4</b>	Defines a curved surface spline for interpolating motion and/or forces for aeroelastic problems on general aerodynamic geometries using either the Infinite Plate, Thin Plate or Finite Plate splining method.

<a href="#">SPLINE5</a>	Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by irregular arrays of aerodynamic points.
<a href="#">SPLINE6</a>	Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.
<a href="#">SPLINE7</a>	Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.
<a href="#">SPLINEX</a>	Defines the input for a spline that will be evaluated with a user-supplied procedure.
<a href="#">SPLINRB</a>	Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.
<a href="#">SPRELAX</a>	Defines relaxation of a spline based on an adjacent spline.

### Axisymmetric Elements

<a href="#">CAXISYM</a>	Defines an axisymmetric shell.
<a href="#">CCONEAX</a>	Defines a conical shell element.
<a href="#">CQUADX</a>	Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
<a href="#">CTRIAX</a>	Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis.
<a href="#">CTRIAX6</a>	Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.
<a href="#">MREVERS</a>	Defines which elements, if any, required node numbering to be reversed in SOL 600.
<a href="#">PCONEAX</a>	Defines the properties of a conical shell element described on a CCONEAX entry.

### Cohesive Zone Modeling Elements

<a href="#">CIFHEX</a>	Defines a solid interface cohesive zone modeling element in SOL 400.
<a href="#">CIFPENT</a>	Defines a solid interface cohesive zone modeling element in SOL 400.
<a href="#">CIFQDX</a>	Defines an axisymmetric interface cohesive zone modeling element in SOL 400.
<a href="#">CIFQUAD</a>	Defines a planar interface element for cohesive zone modeling element in SOL 400.
<a href="#">MCOHE</a>	Defines damage and cohesive energy for interface zone modeling elements in SOL 400.
<a href="#">PCOHE</a>	Defines the properties of a cohesive interface zone modeling elements in SOL 400.

### Connector Elements

<a href="#">CFAST</a>	Defines a fastener with material orientation connecting two surface patches.
<a href="#">CSEAM</a>	Defines a seam-line connecting two surfaces.
<a href="#">CSLOT3</a>	Defines an element connecting three points that solve the wave equation in two dimensions.

<a href="#">CSLOT4</a>	Defines an element connecting four points that solve the wave equation in two dimensions.
<a href="#">CWELD</a>	Defines a weld or fastener connecting two surface patches or points.
<a href="#">PFAST</a>	Defines the CFAST fastener property values.
<a href="#">PSEAM</a>	Defines the CSEAM fastener property values.
<a href="#">PWELD</a>	Defines the property of connector (CWELD) elements.
<a href="#">SWLDPRM</a>	Overrides default values of parameters for connector search.

### Slidelines Contact (SOL 106, 129)

<a href="#">BCONP</a>	Defines the parameters for a contact region and its properties.
<a href="#">BFRIC</a>	Defines frictional properties between two bodies in contact.
<a href="#">BLSEG</a>	Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.
<a href="#">BWIDTH</a>	Defines widths or thicknesses for line segments in 3-D or 2-D slideline contact defined in the corresponding BLSEG Bulk Data entry.

### Gap Elements

<a href="#">CGAP</a>	Defines a gap or friction element.
<a href="#">PGAP</a>	Defines the properties of the gap element (CGAP entry).

### Crack Tip Elements

<a href="#">CRAC2D</a>	Defines a two-dimensional crack tip element.
<a href="#">CRAC3D</a>	Defines a three-dimensional crack tip element.
<a href="#">PRAC2D</a>	Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.
<a href="#">PRAC3D</a>	Defines the properties of the CRAC3D structural element.

### Damping Elements

<a href="#">CBUSH1D</a>	See line elements.
<a href="#">CDAMP1</a>	Defines a scalar damper element.
<a href="#">CVISC</a>	Defines a viscous damper element.
<a href="#">DAMPING</a>	Specifies the values for parameter damping and/or selects optional HYBRID damping.
<a href="#">HYBDAMP</a>	Specifies hybrid damping parameters.
<a href="#">PBUSH1D</a>	See line elements.
<a href="#">PDAMP</a>	Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.



PDAMP5	Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.
PDAMPT	Defines the frequency-dependent properties for a PDAMP Bulk Data entry.
PVISC	Defines properties of a one-dimensional viscous damping element (CVISC entry).
ROTHYBD	Defines Hybrid damping for rotordynamics.

### Dummy Elements

ADUMi	Defines attributes of the dummy elements ( $1 \leq i \leq 9$ )
CDUMi	Defines a dummy element ( $1 \leq i \leq 9$ )
PDUMi	Defines the properties of a dummy element ( $1 \leq i \leq 9$ ). Referenced by the CDUMi entry.
PLOTEL	Defines a one-dimensional dummy element for use in plotting.

### Fluid and Acoustic Elements

CAABSF	Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.
CACINFi	Defines few types of acoustic infinite elements.
CAXIFi	Defines an axisymmetric fluid element that connects $i = 2, 3,$ or $4$ fluid points.
CAXISYM	Defines two or three node axisymmetric thick shell elements.
CFLUIDi	Defines three types of fluid elements for an axisymmetric fluid model.
CHACAB	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHACBR	Defines the acoustic barrier element.
CHEXA	Connection definition for a pentahedron element in coupled fluid-structural analysis.
CPENTA	Connection definition for a tetrahedron element in coupled fluid-structural analysis.
CSLOTi	Defines slot element for acoustic cavity analysis.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
ELIST	Defines a list of structural elements for virtual fluid mass.
MAT10	Fluid Material Property Definition.
PAABSF	Defines the properties of a frequency-dependent acoustic absorber element.
PACABS	Defines the properties of the acoustic absorber element.
PACBAR	Defines the properties of the acoustic barrier element.
PACINF	Defines the properties of acoustic infinite elements.
PANEL	Selects the set of structural grid points that define one or more panels.

- PSOLID** Defines the fluid properties of solid elements (CHEXA, CPENTA, and CTETRA entries).
- SET1** Defines a list of structural grid points for aerodynamic analysis, XY-plots for SORT1 output, and the PANEL entry.

**Equivalent Radiated Power**

- ERPPNL** Equivalent Radiated Power Definition

**Poroelasticity (PEM)**

- ACTRIM** ACTRAN Trimmed Material Matrices for SOL 108/111
- ACLOAD** ACTRAN Acoustic Pressure Load Matrices for SOL 108/111
- MATPEI** Isotropic Poroelastic Material Property definition
- MATF1** Frequency Dependent Isotropic Material Definition

**Heat Transfer Elements**

- BDYOR** Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.
- CHBDYi** Connection definition for surface element (CHBDYE, CHBDYG, CHBDYP).
- CONTRLT** Thermal control element for heat transfer analysis.
- PHBDY** A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

The PRODN1, PSHLN1, PSHLN2, PSLDN1, PSHEARN, PLCOMP, and PCOMPLS may be used to extend the nonlinear capabilities of heat transfer elements in SOL 400. PCOMPLS may also be used to extend the nonlinear capabilities of heat transfer elements in all linear solutions between SOL101 and SOL112.

The following elastic elements may also be used as heat conduction elements:

- Linear:           CBAR, CROD, CONROD, CTUBE, CBEAM, CBEND.
- Membrane:       CTRIA3, CTRIA6, CQUAD4, CQUAD8.
- Axisymmetric:   CTRIAX6.
- Solid:            CTETRA, CHEXA, CPENTA.

**Line Elements**

- BAROR** Default for orientation and property for CBAR.
- BEAMOR** Defines default vlaues for field 3 and fields 6 through 8 of the CBEAM entry.
- CBAR** Defines a simple beam element.
- CBEAM** Defines a beam element.



<a href="#">CBEAM3</a>	Defines a three-node beam element.
<a href="#">CBEND</a>	Defines a curved beam, curved pipe, or elbow element.
<a href="#">CBUSH1D</a>	Defines the connectivity of a one-dimensional spring and viscous damper element.
<a href="#">CFAST</a>	Defines a fastener with material orientation connecting two surface patches.
<a href="#">CINTC</a>	Defines a line interface element with specified boundaries.
<a href="#">CMREBAI</a>	Defines Rebar elements and matching “Matrix” solid element using the Marc REBAR with INSERT Method. (SOL 600)
<a href="#">CMREBAR</a>	Defines Rebar elements with matching “Matrix” solid elements using the Marc REBAR without INSERT Method. (SOL 600)
<a href="#">CONROD</a>	Defines a rod element without reference to a property entry.
<a href="#">CROD</a>	Defines a tension-compression-torsion element.
<a href="#">CTUBE</a>	Defines a tension-compression-torsion tube element.
<a href="#">CWELD</a>	Defines a weld or fastener connecting two surface patches or points.
<a href="#">PBAR</a>	Defines the properties of a simple beam element (CBAR entry).
<a href="#">PBARL</a>	Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.
<a href="#">PBARN1</a>	Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry.
<a href="#">PBCOMP</a>	Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry.
<a href="#">PBEAM</a>	Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.
<a href="#">PBEAM3</a>	Defines the properties of a three-node beam element (CBEAM3 entry).
<a href="#">PBEAML</a>	Defines the properties of a beam element by cross-sectional dimensions.
<a href="#">PBEMN1</a>	Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry.
<a href="#">PBEND</a>	Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).
<a href="#">PFAST</a>	Defines the CFAST fastener property values.
<a href="#">PBMSECT</a>	Defines the shape of arbitrary cross-section for CBEAM element.
<a href="#">PBRSECT</a>	Defines the shape of arbitrary cross-section for CBAR element.
<a href="#">PBUSH1D</a>	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
<a href="#">PBUSH2D</a>	Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).
<a href="#">PMREBAI</a>	Defines Rebar property information for CMREBAI elements. (SOL 600)
<a href="#">PMREBAR</a>	Defines Rebar property information for CMREBAR elements. (SOL 600)

<a href="#">PROD</a>	Defines the properties of a rod element (CROD entry).
<a href="#">PRODN1</a>	Defines nonlinear property extensions for the PROD in SOL 400.
<a href="#">PTUBE</a>	Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).
<a href="#">PWELD</a>	Defines the properties of connector (CWELD) elements.

## Mass Elements

<a href="#">CMASSi</a>	Connection definition for scalar mass, also property definition for i=2 or 4.
<a href="#">CONM1</a>	Defines a 6 x 6 symmetric mass matrix at a geometric grid point.
<a href="#">CONM2</a>	Defines concentrated mass at a grid point.
<a href="#">PMASS</a>	Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).
<a href="#">NSM</a>	Non Structural Mass by ID.
<a href="#">NSM1</a>	Non Structural Mass (alternate form).
<a href="#">NSMADD</a>	Non Structural Mass Set Combination.
<a href="#">NSML</a>	Lumped Non Structural Mass by ID.
<a href="#">NSML1</a>	Lumped Non Structural Mass (alternate form).

## Rigid Elements

<a href="#">RBAR</a>	Defines a rigid bar with six degrees-of-freedom at each end.
<a href="#">RBAR1</a>	Alternative format for RBAR.
<a href="#">RBE1</a>	Defines a rigid body connected to an arbitrary number of grid points.
<a href="#">RBE2</a>	Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.
<a href="#">RBE3</a>	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.
<a href="#">RJOINT</a>	Defines a rigid joint element connecting two coinciding grid points.
<a href="#">RROD</a>	Defines a pin-ended element that is rigid in translation.
<a href="#">RSPLINE</a>	Defines multipoint constraints for the interpolation of displacements at grid points.
<a href="#">RSSCON</a>	Defines multipoint constraints to model clamped connections of shell-to-solid elements.
<a href="#">RTRPLT</a>	Defines a rigid triangular plate.
<a href="#">RTRPLT1</a>	Defines a rigid triangular plate (alternate).

### Scalar and Bushing Elements

CBUSH	Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.
CBUSH1D	Defines the connectivity of a one-dimensional spring and viscous damper element.
CBUSH2D	Defines the connectivity of a two-dimensional Linear-Nonlinear element.
CELASi	Connection definition for scalar spring, also property definition for i=2 or 4.
GENEL	Defines a general element.
MGRSPR	Defines grids to add soft spring to ground. (SOL 600)
PBUSH	Defines the nominal property values for a generalized spring-and-damper structural element.
PBUSHT	Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PELAS	Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
PELAST	Defines the frequency dependent properties for a PELAS Bulk Data entry.

### Solid Elements

CHEXA	Defines the connections of the six-sided solid element with eight to twenty grid points.
CPENTA	Defines the connections of a five-sided solid element with six to fifteen grid points.
CTETRA	Defines the connections of the four-sided solid element with four to ten grid points.
PCOMPLS	Defines the linear/nonlinear properties of an n-ply composite material laminate for a layered solid (CHEXA) element in SOL 400 and all linear solution sequences between SOL101 and SOL112.
PLSOLID	Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
PSLDN1	Defines the nonlinear property extensions for a PSOLID entry in SOL 400.
PSOLID	Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

### Surface Elements

CQUAD	Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.
CQUAD4	Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.
CQUAD8	Defines a curved quadrilateral shell or plane strain element with eight grid points.
CQUADR	Defines an isoparametric membrane and bending quadrilateral plate element.
CSHEAR	Defines the properties of a shear panel (CSHEAR entry).

<a href="#">CTRIA3</a>	Defines an isoparametric membrane-bending or plane strain triangular plate element.
<a href="#">CTRIA6</a>	Defines a curved triangular shell element or plane strain with six grid points.
<a href="#">CTRIAR</a>	Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element.
<a href="#">PCOMP</a>	Defines the properties of an n-ply composite material laminate.
<a href="#">PCOMPF</a>	Defines the integration procedure for through the thickness integration of composite shells. (SOLs 400/600 only)
<a href="#">PCOMPG</a>	Defines global (external) ply IDs and properties for a composite material laminate.
<a href="#">PLCOMP</a>	Defines the linear/nonlinear properties of an n-ply composite material laminate of a plane stress, plane strain, or axisymmetric (CQUAD or CQUADX entry) element in SOL 400.
<a href="#">PLPLANE</a>	Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.
<a href="#">PSHELL</a>	Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.
<a href="#">PSHLN1</a>	Defines nonlinear property extensions for a PSHELL or PCOMP or PCOMPG entry in SOL 400.
<a href="#">PSHLN2</a>	Defines nonlinear property extensions for a PLPLANE entry in SOL 400.
<a href="#">PSHEAR</a>	Defines the properties of a shear panel (CSHEAR entry).
<a href="#">PSHEARN</a>	Defines nonlinear property extensions for a PSHEAR in SOL 400.
<a href="#">SNORM</a>	Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

## Geometry

### Adaptive Meshing

<a href="#">HADACRI</a>	Specifies Mesh adaptivity criterion and corresponding parameters.
<a href="#">HADAPTL</a>	Specifies local adaptive Mesh refinement control parameters.

### Axisymmetry

<a href="#">AXIC</a>	Defines the existence of an axisymmetric conical shell problem.
<a href="#">AXIF</a>	Defines basic parameters and the existence of an axisymmetric fluid analysis.
<a href="#">AXSLOT</a>	Defines the harmonic index and the default values for acoustic analysis entries.
<a href="#">FLSYM</a>	Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.

- POINTAX** Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.
- RINGAX** Defines a ring for conical shell problem.
- SECTAX** Defines a sector of a conical shell.

### Coordinate Systems

- BAROR** Defines default values for field 3 and fields 6 through 8 of the CBAR entry.
- BEAMOR** Defines default values for field 3 and fields 6 through 8 of the CBEAM entry.
- CORD1C** Cylindrical coordinate system definition.
- CORD1R** Rectangular coordinate system definition.
- CORD1S** Spherical coordinate system definition.
- CORD3G** Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system.

### Cyclic Symmetry

- CYAX** Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.
- CYJOIN** Defines the boundary points of a segment in cyclic symmetry problems.
- NLCYSYM** Defines information to perform nonlinear cyclic symmetry analysis using SOL 600

### Fluid Points

- ACMODL** Defines modeling parameters for the Fluid-Structure Interface.
- FREEPT** Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.
- FSLIST** Defines the fluid points (RINGFL entry) that lie on a free surface boundary.
- GRID** Defines fluid points in coupled fluid-structural analysis.
- GRIDB** Grid point location on RINGFL.
- GRIDF** Defines a scalar degree-of-freedom for harmonic analysis of a fluid.
- GRIDS** Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.
- PRESPT** Defines the location of pressure points in the fluid for recovery of pressure data.
- RINGFL** Defines a circle (fluid point) in an axisymmetric fluid model.
- SLBDY** Defines a list of slot points that lie on an interface between an axisymmetric fluid and a set of evenly spaced radial slots.

## Geometric Points and Spline Grids

- GMBNDC** Defines a geometric boundary consisting of edges along a curve interface. The boundary may consist of edges of shell, beam, solid elements
- POINT** Defines the X, Y, Z coordinates of a geometric point (See **PBMSECT**)

## Grid Points

- AERGRID** Defines the location of an aerodynamic grid point.
- GRID** Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.
- GRIDB** Defines the location of a geometric grid point on a fluid point (**RINGFL** entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.
- GRDSET** Defines default options for fields 3, 7, 8, and 9 of all **GRID** entries.
- SEQGP** Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

## Scalar Points

- EPOINT** Defines extra points for use in dynamic problems.
- SEQGP** Grid and scalar point number resequencing.
- SPOINT** Defines scalar points.

## Superelement Analysis

- CSUPER** Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.
- CSUPEXT** Assigns exterior points to a superelement.
- EXTRN** Defines a boundary connection for an external superelement.
- GRID** Defines interior points for a superelement.
- RELEASE** Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.
- SEBNDRY** Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
- SEBULK** Defines superelement boundary search options and a repeated, mirrored, or collector superelement.
- SECONCT** Explicitly defines grid and scalar point connection procedures for a partitioned superelement.
- SEDLINK** Relates one design variable of a **PART SE** to one or more other design variables from other **PART SEs**.



SEDRSP2	Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.
SEDRSP3	Defines constituents from multiple PART SE for an external response using user-supplied routine(s).
SEELT	Reassigns superelement boundary elements to an upstream superelement.
SEEXCLD	Defines grid points that will be excluded during the attachment of a partitioned superelement.
SELABEL	Defines a label or name to be printed in the superelement output headings.
SELOAD	Selects and/or scales external superelement loads
SELOC	Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.
SEMPLN	Defines a mirror plane for mirroring a partitioned superelement.
SEQSEP	Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.
SESET	Defines interior grid points for a superelement.
SETREE	Specifies superelement reduction order.

## Module Analysis

<a href="#">EXCLUDE</a>	Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module.
<a href="#">MDBCNCT</a>	Defines the Touching and Touched Contact Bodies in Different Modules.
<a href="#">MDBCTB1</a>	Defines a Contact Table for Bodies in Different Modules.
<a href="#">MDBNDRY</a>	Module to Module Boundary Point Definitions.
<a href="#">MDBOLT</a>	Defines the Multi-Point Constraints for a Bolt Between Two Modules.
<a href="#">MDBULK</a>	Module Type Definitions.
<a href="#">MDCONCT</a>	Module Boundary Point Connections.
<a href="#">MDDMIG</a>	Direct Matrix Input at Points Defined in Two or More Modules.
<a href="#">MDEXCLD</a>	Module to Module Excluded Boundary Point Definitions.
<a href="#">MDFAST</a>	A Shell Patch Fastener Connection Between Two Modules.
<a href="#">MDFSET</a>	Set Definition for Fluid Elements or Grid Points in Modules.
<a href="#">MDLABEL</a>	Module Output Label.
<a href="#">MDLOC</a>	Module Reposition by Translation and/or Rotation.
<a href="#">MDMIR1</a>	Defines a Module mirror by three non-collinear points on the mirror plane.
<a href="#">MDMIR2</a>	Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane.
<a href="#">MDMOVE</a>	Defines a Module repositioning sequence. References MDMIRi, MDROT <sub>i</sub> , and MDTRAN entries.
<a href="#">MDMPC</a>	Multipoint Constraint Between Two or More Modules.
<a href="#">MDMPLN</a>	Module Reposition by Mirroring.
<a href="#">MDRBE3</a>	Rigid Body Element Between Two or More Modules, Form 2.
<a href="#">MDRBE3</a>	Interpolation Constraint Element Between Two or More Modules.
<a href="#">MDRJNT</a>	Rigid Joint Between Two Modules.
<a href="#">MDROT1</a>	Defines a Module rotation by specifying a rotation vector and reference point.
<a href="#">MDROT2</a>	Defines a Module rotation by specifying a coordinate system axis for the rotation vector.
<a href="#">MDRRD</a>	Rigid Pin-Ended Element Connection Between Two Modules.
<a href="#">MDSEAM</a>	A Shell Patch SEAM Connection Between Two Modules.
<a href="#">MDSSET</a>	Set Definition for Structural Elements or Grid Points in Modules.
<a href="#">MDTRAN</a>	Defines a Module translation by specifying a vector.
<a href="#">MDWELD</a>	Weld or Fastener Element Connection Between Two Modules.

## Loads

### Dynamic Loads

ACSRCE	Defines acoustic source as a function of power vs. frequency.
DAREA	Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.
DELAY	Defines the time delay term $\tau$ in the equations of the dynamic loading function.
DLOAD	Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.
DPHASE	Defines the phase lead term $\theta$ in the equation of the dynamic loading function.
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LSEQ	Defines a sequence of static load sets.
NOLINI	Nonlinear transient load definition.
NLRGAP	Defines a nonlinear transient radial (circular) gap.
RBE3U	Defines methods to distribute applied loads to a surface. (SOL 600)
RLOAD1	Frequency dependent excitation definition.
TABLED1	Tabular functions for generating dynamic loads.
TLOAD1	Time dependent excitation definition.

### Heat Transfer Loads

CONV	Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
CONVM	Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).
PCONV	Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
PCONVM	Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.
QBDY1	Defines a uniform heat flux into CHBDYj elements.
QBDY2	Defines grid point heat flux into CHBDYj elements.
QBDY3	Defines a uniform heat flux load for a boundary surface.
QHBDY	Defines a uniform heat flux into a set of grid points.
QVECT	Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

QVOL	Defines a rate of volumetric heat addition in a conduction element.
RADBC	Specifies an CHBDYi element face for application of radiation boundary conditions.
RADBND	Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.
RADCAV	Identifies the characteristics of each radiant enclosure.
RADLST	Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.
RADMTX	Provides the $F_{ji} = A_j f_{ji}$ exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.
RADSET	Specifies which radiation cavities are to be included for radiation enclosure analysis.
SLOAD	Defines concentrated static loads on scalar or grid points.
SLOADN1	Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPB3	Defines a temperature field for the three-node beam element (CBEAM3 entry).
TEMPBC	Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.
TEMPD	Specifies default initial temperature at grid points.
TEMPN1	TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400
VIEW	Defines radiation cavity and shadowing for radiation view factor calculations.
VIEW3D	Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

### Static Loads

ACCEL	Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.
ACCEL1	Defines static acceleration loads at individual GRID points.
CLOAD	Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOL 106 or 153).
DEFORM	Defines enforced axial deformation for one-dimensional elements for use in statics problems.
FORCE	Defines a static concentrated force at a grid point by specifying a vector.
FORCE1	Defines a static concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.
FORCE2	Defines a static concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.
FORCEAX	Defines a concentrated force on a conical shell ring.

FORCEi	Defines concentrated load at grid point.
GRAV	Defines acceleration vectors for gravity or other acceleration loading.
GRIDA	Defines an associative GRID point to be used in the COMBINE step of PAA.
LOAD	Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADB3, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries.
LOADCLID	Defines a loading combination in PAA using the Load ID from SUBCASEs of the Parts
LOADCNAM	Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts
LOADCSUB	Defines a loading combination in PAA using the SUBCASE IDs from the Parts
LOADCYH	Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.
LOADCYN	Defines a physical static or dynamic load for use in cyclic symmetry analysis.
LOADCYT	Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.
LOADOF	Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.
LOADT	Specifies static loads that will use a table to describe their variation with pseudo-time.
LSEQ	Defines a sequence of static load sets.
MOMAX	Defines a static concentrated moment load on a ring of a conical shell.
MOMENT	Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.
MOMENTi	Defines moment at grid point.
PLOAD	Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.
PLOAD1	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.
PLOAD2	Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.
PLOAD4	Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element.
PLOADB3	Defines a distributed load to a CBEAM3 element over entire length of the beam axis.
PLOADX1	Defines surface traction to be used with the CQUADX, CTRIAX, and CTRIAX6 axisymmetric element.
PRESAX	Defines the static pressure loading on a conical shell element.

RBE3U	Defines methods to distribute applied loads to a surface. (SOL 600)
RFORCE	Defines a static loading condition due to an angular velocity and/or acceleration.
SLOAD	Defines concentrated static loads on scalar or grid points.
SPCD	Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.
SPCR	Defines an enforced relative displacement value for a load step in SOL 400.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TEMPP1	Defines temperature field for surface elements.
TEMPRB	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.

## Materials

### Anisotropic

MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT3	Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries.
MAT5	Defines the thermal material properties for anisotropic materials.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MAT9	Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).
MATORT	Define elastic 3D Orthotropic Material (SOL 400, 600. Also all linear solutions between SOL101 and 112 in conjunction with PCOMPLS.

### Fatigue

MATFTG	Defines cyclic material properties (S-N + $\epsilon$ -N) for use in fatigue analysis.
--------	---

**Fluid**

AXIF	Includes default values for mass density and bulk modulus.
AXSLOT	Includes default values for mass density and bulk modulus.
CFLUIDi	Includes mass density and bulk modulus.
FSLIST	Includes mass density at free surface.
MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis.
MFLUID	Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.
SLBDY	Includes mass density at interface between fluid and radial slots.

**Isotropic**

MAT1	Defines the material properties for linear isotropic materials.
MAT4	Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.
MATHP	Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
RADM	Defines the radiation properties of a boundary element for heat transfer analysis.

**Stress Dependent**

CREEP	Defines creep characteristics based on experimental data or known empirical creep law.
MATS1	Specifies stress-dependent material properties for use in applications involving nonlinear materials.
NLMOPTS	Defines nonlinear material option control for SOL 400.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

**Temperature Dependent**

MATTi	Table references for temperature-dependent MATi materials.
RADMT	Specifies table references for temperature dependent RADM entry radiation boundary properties.
TABLEM1	Tabular functions for generating temperature-dependent material properties.
TABLEST	Table references for temperature dependent MATS1 materials.
TEMP	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPAX	Defines temperature sets for conical shell problems.

<b>TEMPD</b>	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
<b>TEMPP1</b>	Defines temperature field for surface elements.
<b>TEMPRB</b>	Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

## Miscellaneous

### Brake Squeal (SOL 400)

<b>BSQUEAL</b>	Specifies data for brake squeal analysis using Implicit Nonlinear (SOL 400).
----------------	--

### Comments

<b>\$</b>	Used to insert comments into the input file. Comment statements may appear anywhere within the input file.
-----------	--

### Delete

<b>/</b>	Control Input Stream
----------	----------------------

### Direct Matrix Input

<b>CONM1</b>	Defines a 6x6 mass matrix at a geometric grid point.
<b>DMI</b>	Defines matrix data blocks.
<b>DMIG</b>	Defines direct input matrices related to grid, extra, and/or scalar points.
<b>DMIG,UACCEL</b>	Defines rigid body accelerations in the basic coordinate system.
<b>DMIAX</b>	Defines axisymmetric (fluid or structure) related direct input matrix terms.
<b>TF</b>	Defines a dynamic transfer function.

### Direct Matrix Input for Aeroelasticity

<b>DMIJ</b>	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries.
<b>DMIJI</b>	Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2.
<b>DMIK</b>	Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points.



**End of Input**

**ENDDATA** Designates the end of the Bulk Data Section.

**Include File**

**INCLUDE** Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**VCCT** Virtual Crack Closure Technique - SOL 400, SOL 600

**LORENZI** Contour Integral Approach for stress intensity calculation - SOL 600

**Materials (SOLs 400/600)**

**COHESIV** Defines data for cohesive materials. (SOL 600)

**MATDIGI** e-XstreamDigimat material interface (SOL 400)

**MATEP** Elasto-plastic material properties.

**MATF** Specifies material failure model. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS.)

**MATG** Gasket material properties.

**MATHE** Hyperelastic material properties.

**MATHED** Damage model properties for hyperelastic materials. (SOL 600)

**MATNLE** The MATNLEx entries specify advanced forms of nonlinear elastic materials. (SOL 600)

**MATORT** Elastic 3D orthotropic material properties. Also maybe used in all linear solutions between SOL101 and 112 in conjunction with PCOMPLS.

**MATTEP** Thermoelastic-Plastic material properties.

**MATTF** Material Failure Model Temperature Variation

**MATTG** Temperature variation of interlaminar materials.

**MATTHE** Thermo hyperelastic material.

**MATTORT** Thermoelastic orthotropic material

**MATS3** Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions. SOL 400 only.

**MATS8** Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements SOL 400 only.

**MATSMA** Material properties for shape memory alloys.

**MATSORT** Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials. SOL 400)

**MATTVE** Thermo-visco-elastic material properties

**MATVE** Viscoelastic material properties

**MATVP** Viscoplastic or creep material properties

MCOHE	Cohesive materials (SOL 400)
MDELAM	Defines materials for which delamination may occur.

### Non-Structural Mass Distribution Selection

NSM	Non Structural Mass entry by property or element ID, value.
NSMI	Non Structural Mass entry by property or element ID, value.
NSMADD	Non Structural Mass as sum of listed sets.
NSML	Lumped non structural mass entry by property or element ID, value.
NSML1	Lumped non structural mass entry by property or element ID, value.

### OpenFSI (SOL 400)

FSICTRL	Defines the analysis type for OpenFSI fluid structure analysis simulations.
WETELME	Defines a wetted element used in OpenFSI fluid structure interaction simulations.
WETELMG	Defines a wetted element used in OpenFSI fluid structure interaction simulations.
WETLOAD	Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations.
WETSURF	Defines a wetted surface used in OpenFSI fluid structure interaction simulations.

### Output Control

BOUTPUT	Defines secondary nodes at which output is requested.
CBARAO	Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output.
ECHOOFF	Marks the point or points in the input file to deactivate printed echo of the Bulk Data.
ECHOON	Marks the point or points in the input file to activate printed echo of the Bulk Data.
FREEPT	Surface point location for data recovery in hydroelastic problems.
MARPRN	Defines “print” options for SOL 600.
MLAYOUT	Selects layered composite shell output to be placed in Marc’s t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. (SOL 600)
PLOTEL	Defines a one-dimensional dummy element for use in plotting.
POINTAX	Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested.
PRESPT	Defines the location of pressure points in the fluid for recovery of pressure data in hydroelastic problems.
SET1	Defines a set of grid points.

- TSTEP** Specifies time step intervals for data recovery in transient response.
- TSTEPNL** Specifies time step intervals for data recovery in nonlinear transient response.

### Monitor Points

- AECOMP** Defines a component for use in aeroelastic monitor point definition or external splines.
- AECOMPL** Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.
- MONPNT1** Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.
- MONPNT2** Element Monitor Output Results Item.
- MONPNT3** Sums select Grid Point Forces to a user chosen monitor point.
- MONCNC** Stripwise aerodynamic lift and pitching moment coefficients.
- M**
- MONSUM** Linear combination of monitor point components.
- MONSUM1** Linear combination of two or more monitor points.
- MONSUMT** Linear combination of two or more monitor points with moment transfer.

### Parameters

- CAMPBLL** Specifies the parameters for Campbell diagram generation.
- PARAM** Specifies values for parameters used in solution sequences or user-written DMAP programs.
- MDLPRM** Specifies parameters which affect the solution of the structural model.

### Solution Control

- ITER** Defines options for the iterative solver in SOLs 101, 106, 108, 111 and 153.
- RVDOF** Degrees-of-freedom specification for residual vector computations.
- RVDOF1** Degrees-of-freedom specification for residual vector computations (alternate form).

### Tabular Input

- DTI** Defines table data blocks.
- DTI,ESTDATA** Provides override data for time and space estimation for superelement processing operations.
- DTI,INDTA** Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.
- DTI,SETREE** Defines a superelement tree that determines the superelement processing order.
- DTI,SPECSEL** Correlates spectra lines specified on TABLED1 entries with damping values.

DTI,SPSEL	Correlates output requests with frequency and damping ranges.
DTI,UNITS	Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.
TABDMP1	Defines modal damping as a tabular function of natural frequency.
TABLED1	Tabular functions for generating dynamic loads.
TABLEL1	Specifies a table of amplitude vs pseudo-time for static loads specified in LDFTABL entries SOL 600 only.
TABLEM1	Tabular functions for generating temperature-dependent material properties.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).
TABL3D	Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. (SOLs 400/600).
TABLE3D	Specify a function of three variables for the CORD3G entry only.
TABLFTG	Defines cyclic load variation for use in fatigue analysis.
TABLRPC	Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

### 3D Contact Region

BCBMRAD	Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOLs 400/600)
BCBOX	Defines a 3D contact region.
BCHANGE	Changes definitions of contact bodies.
BCMATL	Defines a 3D contact region by element material.
BCMOVE	Defines movement of bodies in contact.
BCPARA	Defines contact parameters.
BCPROP	Defines a 3D contact region by element properties.
BSURF	Defines a contact body or surface by element IDs.
GMNURB	3D contact region made up of NURBS.

#### Old Contact Format

BCTABLE	Defines a contact table.
BCBODY	Defines a flexible rigid contact body in 2D or 3D.

#### New Contact Format

BCTABL1	Defines a contact table.
---------	--------------------------

BCONNECT	Defines the Touching and Touched Contact Bodies
BCONPRG	Geometric Contact Parameters of Touching Bodies
BCONPRP	Physical Contact Parameters of Touching Bodies in SOL 101 and 400
BCBODY1	Flexible or Rigid Contact Body in 2D and 3D
BCBDPRP	Contact Body Parameters in SOLs 101 and 400
BCRIGID	Defines a Rigid Contact Body in SOLs 101 and 400
BCRGSRF	Rigid Contact Surface List in SOLs 101 and 400
BCPATCH	Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400
BCBZIER	Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400
BCNURB2	Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400
BCNURBS	Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400
BCTRIM	Defines the Geometry of a Trimming Curve

### User-Defined Subroutines and Services

BCONUDS	Allows the user to provide contact routines for use with enhanced MSC Nastran contact analysis. (SOL 600 and SOL 400)
GENUDS	User Data for Notify User Defined Service or Subroutine
MATUDS	Allows the user to provide material routines for use with enhanced MSC Nastran material models. (SOL 600 and SOL 400)
MATTUSR	Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.
MATUSR	Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only.
UDSESV	Define the number and names of user state variables for material user subroutines (SOL 400)

## Nastran Implicit Nonlinear (SOL 600)

### 3D Contact Region (SOL 600)

BCBMRAD	Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section. (SOL 600)
SANGLE	Defines automatic analytical contact threshold angle for multiple subcases.
UNGLUE	Defines grids that should be eliminated from glued contact for SOL 600 and SOL 400.
WEAR	Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.

### Analysis Termination Options (SOL 600)

**TERMIN** Control to terminate a SOL 600 analysis under certain conditions

### Bolts (SOL 600)

**MBOLT** Defines a bolt for use in countries outside the USA.

**MBOLTUS** Defines a bolt for use in the USA and all other countries.

### Brake Squeal (SOL 600)

**BRKSQL** Specifies data for brake squeal calculations using SOL 600.

### Composite Integration Options (SOL 600)

**MIXTURE** Defines constituents of “composite” material on original and potentially damaged state.

**PCOMP** Defines the integration procedure for through the thickness integration of composite shells.

### Creep Analysis (SOL 600)

**MACREEP** Controls a transient creep analysis.

**MPCREEP** Specifies input values for Marc’s creep parameter when creep analysis is performed using SOL 600.

**MTCREEP** Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry.

### Element Birth and Death (SOL 600)

**ACTIVAT** This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase.

**DEACTEL** This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase.

### Elements (SOL 600)

**ALIASM** Allows selected elements which normally use a default formulation to be aliased to a different formulation.

**CSSL** Defines a connection for a Solid Shell with 6 or 8 grid points.

**CSSLH** Defines conversion of CHEXA elements to Solid Shell elements.

**CSSLM** Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements.

- CSSHLP** Defines conversion of CPENTA elements to Solid Shell elements.
- MISLAND** Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value.

#### Element Properties (SOL 600)

- NTHICK** Defines nodal thickness values for beams, plates, and/or shells.
- PSSHL** Defines the properties for Solid Shell (CSSHL) elements.

#### Fatigue, Fracture and Crack Propagation (SOL 600)

- LORENZI** This option gives an estimation of the J-Integral for a crack configuration using the domain integration method.
- VCCT** Virtual crack closure technique

#### General Tables (SOL 600)

- MTABRV** Defines a list of tables to reverse positive and negative values and/or add points at the lower and upper end of tables.
- TABD1MD** Defines how TABLED1 entries are internally modified in SOL 600.
- TABL3D** Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc.

#### Heat Transfer (SOL 600)

- MPHEAT** Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.
- MOTHERM** Iteration control for automatic thermal loading for structural analysis following a heat transfer analysis.
- NLHEATC** Defines numerical analysis parameters for SOL 600 heat transfer analysis

#### Inertia Relief (SOL 600)

- SUPPORT6** Inertia relief used in Nastran Implicit Nonlinear (SOL 600 only).

#### Initial Conditions (SOL 600)

- IPSTRN** Defines initial plastic strain values.
- ISTRESS** Defines initial stress values.

#### Input/Output (SOL 600)

- MARCIN** Inserts a text string in Marc.
- MARCOUT** Selects data recovery output.

### Matrix Input/Output (SOL 600)

DMIGOUT	Defines DMIG matrices to be output from the Marc Portion of SOL 600.
DMIGROT	Defines large rotation and other characteristics of a matrix entered using DMIG..
MDMIAUX	Specifies the DOMAINSOLVER command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only.
MDMIOUT	Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600.
MESUPER	Defines external superelement DMIG input for SOL 600 residual analyses
MNF600	Defines auxiliary data for MSC Adams MNF files.

### Solid Composites (SOL 600)

MSTACK	Defines the direction in which 3D solid composites are stacked.
--------	---

### Solution Control (SOL 600)

MPROCS	Provides additional control for parallel processing
NLAUTO	Parameters for automatic load/time stepping.
NLHEATC	Defines Numerical Analysis Parameters for Heat Transfer Analysis.
NLSTRAT	Strategy parameters for nonlinear structural analysis.
PARAMARC	Parallel domain decomposition.
RESTART	Restart data.

### Structural Analysis Following a Heat Transfer Analysis (SOL 600)

MINSTAT	This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file.
---------	---

### t16 Output Control (SOL 600)

MT16SEL	Limits elements and/or grid results to selected elements or grids for t16 and t19 file results.
MT16SPL	Determines how to split a Marc t16 file into one or more smaller t16 files.

### User Subroutine Control (SOL 600)

USRSUB6	Defines user subroutines used in SOL 600 only.
---------	--

## Explicit Nonlinear (SOL 700)

### Air Bags (SOL 700)

GBAG	Defines the pressure within an enclosed volume.
------	---



**ATB Models (SOL 700)**

ATBACC	Defines an acceleration field that will be applied to ATB segments.
ATBJNT	This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor.
ATBSEG	Defines the position and orientation of the ATB segments.
RELEX	Defines a rigid ellipsoid whose properties and motion are defined by either ATB.

**Contact (SOL 700)**

BCBODY	Defines a flexible or rigid contact body in 2D or 3D.
BCBODY1	Defines a flexible or rigid contact body in 2D or 3D.
BCBOX	Defines a 3D contact region - all elements within the region define a contact body.
BCELIPS	Defines a list of ellipsoid names (character strings) for use of contact analysis.
BCGRID (SOL 700 only)	Grids to be included in contact analyses.
BCMATL	Defines a 3D contact region by element material. All elements with the specified materials define a contact body.
BCONNECT	Defines the touching and touched contact bodies used for general contact.
BCONPRG-700	Defines geometric contact parameters of touching bodies.
BCONPRP-700	Defines physical contact parameters of touching bodies.
BCSEG	Grids which are part of an element to be used in contact analyses.
BCTABLE	Defines a contact table (old format).
BCTABL1	Defines a contact table (new format).
BSURF	Defines a contact body or surface by Element IDs.

**Coordinate Systems (SOL 700)**

CORD3R	Defines a moving rectangular coordinate system using three points (SOL 700 only).
--------	---

**Dampers/Springs (SOL 700)**

CDAMP1D	Defines a scalar damper connection for use in SOL 700 only.
CDAMP2D	Defines a scalar damper connection for use in SOL 700 only.
CELAS1D	Defines a scalar spring connection for use in SOL 700 only.
CELAS2D	Defines a scalar spring connection for use in SOL 700 only.

### Direct Text Input (SOL 700)

ENDDYNA	Defines the end of direct text to Dytran.
TODYNA	Defines the start of direct text to Dytran.

### Element Properties (SOL 700)

PBELT	Defines the properties of a belt element referenced by a CROD entry.
PELASI	Defines a spring property designated by a force-deflection curve for SOL 700.
PSHELL1	Defines the properties of SOL 700 shell elements that are much more complicated than the shell elements defined using the PSHELL entry.
PVISC1	Defines the properties of a nonlinear damper where the damping constant varies with the velocity.

### Eulerian Boundary (SOL 700)

BARRIER	Defines a barrier for transport in a Eulerian mesh.
FLOW	Defines the properties of a material for the boundaries of an Eulerian mesh.
FLOWC	Defines the properties of a material for the boundaries of a Eulerian mesh.
FLOWDEF	Definition of default Eulerian flow boundary condition.
FLOWT	Defines the material properties for the in- or outflow of material through the boundary of an Euler mesh.
HYDSTAT	Initializes the Euler element densities in accordance to a hydrostatic pressure profile.

### Eulerian Initial Conditions (SOL 700)

CYLINDR	Cylindrical shape used in the initial condition definition on the TICEUL entry.
SPHERE	Spherical shape used in the initial condition definition on the TICEUL entry.
SURFINI	Defines a surface that is used for initialization of regions of an Eulerian mesh.
TICEL	Defines the initial values of element variables at the beginning of the analysis.
TICEUL1	Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes.
TICREG	Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes.
TICVAL	Defines the initial values of an Eulerian geometric region.

**Eulerian Materials (SOL 700)**

EOSDEF	EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas.
EOSGAM	Defines the properties of a Gamma Law equation of state where the pressure $p$ is defined.
EOSIG	Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives.
EOSJWL	Defines the properties of a JWL equation of state commonly used to calculate the pressure of the detonation products of high explosives.
EOSMG	Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure $p$ in high strain rate processes.
EOSNA	Defines the properties of Noble-Abel equation of state.
EOSPOL	Defines the properties of a polynomial equation of state where the pressure $p$ is defined.
FAILJC	Defines the properties of the Johnson-Cook failure model.
FAILMPS	Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value.
MATDEUL	Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame.
PMINC	Defines a spallation model where the minimum pressure is constant.
SHREL	Defines an elastic shear model with a constant shear modulus.
SHRPOL	Defines an elastic shear model with a polynomial shear modulus.
YLDHY	Defines a yield model with zero yield stress.
YLDJC	Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature.
YLDMSS	Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL.
YLDPOL	Defines a polynomial yield model where the yield stress is a function of effective plastic strain.
YLDRPL	Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate.
YLD SG	Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature.
YLD TM	Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.

YLDVM	Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion.
YLDZA	Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature.

### Eulerian Solid Elements (SOL 700)

PEULER	Defines the properties of Eulerian element.
PEULER1	Eulerian element properties.

### Euler/Lagrange Coupling (SOL 700)

ABINFL	Defines an inflator model suited for airbag analyses.
COUCOHF	Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface.
COHFRIC	Allows friction and sticking during tensile conditions at the coupling surface.
COUOPT	Defines the interaction factor and a pressure load from the covered side acting on a BSURE.
COUPIFL	Defines the surrounding variables when a segment of a coupling surface fails.
COUPINT	Defines the surrounding variables when a segment of a coupling surface fails.
COUPLE	Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain.
INFLCG	Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface.
INFLFRC	Defines the gas fractions as a function of time for hybrid inflators.
INFLGAS	Defines a thermally ideal gas to be used with a standard or hybrid inflator.
INFLHB	Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface.
INFLTNK	Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface.
INFLTR	Defines the inflator characteristics of a COUPLE and/or GBAG subsurface.
INITGAS	Specifies the initial gas composition inside a gasbag or Euler coupling surface.
LEAKAGE	Defines the porosity model to be used with GBAG or COUPLE.
PERMEAB	Defines the permeability of a COUPLE and/or GBAG (sub)surface.
PERMGBG	Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.
PORFCPL	Defines an interaction between two coupling surfaces through a hole.
PORFLOW	Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface.

PORFLWT	Defines a time dependent flow trough a porous area of the couple surface.
PORFGBG	Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG.
PORHOLE	Defines a hole in a COUPLE and/or GBAG surface.
PORHYDS	Prescribes a hydrostatic pressure profile on a porous BSURE.

### Hourglass Control (SOL 700)

HGSUPPR	Defines the hourglass suppression method and the corresponding hourglass damping coefficients.
---------	--

### Initial Conditions (SOL 700)

TIC3	Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in Explicit Nonlinear (SOL 700) only.
------	---

### Materials (SOL 700)

MAT1	Defines the material properties for linear isotropic materials.
MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MATBV	Defines the bulk viscosity for materials.
MATEP	Elasto-plastic material properties for SOL 700 only.
MATF	Specifies failure model properties.
MATFAB	Defines the properties of a bi-directional woven fabric material for shell elements.
MATHE	Specifies hyperelastic (rubber-like) material properties for nonlinear analysis.
MATORT	Specifies elastic orthotropic material properties for 3-dimensional and plane strain and shell behavior for linear and nonlinear analyses.
MATRIG	Defines the properties of a rigid body.
MATVE	Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis

### Miscellaneous (SOL 700)

BIAS	Specifies a variation of the mesh-size in one direction for use in the MESH entry.
CMARKB2	Defines a 2-noded marker beam element by means of connecting two grid points.
CMARKN1	Defines a 1-noded marker element on a grid point.
DETSPPH	Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials.

DYFSISW	Allows activating or deactivating Fluid Structure Interaction and Eulerian solver.
EULFOR	Defines a body force loading (acceleration) on Euler elements per unit mass.
EULFOR1	Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.
EULFREG	Defines the acceleration field for sets of Eulerian regions, The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-dependent acceleration can be defined.
FFCONTR	Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling.
MESH	Defines a mesh.
PMARKER	Defines the behavior of the marker element in the FV domain.

#### Multi-Variable Parameters (SOL 700)

DYPARAM, ATBAOUT	Defines the frequency at which output is written to the main output file of ATB.
DYPARAM, ATBHOUT	A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11.
DYPARAM, ATBTOUT	Defines the frequency at which output is written to the time-history files of ATB.
DYPARAM,AUTOCOUP	Defines the automatic coupling algorithm.
DYPARAM,AXIALSYM	Axial symmetric analysis.
DYPARAM,AXREMAP	Allows import of a 2D axial symmetric Euler archive into a 3D simulation.
DYPARAM,BULKL	Defines the default value of the linear bulk viscosity coefficient.
DYPARAM,BULKQ	Defines the default value of the quadratic bulk viscosity coefficient.
DYPARAM,BULKTYP	Defines the default type of bulk viscosity.
DYPARAM,CFULLRIG	Converts all 123456 constraints to the FULLRIG option on all entries.
DYPARAM,CLUFLIM	This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable.
DYPARAM,CLUMPENR	Sets the definition of the kinetic energy calculation method for Eulerian blended clumps.
DYPARAM,COHESION	Cohesion for Coulomb Friction

DYPARAM,CONM2OUT	Determines if a summary of concentrated masses and their energy and momentum is written to the output file.
DYPARAM,CONTACT	Defines certain defaults for the contact definitions.
DYPARAM,COSUBMXT	Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table.
DYPARAM,COUFRIC	Defines the Coulomb friction scheme.
DYPARAM,ELDLTH	Print initial time step sizes for elements in the first cycle.
DYPARAM,EULERCB	Divides a Euler domain into several cubes.
DYPARAM,EULTRAN	Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver.
DYPARAM,EUSUBCYC	Controls the growth of the subcycling interval in Euler computations.
DYPARAM,EUSUBMAX	Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped.
DYPARAM,FAILDT	Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit.
DYPARAM,FAILOUT	Defines whether failed elements are written to the output file (ARCHIVES).
DYPARAM,FASTCOUP	Defines the fast coupling algorithm.
DYPARAM,FLOWMETH	Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces.
DYPARAM,GEOCHECK	This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed.
DYPARAM,HGCMEM	Defines the default membrane damping coefficient for shell elements.
DYPARAM,HGCOEFF	Defines the global default hourglass damping coefficient.
DYPARAM,HGCSOL	Define the default damping coefficient for solid elements.
DYPARAM,HGCTWS	Defines the default twisting damping coefficient for shell elements.
DYPARAM,HGCWRP	Defines the default warping damping coefficient for shell elements.
DYPARAM,HGSHELL	Defines the default hourglass suppression method for shell elements.

DYPARAM,HGSOLID	Defines the default hourglass suppression method for solid elements.
DYPARAM,HGTYPE	Defines the default type of hourglass suppression method.
DYPARAM,HICGRAV	Defines the value of the gravity to be used by the HIC calculations.
DYPARAM,HVLFAIL	Defines element failure on the hydrodynamic volume limit
DYPARAM,HYDROBOD	Defines a body force for single hydro material in Euler.
DYPARAM,IMM	The option allows to specify the IMM method to be used.
DYPARAM,INFOBJ	Additional information about the BJOIN and spotweld connectivity will be listed in the output file.
DYPARAM,INISTEP	Defines the time step used at the start of the analysis.
DYPARAM,JWLDET	Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries.
DYPARAM,LIMCUB	Defines the maximum number of cubes used to sort the grid points in a contact definition.
DYPARAM,LIMITER	Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe.
DYPARAM,MATRMERG	Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly.
DYPARAM,MATRMRG1	Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties.
DYPARAM,MAXSTEP	Defines the maximum allowable time step.
DYPARAM,MESHPLN	Defines Mesh density for covering rigid planes.
DYPARAM,MINSTEP	Defines the minimum time step that causes the analysis to terminate.
DYPARAM,MIXGAS	Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature.
DYPARAM,NZEROVEL	Set the velocity of a node to zero in case all attached elements have failed.
DYPARAM,OLDLAGT	Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements.
DYPARAM,PARALLEL	The option allows you to gather information on the parallel section.
DYPARAM,PLCOVCUT	Defines time when PLCOVER is cut off.
DYPARAM,PMINFAIL	Defines Lagrangian solid element failure on reaching the spall limit.



DYPARAM,RBE2INFO	The grid points attached to MATRIG and RBE2assemblies are listed to the output file.
DYPARAM,RHOCUT	Defines the minimum density for all Eulerian elements.
DYPARAM,RJSTIFF	Defines the stiffness of a rigid joint.
DYPARAM,SHELLFRM	Sets the default for the shell formulation for quadrilateral elements.
DYPARAM,SHELMSYS	Defines the shell element system for the BLT shells.
DYPARAM,SHPLAST	Specifies the type of calculation used to determine the plane stress plasticity method for shells.
DYPARAM,SHSTRDEF	Specifies the default coordinate system for the stress and strain output of composite shells.
DYPARAM,SHTHICK	Specifies whether or not the thickness of the shell changes with membrane straining.
DYPARAM,SLELM	Defines whether shell sublayer variables are to be stored in the element arrays.
DYPARAM,SMP,BATCHSIZ	Define batch size and number of CPU loops, per entity type.
DYPARAM,SMP,CPUINFO	Define the CPU information per entity type.
DYPARAM,SNDLIM	Defines the minimum value for the speed of sound.
DYPARAM,SPHERSYM	Enables an efficient and accurate 1D spherical symmetric solution for Euleran materials
DYPARAM,STRNOUT	Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output.
DYPARAM,TOLCHK	To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK, write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14.
DYPARAM,VDAMP	Controls the global damping in the dynamic relaxation.
DYPARAM,VELMAX	Defines the maximum velocity in Eulerian meshes.

DYPARAM,VELMAX1	Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes
DYPARAM,VISCPLAS	Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material.

### Rigid Elements (SOL 700)

BJOIN	Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis.
RBJOINT	Defines a joint between two rigid bodies.
WALL	Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Used in Explicit Nonlinear (SOL 700) only.

### Time Step Control (SOL 700)

TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis.
---------	--

### User Defined Subroutines (SOL 700)

COMPUDS	Defines an orthotropic failure model for shell composites specified by a user subroutine.
EOSUDS	Defines the frequency dependent properties for an isotropic poroelastic material.
FAILUDS	User defined simple failure of Eulerian materials.
FLOWUDS	Defines a flow boundary on an Eulerian mesh specified by a user subroutine.
FORCUDS	Defines enforced motion at grid points specified by a user subroutine.
NLOUTUD	User defined output requests for elements or Lagrangian grid points.
PORUDS	Defines a porosity model of a COUPLE surface through a user-written subroutine.
SHRUDS	Specifies that a user subroutine is being used to define the shear modulus.
TABLUDS	Specifies that a user routine is being used to define an arbitrary function.
TICEUDS	User defined simple failure of Eulerian materials.
YLDUDS	Specifies that a user subroutine is being used to define a simple yield model.

## Solution Control

### Aerodynamic Matrix Generation

- MKAERO1** Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
- MKAERO2** Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

### Aerodynamic Parameters

- AERO** Gives basic aerodynamic parameters for unsteady aerodynamics.
- AEROS** Defines basic parameters for static aeroelasticity.

### Aeroelastic Response Analysis

- AEDW** Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG).
- AEFORCE** Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector.
- AEPARM** Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point).
- AEPRESS** Defines a vector of pressure/unit dynamic pressure associated with a particular control vector.
- AESCALE** Defines reference lengths to scale aerodynamic grid points.
- GUST** Defines a stationary vertical gust for use in aeroelastic response analysis.
- TABRNDG** Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.
- TRIM** Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.
- UXVEC** Specification of a vector of aerodynamic control point values.

### Aeroelastic Stability Analysis

- DIVERG** Defines Mach numbers (m) for a static aeroelastic divergence analysis.
- FLFACT** Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
- FLUTTER** Defines data needed to perform flutter analysis.

### Buckling Analysis

- EIGB** Defines data needed to perform buckling analysis.
- EIGRL** Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

## Cyclic Symmetry

**CYSYM** Defines parameters for cyclic symmetry analysis.

## Eigenvalue Analysis

**EIGC** Defines data needed to perform complex eigenvalue analysis.

**EIGR** Defines data needed to perform real eigenvalue analysis.

**EIGRL** Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

**EIGP** Defines poles that are used in complex eigenvalue extraction by the Determinant method.

**RVDOF,**  
**RVDOF1** Degrees-of-freedom specification for residual vector calculation.

## Monte-Carlo simulation

**MONCARL** Defines the parameters for Monte-Carlo simulations.

## Fatigue Analysis

**DTI,UNITS** Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface and Nastran fatigue analysis.

**FTGDEF** Defines elements and their associated fatigue properties to be considered for fatigue analysis.

**FTGPARM** Defines parameters for a fatigue analysis

**FTGSEQ** Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or random vibration fatigue analysis using SOL 108 or SOL 111.

**FTGEVNT** Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111 by referencing FTGLOAD entries.

**FTGLOAD** Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or random vibration fatigue analysis using SOL 108 or SOL 111.

**MATFTG** Defines fatigue material properties.

**PFTG** Defines fatigue elemental properties.

**RANDPS** Defines PSDs and cross PSDs for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.

**TABLFTG** Defines tabular data for specifying fatigue cyclic loading variation for time domain fatigue analysis using SOL 101 or SOL 103.

TABLRPC	Tabular functions for generating dynamic loads by reading the tabular data from an external channel data file.
TABRND1	Defines tabular data for specifying PSD random vibration loading for frequency domain fatigue (random vibration) analysis using SOL 108 or SOL 111.
TIM2PSD	Defines parameters and triggers conversion of time history channel data into corresponding Power Spectral Density (PSD) functions for use in random vibration fatigue analysis using SOL 108 or SOL 111.
UDNAME	Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD.

### Frequency Response

FREQ	Defines a set of frequencies to be used in the solution of frequency response problems.
FREQ1	Defines a set of frequencies for problem solution.
TABDMP1	Defines modal damping as a tabular function of natural frequency.

### Nonlinear Static Analysis

ITER	Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153, 200 and 400.
NLADAPT	Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400.
NLPARAM	Defines a set of parameters for nonlinear static analysis iteration strategy.
NLPCI	Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106).
NLSTEP	Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101.

### Optimization (SOL 200 Only)

BEADVAR	Defines design region for topography (bead or stamp) optimization.
BNDGRID	Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).
DCONADD	Defines the design constraints for a subcase as a union of DCONSTR entries.
DCONSTR	Defines design constraints.
DDVAL	Define real, discrete design variable values for discrete variable optimization.
DEQATN	Defines a design variable for design optimization.
DESVAR	Defines a design variable for design optimization.
DLINK	Relates one design variable to one or more other design variables.
DOPTPRM	Overrides default values of parameters used in design optimization.

DRESP1	Defines a set of structural responses that is used in the design either as constraints or as an objective.
DRESP2	Defines equation responses that are used in the design, either as constraints or as an objective.
DRESP3	Defines an external response using user-supplied routine.
DSCREEN	Defines screening data for constraint deletion.
DTABLE	Defines a table of real constants that are used in equations (see DEQATN entry).
DTABLE2	Defines real constants from a field of property, material or connections bulk data entries which can then be invoked by a DVxREL2, DRESP2, or DRESP3 entry.
DVBSHAP	Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.
DVCREL1	Defines the relation between a connectivity property and design variables.
DVCREL2	Defines the relation between a connectivity property and design variables with a user-supplied equation.
DVGRID	Defines the relationship between design variables and grid point locations.
DVLR1	Defines the linear relation between analysis model loading and design variables
DVMREL1	Defines the relation between a material property and design variables.
DVMREL2	Defines the relation between a material property and design variables with a user-supplied equation.
DVPREL1	Defines the relation between an analysis model property and design variables.
DVPREL2	Defines the relation between an analysis model property and design variables with a user-supplied equation.
DVPSURF	Design Variable to Control Surface Setting Relation
DVSHAP	Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.
MODTRAK	Specifies parameters for mode tracking in design optimization (SOL 200).
SEDLINK	Multiple Design Variable Linking Across PART SE Boundary
SEDRSP2	Design Sensitivity Equation Response Quantities for PART SE
SEDRSP3	Defines External Response with User-Supplied Routines
STOCHAS	Randomization of Model Parameters
TOMVAR	Defines a design region for topometry optimization (element-by-element optimization).
TOPVAR	Defines a topology design region for topology optimization.

## Random Response

<a href="#">RANDPS</a>	Defines load set power spectral density factors for use in random analysis.
<a href="#">RANDT1</a>	Defines time lag constants for use in random analysis autocorrelation function calculation.
<a href="#">RCROSS</a>	Cross-power spectral density and cross-correlation function output.
<a href="#">TABRND1</a>	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

## Rotordynamics

<a href="#">RGYRO</a>	Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.
<a href="#">ROTBENT</a>	Define kinks and offsets to be applied on a rotor, pull the rotor on to its bearing and then carry out rotordynamic analysis (SOL 400 only).
<a href="#">ROTHYBD</a>	Hybrid damping for rotors
<a href="#">ROTOR</a>	Rotor Model Definition
<a href="#">ROTORAX</a>	Axisymmetric Model Rotor Definition
<a href="#">ROTORG</a>	Specifies grids that compose the rotor line model.
<a href="#">ROTORSE</a>	Specifies grids that compose the rotor line model.
<a href="#">RSPINR</a>	Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis.
<a href="#">RSPINT</a>	Specifies rotor spin rates for nonlinear transient analysis.
<a href="#">UNBALNC</a>	Specifies an unbalanced load for transient analysis in terms of a cylindrical system with the rotor rotation axis as the z-axis.

## Transient Response

<a href="#">TIC</a>	Defines values for the initial conditions of variables used in structural transient analysis.
<a href="#">TSTEP</a>	Defines time step intervals at which a solution will be generated and output in transient analysis.
<a href="#">TSTEPNL</a>	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. Is intended for SOLs 129, 159, and 99.

## Entries A - B

\$ Comment

Used to insert comments into the input file. Comment statements may appear anywhere within the input file.

Format:

\$ followed by any characters out to column 80.

Example:

```
$ TEST FIXTURE-THIRD MODE
```

Remarks:

1. Comments are ignored by the program.
2. Comments will appear only in the unsorted echo of the Bulk Data.
3. For free field entries (See Format of Bulk Data Entries above), a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank.
4. For the default NASTRAN SYSTEM (IFPSTAR) = YES, input using FIXED field, a \$ can be used to terminate the field entries and any required remaining fields will be modulo replicated as blank. The OLD IFP (NASTRAN SYSTEM (IFPSTAR) = NO) will fail with this option.



/ Control Input Stream

Allows the user control over field width, and on restart entry deletion, insert and update, and process control. (This entry's execution is dependent on whether NASTRAN SYSTEM(444)= 0 or 1.)

Format:

1	2	3	4	5	6	7	8	9	10
/	Alpha numeric text as described below								

Examples:

/	0	12
/delete all		
/ Delete FORCE1,PARAM MATVP		

Describer	Meaning
/	Must be first character and occur somewhere in fields one through 8. (Character)
<i>If NASTRAN SYSTEM(444)=0, then the n1, n2 directive below is the only valid form:</i>	
n1,n2	Two integers indicating following number of lines of previous input are to be deleted. (Integer. n1 ≥ 10, n2 > n1). /, n1,n2 directive must be entered before any new data is added to the input stream.
<i>If NASTRAN SYSTEM(444)=1, then the above n1, n2 directive will not work and only the following forms are valid:</i>	
DELETE	Keyword indicating following listed items are to be deleted. (Character) /DELETE directive must be entered before any new data is added to the input stream.
ALL	Keyword indicating following a /DELETE directive. All previous input items will be deleted on RESTART. (Character)
List of names	A list of Bulk Data names to be deleted following a /DELETE directive. All previous input Bulk entries with the names indicated will be deleted on RESTART. (Character)
SFW	A Keyword that is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9. See Remark 1. (Character)
n	Single Field Width value n used with SFW. This is used for fixed field entries and allows the user to define the field width of single field entries for Field 2 thru Field 9 to the value n.(Integer 8 ≤ n ≤ 14, Default = 8). If double field is selected by use of the * option, then Field 2 thru Field 9 are of 2n width. (Field 1 and Field 10 always remain of width 8). See Remark 1.

Describer	Meaning
SKIPON	Input data following this keyword would not be processed in the current execution. (Character). This command is terminated by one of three methods: 1) The occurrence of a SKIPOFF. 2) The occurrence of a BEGIN (Such as BEGIN SUPER=10) 3) The occurrence of an ENDDATA.
SKIPOFF	Keyword turning off the SKIPON command. (Character)
INSERT	Keyword that implies that entries when placed into the input stream must not be previously present. See Remark 1. (Character; Default for INITIAL runs.)
UPDATE	Keyword that implies that entries when placed into the input stream are new or are to replace existing entries. See Remark 1. (Character; Default for RESTART runs.)
DUPMETH	Keyword that allows reversion to the original IFP, DUPTOL tolerance selection. /DUPMETH=OLD selects original IFP tolerance selection. See the DPBLKTOL(402) system cell description and Remark 1. (Character; Default NEW for IFPSTAR tolerance selection).
DUPTOL	Keyword that allows a closeness tolerance value when searching for removal of duplicate entries. (Character)
v	Tolerance value, used with DUPTOL, when searching for removal of duplicate entries. See Remark 1. (Real $\geq 0.$ ; Default 1.0-5.)
LISTON	Keyword to indicate a "list of data to follow" See Remark 10. (Character)
LISTOFF	Keyword to indicate a "list of data is terminated" See Remark 10. (Character)

#### Remarks:

1. If the SFW, INSERT, UPDATE, DUPMETH, DUPTOL Keywords and any associated value (n, v) occur in the first BEGIN BULK Section the values set will carry forward to any other BEGIN section included. If a user wishes to have some later BEGIN have different control values, then the BEGIN BULK values can be overridden for that particular BEGIN Section by the inclusion of / entries particular to that specific BEGIN Section. The DUPMETH must come before the DUPTOL entry. Currently IFPSTAR will not recognize the settings from the command line or RC file due to default usage issues at the application state and real value passing through the component level. When DUPMETH is OLD and DPBLKTOL or DUPTOL is zero or positive, remove grids with duplicated id without checking CP, CD, PS ,SE fields and distance tolerance.

#### Example:

```

/DUPMETH=OLD
/DUPTOL=0.1
With the following NASTRAN entry required:
NASTRAN SYSTEM(402)=0.1

```

2. The input stream into Nastran is now part of a Common Data Model component and entries are committed to a data base as soon as they are interpreted in a single unsorted pass. When the user requests a sorted ECHO, the ECHO represents a binary value converted back into a column character location. It does not represent the actual binary entry value! Unsorted ECHO respects the request list of the ECHO command.
3. If an improperly formatted bulk data entry is encountered, the true file line number of the error is printed along with reference to the Parent entry. If the offending entry is in an include file the file is identified.
4. Usage of the continuation mnemonic is no longer needed and its usage is not recommended.
5. Default Fixed Fields are 80 characters (see SFW). Free field supports 128 characters.
6. Because the Common Data Model component entries are committed to a data base as soon as they are interpreted in a single unsorted pass, the /DELETE syntax is now required to be used on restart.
7. If an /INSERT is present on RESTART the Common Data Model component will signal an error condition EXCEPT when certain objects such as GRID entries have been classified as Remove Duplicate or when objects such as SPC that allow superposition are encountered.
8. Entries that allow superposition are always in /INSERT.
9. If NASTRAN SYSTEM(444)=1, then the Nastran Replication feature is not recommended.
10. For convenience in entering an ordered list of elements (tuples) data or, such as table value inputs, the following is allowed for example:

```

tabled1,111
  /liston
  1.0 1.0
  2.0 1.0
  3.0 2.0
  4.0 2.0
  5.0 1.0
  6.0 1.0
  /listoff
,endt

```

or for example if user likes commas:

```

tabled1,112
  /liston
  1.0, 1.0
  2.0, 1.0
  3.0, 2.0
  4.0, 2.0

```

5.0, 1.0

6.0, 1.0

/listoff

,endt

**ABINFL**

**Inflator Model to be Used With GBAG or COUPLE Entries**

Defines an inflator model suited for airbag analyses. The inflator model is defined as part of the GBAG or COUPLE surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ABINFL	CID	INFID	SUBID	INFTYPE	INFTYPID	COEFF	COEFFV		

Example:

ABINFL	201	1	120	INFLHB	11		0.012		
--------	-----	---	-----	--------	----	--	-------	--	--

Describer	Meaning
CID	Unique number of a ABINFL entry. (Integer > 0; Required)
INFID	Number of a set of ABINFL entries NFID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)
SUBID	Number of a BSURE, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the as defined on the GBAG or COUPLE entry. (Integer > 0; Required)
INFTYPE	Defines the type of inflator. (Character; Required)
	INFLTR      The INFLTR logic is used to model inflators in an air bag.
	INFLHB      The INFLHB logic is used to model hybrid inflators in an air bag.
	INFLCG      The INFLCG logic models a cold gas inflator.
	INFLTNK     The INFLTNK logic models the inflator properties (mass flow rate and inflator gas temperature) calculated from the empirical results.
INFTYPID	Number of the entry selected under INFTYPE, for example, INFLTR,INFTYPID. (Integer > 0; Required)
COEFF	Method of defining the area coefficient. (Character, CONSTANT)
	CONSTANT    The area coefficient is constant and specified on COEFFV.
	TABLE        The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.
COEFFV	The area coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or 1 > 0)

Remarks:

1. The INFLTR or INFLHB inflator geometry and location is defined by a BSURF, BCBOX, BCPROP, BCMATL or BCSEG. The area of the hole through which the gas enters is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete subsurface area, while a value of COEFFV = 0.0 will result in a closed inflator area with no inflow.



2. This allows for setting up the exact same model for either a uniform pressure model or an Euler Coupled model. This makes it possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

## ABSNMVB

### Meta data group Boolean name/value pairs

Associate Boolean Name/Value pairs to meta data collectors (ABSTRACT)

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVB	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-	...	

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVB	50		STIFFNER	TRUE	VARSECT	FALSE			

Describer	Meaning
ABSTID	Unique identification number for meta data group (ABSTRACT). (Integer>0).
NAME <sub>i</sub>	Up to eight characters defining the name of this Name/Value pair. (Character).
VAL <sub>i</sub>	Value of this Name/Value pair -Boolean- data (Character) TRUE or FALSE.

Remarks:

1. This entry has no impact on the solution.
2. The only allowed character values are “TRUE” or “FALSE”.

## ABSNMVI

## Meta data group Integer name/value pairs

Associate Integer Name/Value pairs to meta data collectors (ABSTRCT).

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVI	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVI	50		SPANS	12					

Describer	Meaning
ABSTID	Unique identification number for meta data group (ABSTRCT). (Integer>0).
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character).
VALi	Value of this Name/Value pair (Integer).

Remarks:

1. This entry has no impact on the solution.



## ABSNMVR

### Meta data group Real Name/Value pairs

Associate Real Name/Value pairs to meta data collectors (ABSTRCT).

Format:

1	2	3	4	5	6	7	8	9	10
ABSNMVR	ABSTID		NAME1	VAL1	NAME2	VAL2	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSNMVR	50		ANGLEA	15.0	ANGLEB	45.0			

Describer	Meaning
ABSTID	Unique identification number for meta data group (ABSTRCT). (Integer>0)
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character)
VALi	Value of this Name/Value pair (Real)

Remarks;

1. This entry has no impact on the solution.

## ABSNMVS

### Meta data group String name/value pairs

Associate String Name/Value pairs with meta data collectors (ABSTRCT).

Format:

1	2	3	4	5	6	7	8	9	10	
ABSNMVS	ABSTID									
	NAME1	VAL1								
	NAME2	VAL2								
	-etc.-									

Example:

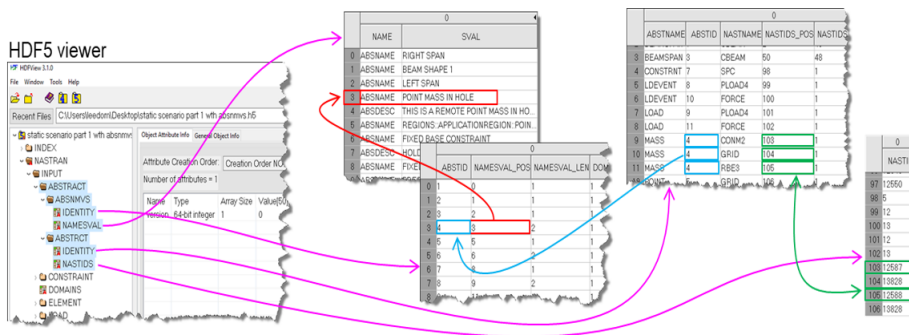
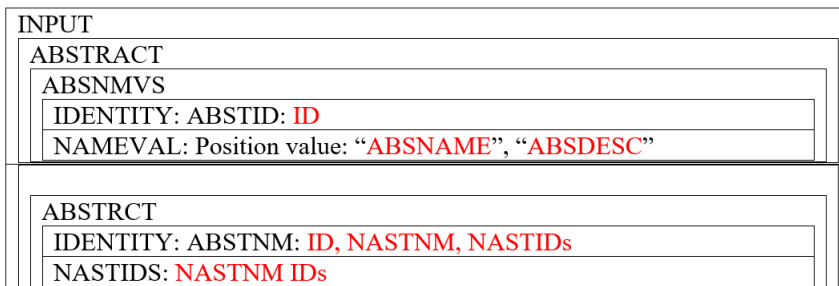
1	2	3	4	5	6	7	8	9	10	
ABSNMVS	50									
	ABSNAME	B pillar model year 2020								
	ABSDESC	B pillar including locking bushing								

Describer	Meaning
ABSTID	Unique identification number for meta data group (ABSTRCT). (Integer>0)
NAMEi	Up to eight characters defining the name of this Name/Value pair. (Character).
VALi	Value of this Name/Value pair. May consist of any Character A-Z, or numbers, and underscore or dash. Limited to 56 characters. (Character).

Remarks:

1. This entry has no impact on the solution.
2. The name/value pair entries may be specified in any order.

3. In the HDF5 viewer



4. In the above Example, the following SET3 entries might include:

```

$ CBEAM elements for B pillar
SET3, 10, ELEM, 7701, THRU, 7799
$ CBUSH elements for locking
SET3, 30, ELEM, 66, 67, 68, 69
$ RIGID ELEMENTS
SET3, 90, RBEIN, 9991, THRU, 9996
    
```

## ABSTRCT

## Meta Data Collector for HDF5 output and post processing

Allows the user to define an arbitrary group of Nastran entities (ABSTRACTion), using any Nastran keyword type including elements, grids, materials, etc. and associate a name and description for the group.

Format:

1	2	3	4	5	6	7	8	9	10
ABSTRCT	ABSTNM	ABSTID	NASTNM	NASTID1	NASTID2	NASTID3	-etc.-		

Example:

1	2	3	4	5	6	7	8	9	10
ABSTRCT	PILLAR	50	CBEAM	10					
ABSTRCT	PILLAR	50	CBUSH	30					
ABSTRCT	PILLAR	50	RBE3	90					

Describer	Meaning
ABSTNM	Up to eight characters for the name of the meta data group. (Character).
ABSTID	Unique identification number for the meta data group. (Integer>0).
NASTNM	Up to eight characters for the name of the Nastran entity being added to the meta data group. (Character).
NASTIDi	ID referencing a Nastran entity of the specified type or a set of such entities.

Remarks:

1. The group definition is transmitted from the input file to the .op2 and/or .h5 results file with no impact on the solution.
2. The group definition can subsequently be recovered from either the .op2 or .h5 file by a post-processor.
3. Associated with this entry are the entries: [ABSNMVS](#), [ABSNMVB](#), [ABSNMVI](#), [ABSNMVR](#).

## ACCEL Acceleration Load

Defines static acceleration loads, which may vary over a region of the structural model. The load variation is based upon the tabular input defined on this Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9	10
ACCEL	SID	CID	N1	N2	N3	DIR			
	LOC1	VAL1	LOC2	VAL2	Continues in Groups of 2				

Example(s):

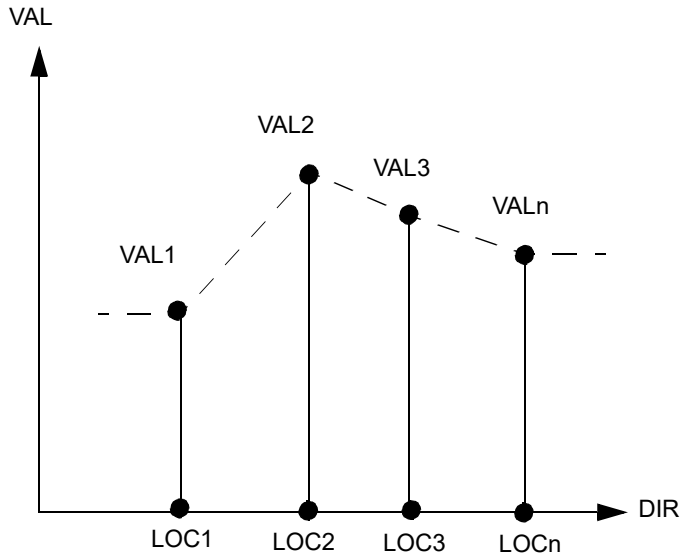
ACCEL	100	2	0.0	1.0	2.0	X			
	0.0	1.0	1000.0	3.0					

Descriptor	Meaning
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
DIR	Component direction of acceleration variation. (Character; one of X, Y, and Z)
LOCi	Location along direction DIR in coordinate system CID for specification of a load scale factor. (Real)
VALi	The load scale factor associated with location LOCi. (Real)

Remarks:

- For all grids of the model, the acceleration vector is defined by  $\vec{a} = VAL \cdot \vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\vec{a}$  is equal to VAL times the magnitude of  $\vec{N}$ . The scale factor VAL for each grid is found linearly interpolating the DIR coordinate of the grid between table values  $LOCi/VALi$ . If the GRID point coordinate in coordinate system CID is outside the range of the table, VAL is determined either from VAL1 or VALn (the last value, see the following figure).
- This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL1 loads, by specification on a LOAD entry. The SID on an ACCEL entry may not be the same as that of any other load entry.
- This acceleration load does not include effects due to mass on scalar points.
- A CID of zero references the basic coordinate system.
- The DIR field must contain one of the characters X, Y, or Z. The DIR direction defines the direction of acceleration load variation along direction 1, 2, or 3 respectively of coordinate system CID.

- A minimum of two pairs of {LOC<sub>i</sub>, VAL<sub>i</sub>} data must be specified.



Definition of Load Scale Factor vs Location

- If Modules are present then this entry may only be specified in the main Bulk Data section.
- In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOAD<sub>i</sub> or TLOAD<sub>i</sub> entry.

## ACCEL1 Acceleration Load

Defines static acceleration loads at individual GRID points.

Format:

1	2	3	4	5	6	7	8	9	10
ACCEL1	SID	CID	A	N1	N2	N3			
	GRIDID1	GRIDID2	-etc.-						

Example(s):

ACCEL1	100	2	10.0	1.0	2.0	0.0			
	1	2	3	4	THRU	10	BY	2	
	20	21	THRU	30	40	52	69	70	
	82	90	100						

Descriptor	Meaning
SID	Load set identification number (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Components of the acceleration vector measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
GRIDIDi	List of one or more GRID point identification numbers. Key words “THRU” and “BY” can be used to assist the listing. (Integer > 0)

Remarks:

- The acceleration vector is defined by  $\vec{a} = A \cdot \vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3).  
The magnitude of  $\vec{a}$  is equal to  $A$  times the magnitude of  $\vec{N}$ .
- This type of acceleration load may be combined with other loads, such as FORCE, MOMENT, GRAV, and ACCEL loads, by specification on a LOAD entry. The SID on an ACCEL1 entry may not be the same as that of any other load entry.
- This acceleration load does not include effects due to mass on scalar points.
- A CID of zero references the basic coordinate system.
- ACCEL1 loads are not supported with super elements.
- In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.

## ACCSSPT

## Access Point Definition

Defines a grid interior to an external superelement whose motion will be retained in the residual structure of the assembled superelement model. This entry is used during the external superelement creation run and is valid only in external superelements.

Format:

1	2	3	4	5	6	7	8	9	10
ACCSSPT	GID1	GID2	GIDi	etc					

Example:

ACCSSPT	117	119							
---------	-----	-----	--	--	--	--	--	--	--

Describer	Meaning
GIDi	Identification number of a grid point designated as an access point (Integer>0)

Remarks:

1. An access point may only be defined in an external superelement.
2. Access point IDs must be unique across all external superelements.
3. Access points must be interior points in the external superelement.
4. Access points will be automatically placed in the m-set of the residual structure of the assembly and will have the same displacements in the residual structure as it does in the external superelement.
5. If the user needs the access points to be in the a-set, or if user needs to connect access points from different superelements via MPCs, the following two parameters are required:  
PARAM, SEP1XOVR, 128  
PARAM, AUTOMSET, YES
6. If Modules are present then this entry may only be specified in the main Bulk Data section.



**ACLOAD****ACTRAN Acoustic Pressure Load Matrices for SOL 108/111**

Defines ACTRAN acoustic pressure load matrices.

Format:

1	2	3	4	5	6	7	8	9	10
ACLOAD	SID	UNIT1	UNIT2	SCLR	SCLI	LSQID			

Example(s):

ACLOAD	101	41	42	1.5					
--------	-----	----	----	-----	--	--	--	--	--

Field	Contents
SID	Set identification number. See Remark 1. (Integer > 0)
UNIT1	Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)
UNIT2	Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)
SCLR	The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)
SCLI	The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)
LSQID	Load SeQuence IDentification number on an ACTRAN file with multiple load cases. (Integer > 0, default=1).

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command `DLOAD = SID` for frequency response analysis.
2. The following type of `ASSIGN` should be specified in the FMS section with the vacant unit number (see `ASSIGN` statements). The unit number cannot be selected doubly.  
`ASSIGN INPUTT2='ACTRAN_pressure.f70' UNIT=41`  
`ASSIGN INPUTT4='ACTRAN_pressure.op4' UNIT=42`
3. `SID` must be unique for all `RLOAD1`, `RLOAD2`, `ACSRCE` and `ACLOAD` entries.
4. Refer the ACTRAN manual for the details of exportation of the acoustic pressure load matrix data for Nastran.
5. The residual vectors for `ACLOAD` will not be computed.
6. Value of `LSQID` that is greater than the actual load cases in the ACTRAN file is a fatal error.

**ACMODL**

**Fluid-Structure Interface Parameters**

Optional entry to define modeling parameters for Fluid-Structure Interface.

Format:

(If METHOD="BW")

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD	SKNEPS	DSKNEPS	
	INTOL	ALLSSET	SRCHUNIT						

(If METHOD="CP")

ACMODL	INTER	INFOR	FSET	SSET	NORMAL	METHOD			
--------	-------	-------	------	------	--------	--------	--	--	--

Example(s):

ACMODL	IDENT				NORMAL				
--------	-------	--	--	--	--------	--	--	--	--

Describer	Meaning
INTER	Type of structure-fluid interface. (Character = "IDENT" or "DIFF"; Default = "DIFF")
INFOR	For METHOD = "BW" and INTER = "DIFF", indicates if FSET and SSET are used to define the fluid-structure interface, "NONE" if not used, and whether they contain grids or elements. (Character = "GRIDS", "ELEMENTS", "ALL", or "NONE", Default = "NONE")  For METHOD = "CP" and INTER = "DIFF", indicates if FSET and SSET are used to define the fluid-structure interface, "NONE" if not used. See Remark 10. (Character = "ALL", or "NONE", Default = "NONE")
FSET	Optional identification of a SET1 entry that contains a list of fluid elements or grids on the fluid "skin". See Remark 2. If Modules are present and the MDFSET entry is to be used instead of SET1 then set FSET to -1. (Integer -1 or >0 or blank).
SSET	Optional identification of a SET1 entry that contains a list of structural elements or grids on the structure-fluid interface. See Remark 2. If Modules are present and the MDSSET entry is to be used instead of SET1 then set SSET to -1. (Integer -1 or >0 or blank).
NORMAL	Fluid normal tolerance. See Remark 5. (Real; Default = 1.0 (Real; .001 for IDENT)
METHOD	Default = "BW" "BW" = Body in White method "CP" = Closed Pressure Vessel Method See Remarks 10. and 11.
SKNEPS	Fluid skin growth tolerance. (Real; Default 0.5)



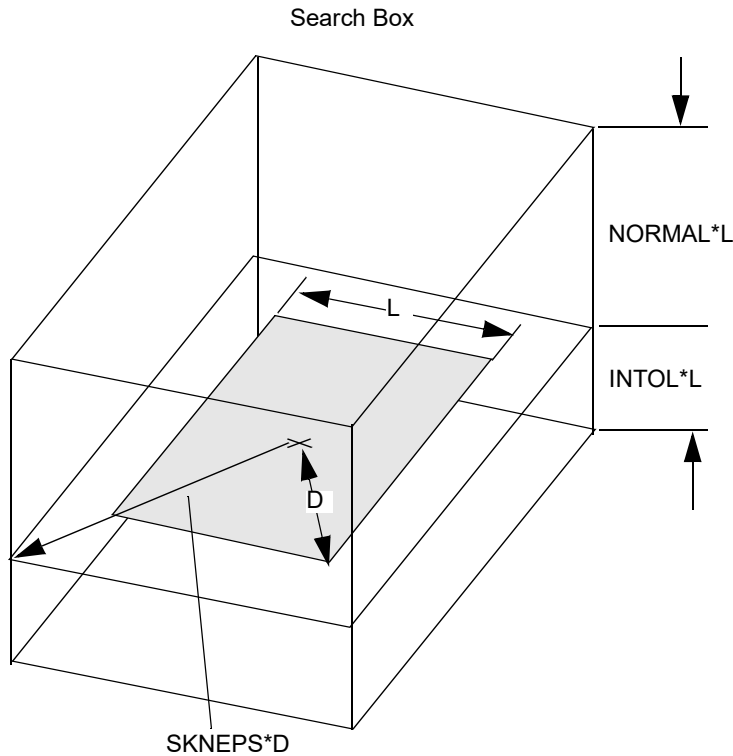
Describer	Meaning
DSKNEPS	Secondary fluid skin growth tolerance (Real; Default .75)
INTOL	Tolerance of inward normal. (Real; Default .5)
ALLSSET	If “NO” then SSET structure is searched and coupled if found. If ‘YES’ then all the structure given by SSET is coupled. (Character = ‘YES’, or ‘NO’; Default = ‘NO’)
SRCHUNIT	Search units. (Character=‘ABS’ for absolute model units or ‘REL’ for relative model units based on element size; Default = ‘REL’)

#### Remarks:

1. Only one ACMODL entry is allowed. In general, for large irregular models, it is recommended that, initially, this entry not be used, so the defaults will be applied.
2. For METHOD = “BW” and INTER = “DIFF” (Default), FSET and SSET refer to either grids or elements as selected below. For INTER = “IDENT”, FSET and SSET refer to grids.
  - a. For INTER = “DIFF”, INFOR = “ELEMENTS”, for FSET, the search algorithm is restricted to elements referenced by FSET. For SSET, the search algorithm is restricted to elements referenced by SSET. This allows the user to de-select specific structural faces of a solid structural element. Both these sets are optional and the user can have one without the other.
  - b. For INTER = “DIFF”, INFOR = “GRIDS”, for FSET, the search algorithm is restricted to grids referenced by FSET. This allows the user to deselect fluid grids. Fluid grid selection is the only way to deselect specific fluid faces. For SSET, the search algorithm is restricted to grids referenced by SSET. This allows the user to de-select structural grids. Both these sets are optional and the user can have one without the other.
  - c. For INTER = “IDENT”, INFOR = “ALL”, the points referenced by FSET and SSET must lie exactly on the fluid-structure interface. These sets are optional, but if used, both must be present or no fluid interface is calculated.
3. For INTER = “DIFF”, a .PCH file is created with a SET1 representing the fluid “skin” and a SET1 representing the structure interface. This file is useful for graphic post-processing for viewing the interface. It also produces the sets that can be used as FSET and SSET.
4. For ALLSSET = ‘NO’ (default) the elements and grids determined by the couplings algorithm are written to the .PCH file. The user can then deselect elements or grids as defined by the .PCH file by editing them out of the SET1 entries defined in the file and referencing the edited SET1 with the SSET. To add structural elements that the coupling algorithm did not include in the .PCH file, it is not sufficient to just include them on the SET1 entry referenced by SSET. In addition, ALLSSET = ‘YES’ must be specified.
5. NORMAL determines the height of the fluid box in the outward normal direction to the fluid surface. The fluid box is used to locate the structural elements used in defining the fluid-structure coupling matrix. If L is the smallest edge of the fluid element surface, then the height of the box is  $L \times \text{NORMAL}$ .

For INTER = “IDENT”, NORMAL = .001 is the default and represents a tolerance, in units of length, used in determining the fluid-structure interface.

- SKNEPS represents the enlargement of the plane of the fluid surface used to define the search box. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal  $\times (1. + SKNEPS)$ ).



- DSKNEPS represents a secondary enlargement of the plane of the fluid surface used to define the search box if SKNEPS fails to find ANY structural elements. The diagonal distance from the center of the fluid surface to each surface grid is pushed out (diagonal  $\times (1. + DSKNEPS)$ ).
- INTOL represents a normal direction into the fluid for the case when the fluid protrudes past the structural interface. It is defined as  $L \times INTOL$  where L is the smallest edge of the fluid element surface.
- The BW method is summarized in the following table:

Table of Fluid/Structure Search Control and Override METHOD=BW	
INFOR	Indicates the types of FSET and SSET entered
	ELEMENTS      Element sets are used to select element fluid and structure faces.
	GRIDS            Grid sets are used to select element fluid and structure faces.
FSET	Fluid Set ID to delete or add fluid faces to fluid/structure interface
	ALLSET=NO      Delete fluid element faces

Table of Fluid/Structure Search Control and Override METHOD=BW

	ALLSET=YES	Add fluid element faces
SSET		Structure Set ID to delete or add structure faces to fluid/structure interface
	ALLSET=NO	Delete structural element faces
	ALLSET=YES	Add structural element faces
		A SET specification will for the specified set given override any search parameters.  Caution: When deleting items from a set, especially structural element sets an element ID may occur more than once and ALL must be removed.
SRCHUNIT	REL	Units are in the units of smallest fluid edge of current fluid face being searched.  L = smallest fluid surface edge of element being searched.  D = Diagonal from center of fluid surface to each grid  NORMAL = Value_input * L, Height of box +normal to plane of fluid.  SKNEPS = D * (1.0 + Value_input), Search plane diagonals.  DSKNEPS = D * (1.0 + Value_input) Secondary search plane diagonals  DSKNEPS is only used when SKNEPS < DSKNEPS  INTOL = Value_input * L, Height of box -normal to plane of fluid.
	ABS	Normal units are in the UNITS of the STRUCTURE. Use when distance between the structure interface and the fluid interface is same constant everywhere.  NORMAL = Value_input  SKNEPS = As defined above but projected using ABS normal units  DSKNEPS = As defined above but projected using ABS normal units  INTOL = Value_input
ALLSSET	NO	Designed to take a modified PARAM SKINOUT PUNCH set and DELETE elements from either the FLUID element set or the STRUCTURE element set or BOTH. Note if the FLUID set has some deleted elements and the STRUCTURE set has added elements this option may result in some change of structure elements in the set and not keep the structural set UNMODIFIED or vice versa. This is because not all of the UNMODIFIED structure elements of the set project with the reduced set of Fluid elements or vice versa.  Recommended for DELETE only.

Table of Fluid/Structure Search Control and Override METHOD=BW		
	YES	<p>Designed to take a modified PARAM SKINOUT PUNCH set and ADD elements to either the FLUID element set or the STRUCTURE element set or BOTH. If some previously found FLUID and STRUCTURE are also deleted the algorithm keeps all the added set elements but will of course recomputed the interface for the deleted boundary so that a sensible set intersection is computed.</p> <p>Recommended for DELETE ADD.</p>
		<p>Do NOT use ALLSET =YES when no SETs are specified or one SET is specified and the search algorithm is being used for the other unspecified SET as ALLSET means USE ALL and all of the elements of the unspecified SET may be used.</p>
Coupling Algorithm		<ol style="list-style-type: none"> <li>1. Use the search box algorithm to locate the fluid free faces and the corresponding structural element faces.</li> </ol>
		<ol style="list-style-type: none"> <li>2. For a fluid free face and its list of structural element faces (that were determined by boxing normal to the fluid element) do as follows:             <ol style="list-style-type: none"> <li>a. For each fluid free face establish a face coordinate system.</li> <li>b. Determine the resultant pressure force for each grid on the fluid element.</li> <li>c. Resolve this resultant pressure force for a unit grid pressure to the grids of the fluid element. (Determined by principal virtual work).</li> <li>d. Using the origin of the free fluid face, determine the center of pressure.</li> <li>e. Using rigid body motion consider only a unit motion normal to the fluid face with the appropriate moment relationships, determine the resulting load distribution at the grids of each of the structural elements. The area of each structural element projected normal to the fluid element is used as a weighting function.</li> <li>f. Loop over each grid of the fluid element and accumulate the forces at the structural grids.</li> <li>g. The algorithm always maintains of rigid body equilibrium.</li> </ol> </li> <li>3. Repeat for the next fluid element and its associated group of structural elements. Accumulate the forces at the structural grids.</li> </ol>
Guidelines		<ol style="list-style-type: none"> <li>1. Run default ACMODL with PARAM , SKINOUT, PUNCH</li> <li>2. View SETS both FLUID and STRUCTURE.</li> <li>3. Adjust ACMODL search parameters until close to desired interface is obtained.</li> <li>4. Modify the FSET, or SSET, or both to include and exclude Grids or Elements as seems appropriate to get final coupling elements.</li> </ol>

Table of Fluid/Structure Search Control and Override METHOD=BW

	5. Run acoustic analysis with ACMODL using final FSET, SSET sets with ALLSET=YES if sets contain both deleted and added elements. Or ALLSET=NO (default) if sets only delete.
	6. The ABS option IS NOT RECOMMENDED for complex acoustic models such as body in white models with multi contouring surfaces.
Parallel Structural Element Meshes in the Search Box	When there are "parallel" overlapping structural meshes, the search algorithm will pick an element of the closest mesh to the fluid face. All elements connected to this first element are then checked to see if they are in the search box. Once an element, such as the elements of the outer parallel mesh, are found not to connect to the inner patch it is eliminated from the search. Thus outer patch elements tend to be excluded from the search unless some connectivity to the inner patch is detected. The ALLSET=YES is to force an interface between the listed structural elements and the nearest fluid faces.

10. The default METHOD is the new "BW" searching algorithm that requires a special license.

The pre-Version 2004 method is selected with METHOD = "CP" in which:

- The search box is not used so the SKNEPS, DSKNEPS, INTOL, ALLSET, SRCHUNIT fields are ignored.
- If INFOR = 'ALL' (METH = 'CP' only), then both FSET and SSET must be specified and matching is checked at only those grid points referenced by FSET and SSET.
- FSET and SSET refer to grids.
- NORMAL = blank is the default (recommended),  $1. \leq \text{NORMAL} \leq 10.$  gets acceptable results. In this case, NORMAL represents a maximum cutoff value measured in physical units. When NORMAL = 'blank', Nastran will compute the cutoff value.

This field replaces the pre-Version 2004 FSTOL field. Different units are also used so pre-Version 2004 FSTOL values may need to be changed to obtain the same results.

11. METH = 'CP' is not recommended when complex acoustic cavities (such as body in white automotive models) exist in the model.
12. If Modules are present then this entry may only be specified in the main Bulk Data section.

## ACPEMCP

## Trim Component Interface Coupling and Constraints Definition

Defines the interface coupling conditions and constraints of a trim component.

Format:

1	2	3	4	5	6	7	8	9	10
ACPEMCP	TID	SGLUED	SSLIDE	SOPEN	SIMPER	OOO	SPM	SAIRGAP	
	SCUX	SCUY	SCUZ	SCRX	SCRY	SCRZ	SCFP	RID	

Example:

ACPEMCP	1		1002		1004				
---------	---	--	------	--	------	--	--	--	--

Describer	Meaning
TID	Identification number of trim component. (Integer>0; Required)
SGLUED	Identification number of SET1/SET3 entry of grids belonging to the solid-phase and/or structure volume elements in the trim component, which are glued to a structure; see Remark 2. (Integer≥0 or Blank)
SSLIDE	Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in the sliding-contact to a structure; see Remark 3. (Integer0 or Blank)
SOPEN	Identification number of SET1/SET3 entry of grids belonging to the fluid-phase elements in the trim component, which has an open interface with the cavities; see Remark 4. (Integer≥0 or Blank)
SIMPER	Identification number of SET1/SET3 entry of grids of the solid-phase and/or structure volume elements in the trim component, which has an impervious interface with the cavities; see Remark 5. (Integer≥0 or Blank)
OOO	Number of blocks for Out-Of-Core solver, see remark 8. (Integer>1, Default=1)
SPM	Flag for selecting single precision MUMPS for computing reduced impedance matrix, see Remark 9. (Integer=>=0, default=0)
SAIRGAP	Identification number of SET1/SET3 entry with grids belonging to the solid-phase and/or structure volume elements in the trim component, which are in contact with structure via air gap. See Remark 10. (Integer = 0 or Blank)
SCUX	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation X direction of output coordinate system.
SCUY	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Y direction of output coordinate system.
SCUZ	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in translation Z direction of output coordinate system.



Describer	Meaning
SCRX	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation X direction of output coordinate system.
SCRY	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Y direction of output coordinate system.
SCRZ	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in rotation Z direction of output coordinate system.
SCFP	ID of SET1/SET3 with which grid IDs in the trim component TID is zero-constrained in fluid pressure.
RID	ID of a region of a TRMC. (Blank or Integer>0, Default=0).

#### Remarks:

1. ACPEMCP is a required entry for each trim component that is referenced by TRIMGRP. It must be put in the main Bulk Data section or BEGIN BULK. It specifies the boundary coupling conditions as well as the coupling degrees of freedom for computing the reduced boundary impedance matrices of the trim component.
2. A glued interface means that the trim component is completely constrained on to the structure, such as a car body. There is no relative movement between the trim component and the structure at the interface.
3. A sliding interface means that the trim component is laid on the surface of structure, such as a car body. There is no relative movement in the normal direction of the interface between the trim component and the structure. Relative sliding movement is allowed in the contact surface.
4. An open interface means that the free flow of fluid between the fluid-phase and cavities occurs at the interface.
5. An impervious or closed interface means that the fluid flow between the trim component and cavities is prohibited at the interface. This can happen when the porous material is covered by a thin film which stops the fluid flow or the cavity is in touch with a structure part of the trim component.
6. The SET1/SET3 entries must reside in the Bulk Data section introduced by Case Control, BEGIN BULK TRMC.
7. Under current PEM implementation, only 3D elements are allowed for trim components. Hence, SCRX, SCRY and SCRZ fields of ACPEMCP are not processed.
8. OOC is intended for large trim component. The number of OOC field defines the number of blocks to be used for the Schur complement evaluation which reduces the memory requirement. A side effect of OOC>1 is full RIM is generated, instead of lower triangular RIM, which is similar to PARAM,PEMFRIM,1.
9. SPM is a flag for selecting single precision MUMPS in ACTRAN. With default value of 0, double precision MUMPS is utilized. Single precision MUMPS will be used with SPM field having value greater 0. Single precision MUMPS has the benefits of reduced memory requirement and better performance. However, single precision MUMPS may suffer minor degradation on accuracy.

10. An air gap interface means the trim component is assumed to be coupled to the structure through a very thin air layer. The thin air layer is taken into the analysis without being modeled (not being defined in the trim model file).
11. A TRMC can have many ACPEMCP entries. However, an ACPEMCP of a TRMC with blank RID field is a must before ACPEMCP with non-blank RID can show up.

ACSRCE

Acoustic Source Specification

Defines acoustic source as a function of power vs. frequency.

$$\text{Source Strength} = \{A\} \cdot \left[ \frac{1}{2\pi f \sqrt{\rho}} \sqrt{8\pi CP(f)} \right] e^{i(\theta + 2\pi f\tau)}$$

$$C = \sqrt{B/\rho}$$

Format:

1	2	3	4	5	6	7	8	9	10
ACSRCE	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TP/RP	RHO	B		

Example:

ACSRCE	103	11	20	5.0	12	1.0	15.0		
--------	-----	----	----	-----	----	-----	------	--	--

Describer	Meaning
SID	Load set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of DAREA, FBALOAD (in FRF Based Assembly or FBA process) or SLOAD entry set the defines {A}. See Remark 6. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ. See Remarks 4. and 5. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
DPHASEI	Identification number of DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ. (See Remarks 4. and 5. (Integer > 0 or blank)
DPHASER	Value of phase angle θ (in degrees) that will be used for all fluid degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
TP	Identification number of a TABLEDi entry that defines power versus frequency, P(f). (Integer > 0)
RP	Value of power P to be used for all frequencies (Real, non-zero)
RHO	Density of the fluid. (Real > 0.0)
B	Bulk modulus of the fluid. (Real > 0.0)

Remarks:

1. Acoustic sources must be selected in the Case Control Section with DLOAD = SID.
2. For additional remarks, see the RLOAD1 entry description.



3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The referenced EXCITEID, DELAY, and DPHASE entries must specify fluid points only.
5. If any of the DELAYI/DELAYR or DPHASEI/DPHASER fields are blank or zero, the corresponding  $\tau$  or  $\theta$  will be zero.
6. If there is no LOADSET Case Control command, then EXCITEID may reference DAREA and SLOAD entries. If there is a LOADSET Case Control command, then EXCITEID may reference DAREA entries as well as SLOAD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

## ACTIVAT Defines Elements That Were Previously Deactivated and Should Be Reactivated for a Particular Subcase in SOL 600

This entry allows the user to re-activate certain elements that were previously deactivated in a previous subcase in SOL 600.

### Format:

1	2	3	4	5	6	7	8	9	10
ACTIVAT	ID	IGEOM	ISET						

### Example:

ACTIVAT	3	0	300						
---------	---	---	-----	--	--	--	--	--	--

Describer	Meaning
ID	Identification number of a matching Case Control ACTIVAT command defining the subcase to which these elements should be activated. (Integer; no Default)
IGEOM	Flag to control whether to update the geometry on the post file. (Integer; Default = 0) 0        Update the geometry. 1        Do not update the geometry.
ISET	ID of a list of elements described by SET3 is ID=ISET to be activated. (Integer; no Default)

### Remark:

1. This entry maps to Marc's ACTIVATE History definition option.

## ACTRIM

## ACTRAN Trimmed Material Matrices for SOL 108/111

Defines ACTRAN trimmed material matrices.

Format:

1	2	3	4	5	6	7	8	9	10
ACTRIM	NAME	UNIT1	UNIT2	SCLR	SCLI				

Example(s):

ACTRIM	FLOOR	31	32	2.0	0.5				
--------	-------	----	----	-----	-----	--	--	--	--

Field	Context
NAME	Name of the ACTRAN trimmed material matrices. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic)
UNIT1	Fortran unit number of mapped data from ACTRAN. See Remark 2. (Integer > 0)
UNIT2	Fortran unit number of property matrices from ACTRAN. See Remark 2. (Integer > 0)
SCLR	The real part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 1.0)
SCLI	The imaginary part of complex scale factor to be multiplied to ACTRAN matrices. (Real; Default = 0.0)

Remarks:

1. ACTRAN trimmed material matrices defined by this entry will be used for frequency response analysis if it is selected via the Case Control ACTRIM = NAME.
2. The following type of ASSIGN should be specified in the FMS section with the vacant unit number (see ASSIGN statements). The unit number cannot be selected doubly.  
 ASSIGN INPUTT2='ACTRAN\_trimmed.f70' UNIT=31  
 ASSIGN INPUTT4='ACTRAN\_trimmed.op4' UNIT=32
3. Refer the ACTRAN manual for the details of exportation of the trimmed material matrix data for Nastran.

## ADUMi Dummy Element Attributes

Defines attributes of the dummy elements ( $3 \leq i \leq 9$ ).

Format:

1	2	3	4	5	6	7	8	9	10
ADUMi	NG	NC	NP	ND	ELNM				

Example:

ADUM8	18	0	5	0	CRAC2D				
-------	----	---	---	---	--------	--	--	--	--

Describer	Meaning
NG	Maximum number of grid points that may be connected by DUMi dummy element (Integer>0.)
NC	Number of additional fields (Ai) on the CDUMi connection entry. (Integer $\geq 0$ )
NP	Number of additional fields (Ai) on the PDUMi property entry. ( $24 \geq \text{Integer} \geq 0$ )
ND	Number of displacement components at each grid point used in generation of the differential stiffness matrix. Zero implies no differential stiffness. (Integer 3 or 6)
ELNM	The name of the element connection and property entry. In the example above, the connection entry is named "CRAC2D" and the property entry is named "PRAC2D".

Remarks:

ADUM8 and ADUM9 are used exclusively for the CRAC2D and CRAC3D entries respectively.

## AECOMP

## Component for an Integrated Load Monitor Point

Defines a component for use in monitor point definition or external splines.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMP	NAME	LISTTYPE	LISTID1	LISTID2	LISTID3	LISTID4	LISTID5	LISTID6	
	LISTID7	-etc.-							

Example:

AECOMP	WING	AELIST	1001	1002					
--------	------	--------	------	------	--	--	--	--	--

Describer	Meaning
NAME	A character string of up to eight characters identifying the component. (Character)
LISTTYPE	One of CAERO, AELIST or CMPID for aerodynamic components and SET1 for structural components. Aerodynamic components are defined on the aerodynamic ks-set mesh while the structural components are defined on the g-set mesh. See Remarks 2. and 4.
LISTID <sub>i</sub>	The identification number of either SET1, AELIST or CAERO <sub>i</sub> entries that define the set of grid points that comprise the component. See Remarks 2. and 4.

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. If the component is defined on the structure, LISTIDs must refer to SET1 entry(ies) that define the list of associated GRID points. For the AELIST or CAERO option, the LISTIDs must refer to AELIST or CAERO *i* entries, respectively. Note that, for DLM models (CAERO1/2), the set of points defined by the AELIST are the box identification numbers. For example, if the control surface's grids are desired, the same AELIST used for the AESURF can be referred to here. An AECOMP component must be defined as either an aerodynamic mesh component or a structural component. The two mesh classes cannot be combined into a single component.
3. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.
4. If LISTTYPE = CMPID, LISTID1 identifies the associated set of AEQUAD4, AETRIA3 elements that define the aero component.



**AECOMPL****Component for an Integrated Load Monitor Point**

Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.

Format:

1	2	3	4	5	6	7	8	9	10
AECOMPL	NAME	LABEL1	LABEL2	LABEL3	LABEL4	LABEL5	LABEL6	LABEL7	
	LABEL8	-etc.-							

Example:

AECOMPL	HORIZ	STAB	ELEV	BALANCE					
---------	-------	------	------	---------	--	--	--	--	--

Describer	Meaning
NAME	A character string of up to eight characters Identifying the component. (Character)
LABELi	A string of 8 characters referring to the names of other components defined by either AECOMP or other AECOMPL entries.

Remarks:

1. The Identification name must be unique among all AECOMP and AECOMPL entries.
2. The AECOMPL entry can be used to combine AECOMP entries into new components. When combining components, the structural and aerodynamic classes must be kept separate.

## AEDW

## Parametric Normal Wash Loading for Aerodynamics

Defines a downwash vector associated with a particular control vector of the associated aerodynamic configuration (AECONFIG). From this downwash vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEDW	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEDW	0.90	SYMM	ASYMM	101	ALP1				
------	------	------	-------	-----	------	--	--	--	--

Describer	Meaning
MACH	The Mach number for this force, see Remark 2. (Real $\geq 0.0$ , $\neq 1.0$ )
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI (Character).
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this downwash vector.
DMIJ	The name of a DMI or DMIJ entry that defines the downwash.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE and AEPRESS are associated with the current AECONFIG using either Case Control (if in the main Bulk Data Section) or using the BEGIN AECONFIG=<config> if in a partition of the Bulk Data.
2. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.
3. Mach numbers > 1.0 require that the supersonic aerodynamic option be available.

## AEFACT Aerodynamic Lists

Defines real numbers for aeroelastic analysis.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
AEFACT	SID	D1	D2	D3	D4	D5	D6	D7	
	D8	D9	-etc.-						

Example:

AEFACT	97	.3	.7	1.0					
--------	----	----	----	-----	--	--	--	--	--

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
Di	Number. (Real)

Remarks:

1. AEFACT entries must be selected by a CAEROi, PAEROi or SPLINEX entry.
2. Embedded blank fields are not allowed.
3. To specify division points, there must be one more division point than the number of divisions.
4. When referenced by the CAERO3 entry, AEFACT defines the aerodynamic grid points. The ID number of the first point defined by each AEFACT entry is the value of the CAERO3 ID that selected the AEFACT entry. The ID of each following point defined on the AEFACT is incremented by 1.

## AEFORCE

## Parametric Force for Aerodynamics

Defines a vector of absolute or “per unit dynamic pressure” forces associated with a particular control vector. This force vector may be defined on either the aerodynamic mesh (ks-set) or the structural mesh (g-set). The force vector will be used in static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEFORCE	MACH	SYMXZ	SYMXY	UXID	MESH	LSET	DMIK	PERQ	

Example:

AEFORCE	0.90	SYMM	ASYMM	101	AERO		BETA		
---------	------	------	-------	-----	------	--	------	--	--

Descriptor	Meaning
MACH	The Mach number for this force. (Real $\geq 0.0$ , $\neq 1.0$ )
SYMXZ,SYMXY	The symmetry conditions for this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this force vector.
MESH	One of AERO or STRUCT that declares whether the force vector is defined on the aerodynamic ks-set mesh or the structural g-set mesh.
LSET	SID of a load set that defines the vector. See Remark 2. (Integer $> 0$ if MESH=STRUCT)
DMIK	The name of a DMIK entry that defines the aerodynamic force vector. See Remark 3. (Character; Required if MESH=AERO)
PERQ	The string PERQ or blank. If PERQ and MESH=STRUCT, the input FORCE set is multiplied by the dynamic pressure. (Default = blank; see Remark 4.)

Remarks:

1. The AEFORCE is associated with the current AECONFIG and must be entered for the appropriate Mach numbers and aerodynamic symmetries.
2. For the STRUCT mesh option, the LSET can refer to any existing load type (e.g., FORCE1, PLOAD4 or LOAD) that is available to define static loads.
3. For the AERO mesh option, the DMIK Bulk Data are used. Any forces associated with the aerodynamic model’s permanently SPC’d degrees-of-freedom (which are dependent on the type of aerodynamic model being used) will be ignored.
4. For the AERO mesh option, the PERQ field is not used. The FORCE data input on the DMIK entry is always multiplied by dynamic pressure.

## AERGRID

## Aerodynamic Grid Point

Defines the location of an aerodynamic grid point.

Format:

1	2	3	4	5	6	7	8	9	10
AERGRID	GID	CP	X1	X2	X3	CD	ASID		

Example:

AERGRID	1	1	2.1	3.2	2.0	3	4		
---------	---	---	-----	-----	-----	---	---	--	--

Describer	Meaning
GID	Grid Point Identification Number ( $0 < \text{Integer} < 100,000,000$ )
CP	Identification number of a coordinate system in which the location of the grid point is defined. ( $\text{Integer} \geq 0$ or blank)
X1, X2, X3	Location of the grid point in the coordinate system CP.
CD	Identification number of coordinate system in which the degrees-of-freedom of the grid point are defined. ( $\text{Integer} \geq 0$ or blank)
ASID	Identification number of an AESCALE Bulk Data entry. ( $\text{Integer} > 0$ or blank)

Remarks:

1. All grid point identification numbers must be unique with respect to all other aerodynamic grid point identification numbers.
2. If both CP and ASID are defined, coordinates are first scaled and subsequently transformed to the basic coordinate system.
3. The meaning of X1, X2 and X3 depends on the type of coordinate system CP. (See Remark 2 of the GRID entry).
4. A zero or blank in the CP and CD fields refers to the basic coordinate system.
5. The AERGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

## AELINK Links Aeroelastic Variables

Defines relationships between or among AESTAT and AESURF entries, such that:

$$u^D + \sum_{i=1}^n C_i u_i^I = 0.0$$

Where:

$u^D$  = dependent variable  
 $u_i^I$  = independent variable

Format:

1	2	3	4	5	6	7	8	9	10
AELINK	ID	LABLD	LABL1	C1	LABL2	C2	LABL3	C3	
	LABL4	C4	-etc.-						

Example:

AELINK	10	INBDA	OTBDA	-2.0					
--------	----	-------	-------	------	--	--	--	--	--

Describer	Meaning
ID	If an integer > 0 is specified, this is the TRIM set ID selected in Case Control and the AELINK only applies to that subcase. If an integer value of 0 or the character string "ALWAYS" is specified, this AELINK is applicable to all subcases. (Integer $\geq 0$ or the "ALWAYS" character string.)
LABLD	Character string to identify the dependent aerodynamic variable. (Character)
LABLi	Character string to identify the i-th independent aerodynamic variable. (Character)
Ci	Linking coefficient for the i-th variable. (Real)

Remarks:

1. If the ID is a positive integer, the AELINK entry (or entries) is selected by the TRIM=ID in Case Control.
2. If the ID is 0 or the character string ALWAYS, the linking relationship applies to all subcases.
3. The entry constrains the dependent variable to be a linear combination of the independent variables.
4. LABLD data must be unique for a given ID or if ID=0 or ALWAYS is being used (i.e., the variable cannot be constrained more than once).
5. LABLD and LABLi refer to AEPARM, AESTAT or AESURF Bulk Data entries.

## AELIST

## Aerodynamic List

Defines a list of aerodynamic elements or grid ID's.

Format:

1	2	3	4	5	6	7	8	9	10
AELIST	SID	E1	E2	E3	E4	E5	E6	E7	
	E8	-etc.-							

Example:

AELIST	75	1001	THRU	1075	1101	THRU	1109	1201	
	1202								

Describer	Meaning
SID	Set identification number. (Integer > 0)
Ei	List of aerodynamic box ID's or aerodynamic grid ID's (Integer>0 or "THRU")

Remarks:

1. The AELIST entry can be referenced by AESURF, AECOMP, SPLINEi, SPLINRB, SPBLND2 and SPRELAX entries. Refer to these entries for the meaning of the data provided.
2. When the "THRU" option is used, all intermediate grid points must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
3. Intervening blank fields are not allowed.

## AELISTC

## Character Item List

Defines a list of 8-character strings.

Format:

1	2	3	4	5	6	7	8	9	10
AELISTC	SID	C1	C2	C3	C4	C5	C6	C7	
	C8	-etc.-							

Example:

AELISTC	101	FBS	STRING12						
---------	-----	-----	----------	--	--	--	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
Ci	List of 8-character strings.

Remark:

1. Intervening blank fields are not allowed.



## AEPARM

### General Controller for Use in Trim

Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point). The forces associated with this controller will be derived from AEDW, AEFORCE and AEPRESS input data.

Format:

1	2	3	4	5	6	7	8	9	10
AEPARM	ID	LABEL	UNITS						

Example:

AEPARM	5	THRUST	LBS						
--------	---	--------	-----	--	--	--	--	--	--

Describer	Meaning
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. See Remark 1. (Character)
UNITS	Label used to describe the units of the controller values. (Character)

Remarks:

1. Controller LABELs that comprise the unique set relative to all the AESURF, AESTAT and AEPARM entries will define the set of trim variable degrees-of-freedom for the aeroelastic model.
2. Unit labels are optional and are only used to label outputs. No units will be associated with the controller if left blank.

## AEPRESS

## Parametric Pressure Loading for Aerodynamics

Defines a vector of pressure/unit dynamic pressure associated with a particular control vector. From this pressure vector, a force vector on the aerodynamic grids will be defined for use in nonlinear static aeroelastic trim.

Format:

1	2	3	4	5	6	7	8	9	10
AEPRESS	MACH	SYMXZ	SYMXY	UXID	DMIJ	DMIJI			

Example:

AEPRESS	0.90	SYMM	ASYMM	101	ALP1				
---------	------	------	-------	-----	------	--	--	--	--

Descriptor	Meaning
MACH	The Mach number for this force, see Remark 2. (Real $\geq 0.0$ , $\neq 1.0$ )
SYMXZ,SYMXY	The symmetry of this force vector. One of SYMM, ASYMM or ANTI. (Character)
UXID	The identification number of a UXVEC entry that defines the control parameter vector associated with this pressure vector.
DMIJ	The name of a DMI or DMIJ entry that defines the pressure per unit dynamic pressure.
DMIJI	The name of a DMIJI entry that defines the CAERO2 interference element “downwashes”.

Remarks:

1. The AEDW, AEFORCE, and AEPRESS are associated with the current AECONFIG using Case Control.
2. Mach numbers  $> 1.0$  require that the supersonic aerodynamic option be available.
3. The DMIJ field refers to either a DMI or a DMIJ entry. The DMIJI is only applicable to CAERO2 and is only required if nonzero “downwash” (j-set) input is needed on the interference body elements.

## AEQUAD4 Quadrilateral Aerodynamic Element Connection

Defines the connectivity of a quadrilateral aerodynamic element.

Format:

1	2	3	4	5	6	7	8	9	10
AEQUAD4	EID	CMPID	G1	G2	G3	G4			

Example:

AEQUAD4	100	2	12	32	41	50			
---------	-----	---	----	----	----	----	--	--	--

Describer	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
CMPID	Aerodynamic Component Identification Number. (Integer > 0)
Gi	Grid Point Identification Numbers of Connection Points. (Integer > 0)

Remarks:

1. The geometry of a quadrilateral aerodynamic element may collapse to a triangle, i.e., two connection points may have the same geometric location. However, all four grid point identification numbers must be different.
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

## AERO Aerodynamic Physical Data

Gives basic aerodynamic parameters for unsteady aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
AERO	ACSID	VELOCITY	REFC	RHOREF	SYMZX	SYMXY			

Example:

AERO	3	1.3+4	100.	1.-5	1	-1			
------	---	-------	------	------	---	----	--	--	--

Describer	Meaning
ACSID	Aerodynamic coordinate system identification. See Remark 2. (Integer $\geq 0$ ; Default is the basic coordinate system)
VELOCITY	Velocity for aerodynamic force data recovery and to calculate the BOV parameter. See Remark 5. (Real $> 0.0$ or blank)
REFC	Reference length for reduced frequency. (Real $> 0.0$ )
RHOREF	Reference density. (Real $> 0.0$ )
SYMZX	Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effect. (Integer = -1 for symmetry, 0 for no symmetry, and +1 for antisymmetry; Default = 0)

Remarks:

1. This entry is required for aerodynamic problems. Only one AERO entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction.
3. Set SYMXY = -1 to simulate ground effect.
4. PARAM,WTMASS does not affect aerodynamic matrices. RHOREF must be input in mass units.
5. VELOCITY is used only in aeroelastic response analysis, and it must be equal to V on the GUST Bulk Data entry.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMX, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

## AEROS

### Static Aeroelasticity Physical Data

Defines basic parameters for static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AEROS	ACSID	RCSID	REFC	REFB	REFS	SYMXZ	SYMXY		

Example:

AEROS	10	20	10.	100.	1000.	1			
-------	----	----	-----	------	-------	---	--	--	--

Describer	Meaning
ACSID	Aerodynamic coordinate system identification. See Remark 2. (Integer $\geq 0$ ; Default is the basic coordinate system)
RCSID	Reference coordinate system identification for rigid body motions. (Integer $\geq 0$ ; Default is the basic coordinate system)
REFC	Reference chord length. (Real $> 0.0$ )
REFB	Reference span. (Real $> 0.0$ )
REFS	Reference wing area. (Real $> 0.0$ )
SYMXZ	Symmetry key for the aero coordinate x-z plane. See Remark 6. (Integer = +1 for symmetry, 0 for no symmetry, and -1 for antisymmetry; Default = 0)
SYMXY	The symmetry key for the aero coordinate x-y plane can be used to simulate ground effects. (Integer = +1 for antisymmetry, 0 for no symmetry, and -1 for symmetry; Default = 0)

Remarks:

1. This entry is required for static aeroelasticity problems. Only one AEROS entry is allowed.
2. The ACSID must be a rectangular coordinate system. Flow is in the positive x-direction (T1).
3. The RCSID must be a rectangular coordinate system. All AESTAT degrees-of-freedom defining trim variables will be defined in this coordinate system.
4. REFB should be full span, even on half-span models.
5. REFS should be half area on half-span models.
6. The symmetry fields on this entry are only used if neither of the Case Control commands (AESYMY, AESYMXZ) are supplied. If either Case Control command is supplied, even the defaults from Case will override these Bulk Data entries. The Case Control symmetry is the preferred means of declaring the flow and geometric symmetry for aeroelastic analysis.

## AESCALE

## Aerodynamic Grid Point Scaling

Defines reference lengths to scale aerodynamic grid points.

Format:

1	2	3	4	5	6	7	8	9	10
AESCALE	ASID	X1REF	X2REF	X3REF					

Example:

AESCALE	100	1	1	1					
---------	-----	---	---	---	--	--	--	--	--

Describer	Meaning
ASID	Identification Number called out on an AEGRID entry. ( $0 < \text{Integer} < 100,000,000$ )
X1REF	Reference length to scale X1. (Real or blank; Default = 1.0)
X2REF	Reference length to scale X2. (Real or blank; Default = 1.0)
X3REF	Reference length to scale X3. (Real or blank; Default = 1.0)

Remark:

1. The scaled coordinates are computed from  $X_iS = X_i \cdot X_iREF$ .
2. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

## AESTAT

## Static Aeroelasticity Trim Variables

Specifies rigid body motions to be used as trim variables in static aeroelasticity.

Format:

1	2	3	4	5	6	7	8	9	10
AESTAT	ID	LABEL							

Example:

AESTAT	5001	ANGLEA							
--------	------	--------	--	--	--	--	--	--	--

Describer	Meaning
ID	Identification number of an aerodynamic trim variable degree-of-freedom. See Remark 1. (Integer > 0)
LABEL	An alphanumeric string of up to eight characters used to identify the degree-of-freedom. See Remark 1. (Character)

Remarks:

- The degrees-of-freedom defined with this entry represent rigid body motion in the reference coordinate system defined on the AEROS entry. The standard labels that define the various rigid body motions are as follows:

**Table 1      Standard Labels Defining Rigid Body Motions**

LABEL	Degree-of-Freedom Motion	Description
ANGLEA	$u_r$ (R2)	Angle of Attack
SIDES	$u_r$ (R3)	Angle of Sideslip
ROLL	$\dot{u}_r$ (R1)	Roll Rate = pb/2V
PITCH	$\dot{u}_r$ (R2)	Pitch Rate = qc/2V
YAW	$\dot{u}_r$ (R3)	Yaw Rate = rb/2V
URDD1	$\ddot{u}_r$ (T1)	Longitudinal (See Remark 3.)
URDD2	$\ddot{u}_r$ (T2)	Lateral
URDD3	$\ddot{u}_r$ (T3)	Vertical
URDD4	$\ddot{u}_r$ (R1)	Roll
URDD5	$\ddot{u}_r$ (R2)	Pitch
URDD6	$\ddot{u}_r$ (R3)	Yaw

These reserved names may be defined on the AEPARM entry instead, in which case the incremental load due to the unit perturbation of the rigid body degree-of-freedom (as it will with AESTAT). See the AEPARM, AEPRESS, and AEFORCE entries.

2. The degrees-of-freedom defined with this entry are variables in the static aeroelastic trim solution, unless they are constrained by referencing them with a TRIM Bulk Data entry.
3. If a label other than those above is specified, then the user must either generate the corresponding forces with an AELINK or via a DMI Bulk Data entry along with a DMAP alter that includes the DMIIN module and additional statements to merge into the appropriate matrices. Or, using AEPARM and the AEDW, AEPRESS, and/or AEFORCE, you can accomplish this purpose without the need for any alters.



## AESURF

## Aerodynamic Control Surface

Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points. The forces associated with this controller will be derived from rigid rotation of the aerodynamic model about the hinge line(s) and from AEDW, AEFORCE and AEPRESS input data. The mass properties of the control surface can be specified using an AESURFS entry.

Format:

1	2	3	4	5	6	7	8	9	10
AESURF	ID	LABEL	CID1	ALID1	CID2	ALID2	EFF	LDW	
	CREFC	CREFS	PLLIM	PULIM	HMLLIM	HMULIM	TQLLIM	TQULIM	

Example:

AESURF	6001	ELEV	100	100	200	200			
	10.0	180.0			-1.4E4	1.2E4	20	30	

Describer	Meaning
ID	Controller identification number. (Integer > 0)
LABEL	Controller name. (Character)
CIDi	Identification number of a rectangular coordinate system with a y-axis that defines the hinge line of the control surface component. (Integer > 0)
ALIDi	Identification of an AELIST Bulk Data entry that identifies all aerodynamic elements that make up the control surface component. (Integer > 0)
EFF	Control surface effectiveness. See Remark 4. (Real ≠ 0.0; Default = 1.0)
LDW	Linear downwash flag. See Remark 2. (Character, one of LDW or NOLDW; Default = LDW).
CREFC	Reference chord length for the control surface. (Real > 0.0; Default = 1.0)
CREFS	Reference surface area for the control surface. (Real > 0.0; Default = 1.0)
PLLIM,PULIM	Lower and upper deflection limits for the control surface in radians. (Real, Default = ± π/2)
HMLLIM,HMULIM	Lower and upper hinge moment limits for the control surface in force-length units. (Real; Default = no limit)
TQLLIM,TQULIM	Set identification numbers of TABLEDi entries that provide the lower and upper deflection limits for the control surface as a function of the dynamic pressure. (Integer > 0; Default = no limit)

**Remarks:**

1. The ID on AESURF, AESTAT, and AEPARM entries are ignored. AESURFS can be used to define mass properties of the control surface.
2. The degrees-of-freedom defined on this entry represent a rigid body rotation of the control surface components about their hinge lines. In the default LDW (Linear DownWash) case, the downwash due to a unit perturbation of the control surface will be computed as part of the database. In the NOLDW case, the user must prescribe the controller's effects by direct definition of the induced forces using the AEPRESS, AEDW and/or AEFORCE entries.
3. Either one or two control surface components may be defined.
4. If EFF is specified, then the forces produced by this surface are modified by EFF (e.g., to achieve a 40% reduction, specify EFF=0.60).
5. The continuation is not required.
6. The CREFC and CREFS values are only used in computing the nondimensional hinge moment coefficients.
7. Position limits may be specified using either PiLIM or TQiLIM, but not both.
8. Position and hinge moment limits are not required.

## AESURFS

### Structural Grids on an Aerodynamic Control Surface

Optional specification of the structural nodes associated with an aerodynamic control surface that has been defined on an AESURF entry. The mass associated with these structural nodes define the control surface moment(s) of inertia about the hinge line(s).

Format:

1	2	3	4	5	6	7	8	9	10
AESURFS	ID	LABEL		LIST1		LIST2			

Example:

AESURFS	6001	ELEV		6002		6003			
---------	------	------	--	------	--	------	--	--	--

Describer	Meaning
ID	Controller identification number, see Remark 1. (Integer > 0)
LABEL	Controller name, see Remark 1. (Character)
LISTi	Identification number of a SET1 entry that lists the structural grid points that are associated with this component of this control surface. (Integer > 0)

Remarks:

1. The LABEL on the AESURFS entry must match one on an AESURF entry. The ID is ignored.
2. The mass of the GRID points listed on the SETi entries is used to compute the mass moment of inertia of the control surface about its i-th hinge line. The presence of these data will allow the hinge moments to include the inertial forces in the computations. These data are optional, and, if omitted, result in hinge moments which include only the applied, aeroelastically corrected, forces.
3. These data will be associated to a structural superelement by grid list or partitioned SUPER=<seid> if the AESURFS is defined in the main bulk data section.

**AETRIA3**

## Triangular Aerodynamic Element Connection

Defines the connectivity of a triangular aerodynamic element.

Format:

1	2	3	4	5	6	7	8	9	10
AETRIA3	EID	CMPID	G1	G2	G3				

Example:

AETRIA3	3768	8	368	872	999				
---------	------	---	-----	-----	-----	--	--	--	--

Describer	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
CMPID	Aerodynamic Component Identification Number. (Integer > 0)
Gi	Grid Point Identification Numbers of Connection Points. (Integer > 0)

Remarks:

1. The AEGRID, AETRIA3, AEQUAD4, and AESCALE entries provide an aerodynamic mesh in a readable, portable format. There are no internal aerodynamics created on this mesh.

## ALIASM

## Alias Element Formulation in SOL 600

Allows selected elements which normally use a default formulation to be aliased to a different formulation in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
ALIASM	TYPE	ID1	THRU	ID2	BY	ID			

Example:

ALIASM	139	101	THRU	300	BY	2			
	138	501	THRU	600					

Describer	Meaning
ITYPE	Desired element formulation type. (Integer; no Default.) See Marc Vol B for a list of element types in the example, type 139 is a bilinear 3-node thin shell. ITYPE = -1 means to use the companion reduced integration element formulation (if it exists) default full integration formulation. (See Note 4)
ID1	Starting element number to be aliased. (Integer; no Default)
THRU	Enter "THRU" if a range of elements is to be specified. (Character; no Default)
ID2	Ending element number to be aliased. (Integer; no Default)
BY	Enter "BY" if the range of elements is not to be incremented by one.
ID3	Element "increment by" value. (Integer; Default = 1; must be positive)

Remarks:

1. This entry should only be used if the Marc GEOMETRY entries are identical for the original and new element types.
2. This entry may be repeated as often as desired to identify all elements requiring aliases.
3. ITYPE and ID1 are required fields. All other data fields may be blank.
4. This entry may not be combined with parameters MRALIAS, MALIAS02, MALIAS03, etc.
5. If all elements of a particular type should have alias values, it may be easier to use parameters MRALIAS, MALIAS02, MALIAS03, etc. than this entry.
6. For ITYPE = -1, the following reduced integration element types will be used:

Original Type	Reduced Integration Type	Description
26	53	Plane stress 8-node quad
27	54	Plane strain 8-node quad
28	55	Axisymmetric 8-node quad
29	56	Generalized plane strain 8+2 node quad
21	57	20-node brick
32	58	Plane strain Herrmann quad
33	59	Axisymmetric Herrmann 8-node quad
34	60	Plane strain Herrmann 8+2 node quad
35	61	Herrmann 20-node brick
62	73	Axisymmetric 8-node quad, arbitrary
63	74	Axisymmetric Herrman 8-node quad
3	114	Plane stress quad
11	115	Plane strain quad
10, 20	116	Axisymmetric ring (quad)
7	117	8-node brick
80	118	Plane strain incompressible quad
82, 83	119	Axisymmetric incompressible ring
84	120	8-node incompressible brick
75	140	4-node quad

## ASET Degrees-of-Freedom for the a-set

Defines degrees-of-freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

ASET	16	2	23	3516	1	4			
------	----	---	----	------	---	---	--	--	--

Describer	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive a-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.
4. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:
    - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## ASET1 Degrees-of-Freedom For the a-set, Alternate Form of ASET Entry

Defines degrees-of-freedom in the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
ASET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

ASET1	345	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

ASET1	C	ID1	“THRU”	ID2					
ASET1	123456	7	THRU	109					

Describer	Meaning
C	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; for THRU option, ID1 < ID2)

Remarks:

1. Degrees-of-freedom specified on this entry form members of the a-set that is exclusive from other sets defined by Bulk Data entries. See [Degree-of-Freedom Sets](#) for a list of these entries.
2. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
3. If the alternate format is used, all points in the sequence ID1 through ID2 are not required to exist, but there must be at least one degree-of-freedom in the a-set for the model, or a fatal error will result. Any points implied in the THRU that do not exist will collectively produce a warning message but will otherwise be ignored.
4. In nonlinear analysis, all degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, if the ASET or ASET1 entry is specified then all nonlinear degrees-of-freedom must be specified on the ASET or ASET1 entry.
5. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:



- a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
- b. Linear perturbation with:
  - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
  - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
- c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## ATBACC

## Acceleration Field applied to ATB Segments- SOL 700

Defines an acceleration field that will be applied to ATB segments. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
ATBACC	LID		SCALE	NX	NY	NZ			
+	NAME1	NAME2	NAME3	NAME4	NAME5	NAME6	NAME7	NAMEi	

Example:

ATBACC	32		386.088	1.0	0.0	0.0	0	0	
+	LT	MT	UT	N	H	RUL	RLL		

Field	Content
LID	Number of a set of loads. (Integer > 0; required)
SCALE	ATBACC scale factor. (Real ≥ 0.0; default=1.0)
NX, NY, NZ	Components of gravity vector. At least one component must be nonzero. (Real; default=0.0)
NAMEi	Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required)

Remarks:

- The acceleration is defined as:  

$$\alpha(t) = T(t).SCALE. N$$

where SCALE is the acceleration scale factor; is the vector defined by NX, NY, and NZ; is the value interpolated at time from the table referenced by the TLOADn entry.
- LID must be referenced by a TLOADn entry.
- The type field on the TLOADn entry must be set to zero.
- More than one ATBACC acceleration field can be defined per problem.
- This acceleration field is intended to apply a crash pulse to ATB segments that define a crash dummy. The acceleration is multiplied by the mass of the segment and the resulting force is added as an external force.
- To compare the accelerations of the ATB segments to experiments, the crash pulse needs to be subtracted from the total acceleration. The acceleration of the segments as defined on the H1 entries in the ATB input file are automatically corrected.

## ATBJNT Interface to ATB Joints- SOL 700

This entry can only be used together with the ATBSEG entries that this joint connects. The ATBSEG entries overwrite the position and orientation of the ATB segments as specified in the ATB input file. The ATBJNT entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment together with its joints. This visualization of the joints makes it possible to position the ATB model in any available preprocessor. Used in SOL700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
ATBJNT	ID	NAME							
+	G0	G1	G2	G3	EID1	EID2	EID3		
+	G4	G5	G6	G7	EID4	EID5	EID6		

### Example:

ATBJNT	1	HN							
+	1010	1011	1012	1013	1004	1005	1006		
+	2010	2011	2012	2013	2004	2005	2006		

Field	Content
LID	Unique ATBJNT number. (Integer > 0; required)
NAME	Name of an ATB joint as given in the first field of a B.3 entry in the ATB input file. (Character; required)
G0-G7	An ATB joint connects two segments. A local joint coordinate system is attached to each of these two segments. The position and orientation of these two coordinate systems relative to the segment coordinate systems is given on entry B.3 in the ATB input file. For each joint (J = 1, NJNT) a B.3 entry is defined in the ATB input file. The joint J connects the segment JNT(J) as given on the B.3 entry and the segment J + 1. SOL700 finds the two segments that are connected by the joint with name = NAME. The grid points G0-G3 and G4-G7 define the joint coordinate systems for the segments JNT(J) and J + 1, respectively. (Integer > 0; required)
	G0 located at the origin of the joint coordinate system for the ATB segment JNT (J)
	G1 located on the local x-axis.
	G2 located on the local y-axis.
	G3 located on the local z-axis.
	G4 located at the origin of the joint coordinate system for the ATB segment J + 1.

Field	Content
G5	located on the local x-axis.
G6	located on the local y-axis.
G7	located on the local z-axis.

**Remarks:**

All elements related to an ATB segment refer to the same material number. This material number is defined on the ATBSEG entry. If the material is defined to be rigid by means of a MATRIG entry, all elements can be easily connected to the contact ellipsoid of the ATB segment by means of METHOD=RELLIPS in BCTABLE or BCONPRG entry referencing the MATRIG entry. In this way, all elements related to an ATB segment move together with the ATB segment during the analyses and can be postprocessed.

**ATBSEG**

Interface to ATB Segments- SOL 700

Defines the position and orientation of the ATB segments. The position and orientation as specified on the G.2 and G.3 entries in the ATB input file will be overruled by the definitions given here.

This entry can be used to create a Bulk Data file containing elements and grid points to visualize the ATB segment, together with the contact ellipsoid and the joints it is connected by. See also ATBJNT and DYPARAM, ATBSEG. Used in SOL700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
ATBSEG	ID	NAME	COVER	NUMELM	GSTART	ESTART	MID	PIDCOV	
+	G0	G1	G2	G3	EID1	EID2	EID3	PIDCG	

**Example:**

ATBSEG	1	HEAD	YES	100	1000	1000	1000		
+	1010	1011	1012	1013	1001	1002	1003	1001	

Field	Content
ID	Unique ATBSEG number. (Integer > 0; required)
NAME	Name of an ATB segment as given in the first field of a B.2 entry in the ATB input file. (Character; required)
G0-G3	The grid points span the local coordinate system of the ATB segment. G0-G3 are used by SOL700 to overwrite the initial position and orientation of the segments as specified in the ATB input file. EID1-EID3 are about how to generate the grid points for an existing ATB input file. (Integer > 0; required)
G0	located at the origin of the ATB segment.
G1	located on the local x-axis.
G2	located on the local y-axis.
G3	located on the local z-axis.



## AXIC Conical Shell Problem Flag

Defines the existence of an axisymmetric conical shell problem.

Format:

1	2	3	4	5	6	7	8	9	10
AXIC	H								

Example:

AXIC	15								
------	----	--	--	--	--	--	--	--	--

Describer	Meaning
H	Highest harmonic defined for the problem. ( $0 \leq \text{Integer} < 998$ )

Remarks:

- Only one AXIC entry is allowed. When the AXIC entry is present, most other entries are not allowed. The types that are allowed with the AXIC entry are listed below:

CCONEAX	MATT1	SPCADD
DAREA	MOMAX	SPCAX
DELAY	MOMENT	SUPAX
DLOAD	MPCADD	TABDMP1
DMI	MPCAX	TABLED1
DMIG	NOLIN1	TABLED2
DPHASE	NOLIN2	TABLED3
EIGB	NOLIN3	TABLED4
EIGC	NOLIN4	TABLE11
EIGP	NSM	TABLEM2
EIGR	NSM1	TABLEM3
EIGRL	NSMADD	TABLEM4
EPOINT	OMITAX	TEMPAX
FORCE	PARAM	TF
FORCEAX	PCONEAX	TIC
FREQ	POINTAX	TLOAD1
FREQ1	PRESAX	TLOAD2
FREQ2	RINGAX	TSTEP

GRAV	RFORCE
LOAD	RLOAD1
MAT1	RLOAD2
MAT2	SECTAX

2. For a discussion of the conical shell element, see the [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

## AXIF

## Fluid Related Axisymmetric Parameters

Defines basic parameters and the existence of an axisymmetric fluid analysis.

Format:

1	2	3	4	5	6	7	8	9	10
AXIF	CID	G	DRHO	DB	NOSYM	F			
	N1	N2	N3	N4	N5	-etc.-			

Example:

AXIF	2	32.2	0.12	2.4+5	YES				
	1	2	3		4		7	10	

Alternate Formats and Examples of Continuation Data:

	N1	"THRU"	Ni						
	0	THRU	10						

	N1	"THRU"	Ni	"STEP"	NS				
	0	THRU	9	STEP	3				

AXIF	100	-386.0		0.0	NO				
	0	THRU	50	STEP	5				
	52								
	54	THRU	57						
	61	THRU	65						
	68		71		72	75			
	81	92							

Describer	Meaning
CID	Fluid coordinate system identification number. (Integer > 0)
G	Value of gravity for fluid elements in the axial direction. (Real)
DRHO	Default mass density for fluid elements. (Real > 0.0 or blank)
DB	Default bulk modulus for fluid elements. (Real)
NOSYM	Request for nonsymmetric (sine) terms of series. (Character: "YES" or "NO")
F	Flag specifying harmonics. (Blank if harmonic is specified, or Character: "NONE")



Describer	Meaning
Ni	Harmonic numbers for the solution, represented by an increasing sequence of integers. On continuation entries, without the “THRU” option, blank fields are ignored. “THRU” implies all numbers including upper and lower harmonics. (0 ≤ Integer < 100, or Character: “THRU”, “STEP” or blank)
NS	Every NSt <sup>h</sup> step of the harmonic numbers specified in the “THRU” range is used for solution. If field 5 is “STEP”, Ni = i*NS+N1 where i is the number of harmonics. (Integer)

Remarks:

1. Only one AXIF entry is allowed.
2. CID must reference a cylindrical or spherical coordinate system.
3. Positive gravity (+G) implies that the direction of free fall is in the -Z direction of the fluid coordinate system.
4. The DRHO value replaces blank values of RHO on the FSLIST, BDYLIST and CFLUIDi entries.
5. The DB value replaces blank values of B on the CFLUIDi entries. If the CFLUIDi entry is blank and DB is zero or blank, the fluid is incompressible.
6. IfNOSYM = “YES”, both sine and cosine terms are specified. IfNOSYM = “NO”, only cosine terms are specified.
7. If F = “NONE”, no harmonics are specified, no fluid elements are necessary, and no continuations may be present. In this case, AXIS = “FLUID” should not be specified in the Case Control Section.
8. Superelements cannot be used.

## AXSLOT

## Axisymmetric Slot Analysis Parameters

Defines the harmonic index and the default values for acoustic analysis entries.

Format:

1	2	3	4	5	6	7	8	9	10
AXSLOT	RHOD	BD	N	WD	MD				

Example:

AXSLOT	0.003	1.5+2	3	0.75	6				
--------	-------	-------	---	------	---	--	--	--	--

Descriptor	Meaning
RHOD	Default density of fluid in units of mass/volume. (Real > 0.0 or blank)
BD	Default bulk modulus of fluid in units of force/volume ratio change. (Real ≥ 0.0 or blank)
N	Harmonic index number. (Integer ≥ 0 or blank)
WD	Default slot width. (Real > 0.0 or blank)
MD	Default number of slots. (Integer ≥ 0 or blank)

Remarks:

1. Only one AXSLOT entry is allowed.
2. If any of the RHO, B, W, and M fields on the GRID, SLBDY, CAXIFi, and CSLOTi entries are blank, then values must be specified for the RHOD, BD and MD fields.
3. If the number of slots (M) is different in different regions of the cavity, this fact may be indicated on the CSLOTi and SLBDY entries. If the number of slots is zero, no matrices for CSLOTi elements are generated.
4. BD = 0.0 implies the fluid is incompressible.

## BAROR      CBAR Entry Default Values

Defines default values for field 3 and fields 6 through 8 of the CBAR entry.

Format:

1	2	3	4	5	6	7	8	9	10
BAROR		PID			X1	X2	X3	OFFT	

Example:

BAROR		39			0.6	2.9	-5.87	GOG	
-------	--	----	--	--	-----	-----	-------	-----	--

Alternate Format and Example:

BAROR		PID			G0			OFFT	
-------	--	-----	--	--	----	--	--	------	--

BAROR		39			18			GOG	
-------	--	----	--	--	----	--	--	-----	--

Describer	Meaning
PID	Property identification number of the PBAR entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ , using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBAR entry)
OFFT	Offset vector interpretation flag. See Remark 5. (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBAR entry whose corresponding fields are blank.
2. Only one BAROR entry is allowed.
3. For an explanation of bar element geometry, see the [Three-Node Beam Element \(CBEAM3\)](#) in the *MSC Nastran Reference Guide*.
4. If field 6 is an integer, then G0 is used to define the orientation vector and X2 and X3 must be blank. If field 6 is real or blank, then X1, X2, and X3 are used.
5. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character “O” in the table replaces the obsolete character “E”.)

## BARRIER

## Barrier for Eulerian Transport

Defines a barrier for transport in a Eulerian mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BARRIER	BID	BCID	MESH	DIR	SKFRIC				
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			

Example:

BARRIER	100	20							

Describer	Meaning
BID	Unique barrier number. (Integer > 0; Required)
BCID	Number of a set of BCSEG entries that define the element faces that are barriers to Eulerian transport. See Remark 4. (Integer $\geq 0$ )
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 5. (Integer $\geq 0$ )
DIR	Allowed values are NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 6. (Character)
SKFRIC	Skin friction value. See Remark 8. (Real $\geq 0.0, 0.0$ )
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 7. (Real)

Remarks:

1. Material cannot pass through any of the faces referenced.
2. Barriers can be modeled on the edge as well as the inside of an Eulerian mesh.
3. BARRIER overrules FLOW and FLOWT definition.
4. BCID is optional. If used, all other inputs are ignored. If not used, the barrier can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
5. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
6. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.

7. XMIN, XMAX, YMIN, etc are only used when both BCID and DIR are blank. The XMIN,YMIN option defines an area on the MESH BOX boundary as shown in Figure 9-1. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1e+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.
8. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here  $\tau_w$  denotes the shear friction in an Euler element adjacent to a couple surface segment where  $\rho$  is the density and  $u$  is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment. SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

9. Internal barriers can only be defined on existing Euler element faces using the BCID option. They cannot be defined by using the XMIN,YMIN option.

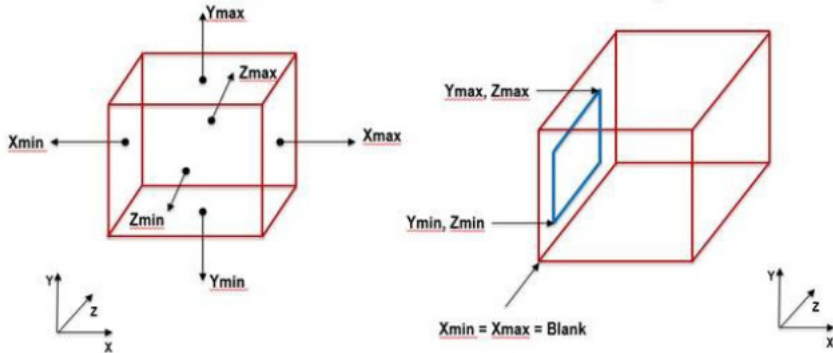


Figure 9-1 MESH BOX boundary

## BCAUTOP

## Automatic contact generation property.

Specifies parameters for automatic contact generation (ACG). With ACG, the code automatically generates the contact bodies based on the grids and elements given, then establishes the contact pairs in which two contact bodies are in contact or may come into contact potentially based on the DISTANCE tolerance. Please refer to the section of [Chapter 9: Contact](#), SOL 400 User's Guide for details.

### Format:

1	2	3	4	5	6	7	8	9	10
BCAUTOP	PID		Param1	Value1	Param2	Value2	Param3	Value3	
	Param4	Value4	etc						

### Example:

BCAUTOP			CTYPE	PGLUE	DISTANCE	0.1			
---------	--	--	-------	-------	----------	-----	--	--	--

Describer	Meaning
PID	Reserved for future use (integer >=0, default is 0)
Param(i)	Name of a parameter. Allowable names are given below (Character).
Value(i)	Value of the parameter. See below (Real, Integer or character). See Remark 3.

### Remarks:

1. Only one entry of BCAUTOP is allowed for non-modules model. If there are modules, only one BCAUTOP per module is allowed, and multiple modules can have BCAUTOP.
2. This entry does not have effect if BCONTACT=AUTO is not present in case control section.
3. The parameters in this entry may be divided into two types – primary parameters and secondary parameters. The primary parameters are the most important parameters for build of contact bodies, match of contact pairs. The secondary parameters are helpful for user to provide the more detail and further requirement for contact model establishment and contact analysis with automatic contact generation. The primary and secondary parameters are listed in the tables as follows.
4. When modules are present, contact data is generated for the module if there is a BCAUTOP exists in the module, otherwise no contact data will be generated for the module, in this case, user supplied contact data can be used if there is a BCTABL1 ID=1 and/or ID=0 exists.

Table 9-1 Primary Parameters in BCAUTOP

Describer	Meaning
CTYPE	Characters, one of TOUCH(default), PGLUE, SGLUE and GGLUE. If CTYPE is specified in both BCAUTOP entry and Case Control Command BCONTACT=AUTO, the specification in BCAUTOP entry will be used.
DISTANCE	Distance tolerance of contact pair. If the distance between any two points which are belonging to two different contact bodies is less than this value, these two contact bodies are recognized to be a contact pair. Default value of DISTANCE is 100 times of contact ERROR tolerance. ERROR may be defined in either BCAUTOP or left blank.  With default of ERROR, the code calculates ERROR as the smallest one of the following values: <ol style="list-style-type: none"> <li>1. 1/20 of the smallest nonzero element dimension (plates or solids) in the contact body;</li> <li>2. 1/4 of the thinnest shell thickness in the contact body.</li> </ol> Please see BCPARA for ERROR definition in detail.
ESET	ID of SET1 to specify element ids consisting contact bodies. If it is not specified, all elements in the model will be used. Only one of ESET, SeedESet, SeedGSet and PropSet can be specified in the same entry.
SeedESet	ID of SET1 to specify element ids as seeds of contact bodies. The program searches elements as contact bodies from the seed elements. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
SeedGSet	ID of SET1 to specify grids ids as seeds of contact bodies. The program searches elements as contact bodies from the elements connecting to the seed grid ids. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
PropSet	ID of SET1 to specify property ids to construct contact bodies. Only one of ESET,SeedESet, SeedGSet and PropSet can be specified.
RigidSet	ID of SET1 to specify IDs of rigid bodies to be in contact



Table 9-2      Secondary Parameters in BCAUTOP

Describer	Meaning
BEAMCNT	<p>Characters, YES or NO (default). Determining if beam is included in contact definition.</p> <p>YES: BEAMs are included in the contact description.</p> <p>NO: No BEAM is included in the contact description.</p>
BEAMCAP	<p>Characters, FREEEND(default) or NO. If set to FREEEND(default), BCSCAP entries will be added to free end of beam elements, if NO, no BCSCAP will be added. It is mainly used for contact pair detection where the contact occurs at free beam end.</p>
DIMENS	<p>Characters, 2D or 3D (default), dimension of contact bodies.</p>
EDGECONT	<p>Characters, YES or NO (default). Determining if shell edge is included in contact description.</p> <p>YES: The free and hard shell edges are included in the contact definition</p> <p>NO: No shell edges are included in the contact definition.</p>
FTYPE	<p>Characters, BLCOUL/BLSHEAR; or Integer, 6 / 7. BLCOUL or 6 is Bilinear Coulomb friction. BLSHEAR or 7 is Bilinear Shear friction. Default is No Friction.</p>
FRIC	<p>Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real <math>\geq 0.0</math> or Integer <math>&gt; 0</math>; Default = 0.0)</p>
IGNTHK	<p>Character, YES or NO (default), Ignore thickness of shell for contact</p> <p>YES: ignore the thickness of the shell</p> <p>NO: include the thickness of the shell.</p>
INISTF	<p>Character, YES or NO (default). Set the option of initial stress free.</p> <p>Yes: initial stress free contact</p> <p>No: general contact without implementation of initial stress free.</p>
SelfCont	<p>Characters, YES or NO (default). Option of self-contact.</p>

**Note:** Note that the parameters defined in BCAUTOP are applied only for BONATCT=AUTO. Their default values are defined only when BCONTACT=AUTO is applied. All the parameters in BCPARA, BCONPRP, BCONPRG and BCBDPRP entries may be used in BCAUTOP directly. For the detailed list of these parameters, please refer to BCPARA, BCONPRP, BCONPRG and BCBDPRP.

## BCBDPRP Contact Body Parameters in SOLs 101 and 400

Defines contact body parameters used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCBODY1 entry.

Format:

1	2	3	4	5	6	7	8	9	10
BCBDPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCBDPRP	90		FRIC	0.05	ISTYP	0			
---------	----	--	------	------	-------	---	--	--	--

Describer      Meaning

PID              Parameter identification number (Integer > 0).

PARAMi        Name of a parameter. Allowable names are given in [Table 9-3](#) (Character).

VALi            Value of a parameter. See [Table 9-3](#) (Real or Integer).

Table 9-3 Contact Body Parameters in SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
BNC	Exponent associated with the natural convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables.
BNCE	Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables.
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables.
BNLE	Exponent associated with the nonlinear convective heat flow to the environment. (Real or Integer; Default = 1.0) If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.
CFILM	Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.
CMB	Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real $\geq 0.0$ ; Default = 0.0) For nonzero values, a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
CMS	Heat capacity of the environment, when associated with a scalar point. (Real $\geq 0.0$ ; Default = 0.0) For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
COPTB	Flag to indicate how body surfaces may contact. See Remark 2. on the BCONPRG entry. (Integer; Default = 0)

EMISS	Emissivity for radiation to the environment or near thermal radiation. (Real or Integer; Default = 0.0) If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq 0.0$ or Integer $> 0$ ; Default = 0.0)  When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.  In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCONNECT/BCONPRP is strongly recommended.
HBL	Separation distance dependent thermal convection coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the separation distance dependent thermal convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.
HCT	Contact heat transfer coefficient. (Real or Integer; Default = 0.0) If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. See Remark 3.
HCV	Convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.
HNC	Natural convection coefficient for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.
HNCE	Natural convection coefficient for heat flow to the environment. (Real or Integer, Default = 0.0) If Real, the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0) If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.

HNLE	Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real $\geq 0.0$ or Integer; Default = 0.0) If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.
IDSPL	Controls geometric smoothing of boundary of deformable body option. (Integer; Default = 0) =0 or blank, discrete geometric representation. >0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs (IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 1. <0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE.
ISTYP	<a href="#">Check</a> of contact conditions. (Integer $\geq 0$ ; Default = 0 for all solutions) ISTYP is not necessary in segment-to-segment contact. For a deformable body: =0 check each body, versus the other. =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”).  Note that ISTYP is supported with ISEARCH=0 in <a href="#">BCTABLE</a> or <a href="#">BCONPRG</a> only.
ITYPE	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink (rigid body) 2 - Deformable body (with heat conduction) 4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis, i.e. the same contact body with ITYPE=4 cannot be used in mechanical analysis of SOL 400.)

MIDNOD	Mid-side node projection flag. (Integer $\geq 0$ ; Default = 0) When MIDNOD > 0 and IDSPL $\neq 0$ , the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact. Only used if IDSPL is not zero.
SANGLE	<b>Threshold</b> for automatic discontinuity detection in degrees. (Real; Default = 60.0) Used for geometric smoothing option in SOL 400 only. SANGLE is not used and is always set to 0.0 when IDSPL $\geq 0$ .
TBODY	Body temperature. (Real or Integer; Default = 0.0) If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer, its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8.
TSINK	Environment sink temperature. (Real or Integer, Default = 0.0) If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry, it need not be defined on an SPOINT entry. See Remark 8.

Remarks:

1. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See *MSC Nastran Nonlinear User's Guide*, Chapter 9, Contact for more details.
2. For hard contact, with HGLUE=1 (see BCONPRP for the meaning of HGLUE):
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
  - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** "Glued" thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

3. For hard contact, with HGLUE=0:  
The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{\text{BODY}}$  temperature in case of a rigid geometry.

It is recommended to enter the HCT value on the BCONPRP Bulk Data entry, since it generally applies to body pairs and is not a property of a single body.

4. For near contact:

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot \text{EMISS} \cdot (T_A^A - T_B^A) +$$

$$\left[ HCT \cdot \left( 1 - \frac{\text{dist}}{DQNEAR} \right) + HBL \cdot \frac{\text{dist}}{DQNEAR} \right] (T_A - T_B)$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{\text{BODY}}$  temperature in case of a rigid geometry.

It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCONPRP Bulk Data entry, since they generally apply to body pairs and are not properties of a single body.

5. For no contact:

$$q = CFILM \cdot (T_A - T_{\text{SINK}}) +$$

$$HNCE \cdot (T_A - T_{\text{SINK}})^{BNCE} +$$

$$HNLE \cdot (T_A^{BNLE} - T_{\text{SINK}}^{BNLE}) +$$

$$\sigma \cdot \text{EMISS} \cdot (T_A^A - T_{\text{SINK}}^A)$$

6. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
7. The  $T_{\text{SINK}}$  and  $T_{\text{BODY}}$  temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).

8. **TBODY** entries only apply to rigid bodies (i.e., **RIGID** as the **BEHAV** value in **BCBODY1** entry). **TSINK** entries only apply to deformable bodies (meshed regions with **DEFORM** or **HEAT** as the **BEHAV** value in **BCBODY1** entry).



## BCBMRAD

### Modify Equivalent Radius for Beam-to-Beam Contact

Allows the equivalent radius in beam-to-beam contact to be different for each beam cross section in SOL 101, SOL 400 and SOL 600. The BCBMRAD entry is only used for node-to-segment beam-to-beam contact. For segment-to-segment beam contact, the beam cross section geometry is defined via PBARL or PBEAML

Format:

1	2	3	4	5	6	7	8	9	10
BCBMRAD	RADIUS	TYPE	ID1	ID2	THRU	ID3	BY	N	
	ID4	THRU	ID5	ID6	ID7	ID8	ID9		

Example 1:

BCBMRAD	2.5	EID	100	20	THRU	300	BY	2	
	200	3457	8456	4712	1000	THRU	2000		
	3.0		4112	THRU	4700				
	2.8	BODY	502	517	3459				

Example 2:

BCBMRAD	2.5	ALL							
	2.8	EID	2567	1240	THRU	1760			

Describer	Meaning	
RADIUS	Equivalent radius to be used for beam-beam contact problems. See Remark 6. (Real; no Default)	
TYPE	The attribute of all following ID's. (Character; Default = "EID")	
	EID	Defines all the following entries are the IDs of beam-type elements.
	BODY	Defines all the following entries are the IDs of BCBODYs.
	ALL	Defines the default RADIUS for all beam-type elements.
ID <sub>i</sub>	ID of a beam-type element, CROD, CBAR, CBEAM and CBEAM3, or a BCBODY with the specified radius. (Integer; no Default)	

Remarks:

1. Multiple BCBMRAD Bulk Data entries, which are open-ended are allowed in one file.
2. In each entry of BCBMRAD, there is only one RADIUS input allows, on the field 2. From the field 4 to the rest fields, including all continuation entries, user can input all ID's in any combination of the following 3 basic formats
  - ID1 ID2 ID3 ID4 ...
  - ID1 THRU ID2

- ID1 THRU ID2 BY N

Note that blank fields are allowed for readability.

- When all beam contact radii are the same, user can put "ALL" on the field 3 following a RADIUS value on the BCBMRAD Bulk Data entry. This value of RADIUS will be applied to all beam-type elements (See the previous Example 2). User can also use this way to give default radius of all beam-type elements.
- The RADIUS value of the other BCBMRAD's (TYPE=BODY or EID) can override the default value (TYPE=ALL) in the following order:
  - When any BCBODY is selected on BCBMRAD with TYPE=BODY, the specified RADIUS will be applied to all beam-type elements in this BCBODY. This value always overrides the default (TYPE=ALL).
  - The RADIUS with TYPE=EID always overrides the value from TYPE=BODY and ALL.
- When TYPE=ALL, then all IDi must be blank.
- For tubes or round bars, enter the outer radius. For beam, enter an equivalent radius calculated as follows:

$$I = 0.5 \cdot (I_x + I_y)$$

$$R = \sqrt{\left(\frac{A}{\pi} + 2 \cdot \frac{I}{A}\right)}$$

- SOL 600 does not support the TYPE=ALL option.
- This is only used for node-to-segment contact.

## BCBODY

## Flexible or Rigid Contact Body in 2D and 3D

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only. The BCBODY1 option is the preferred method to defining contact bodies for SOL 400.

Use only as many forms (i.e. HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The "RIGID" header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	"ADVANCE"	SANGLE	COPTB		MIDNO D				
	"RIGID"	CGID	NENT	--- Rigid Body Name ---					
	"APPROV"	A	N1	N2	N3	V1	V2	V3	
	"GROW"	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		
	"HEAT"	CFILM	TSINK	CHEAT	TBODY	HCV	HNC	ITYPE	
		BNC	EMISS	HBL	HNL	BNL	HNLE	BNLE	
		HNCE	BNCE	CMB	CMS				
	"PATCH3D"	NPATCH							
		IDP	G1	G2	G3	G4			
		IDP	G1	G2	G3	G4			
		etc.	(npatch entries)						
	"BEZIER"	NP1	NP2	NSUB1	NSUB2				
		G1	G2	G3	G4	etc		(np1*np2 values)	
	"NURBS2D"	NPTU	NORU	NSUB				(2D Contact)	
		G1 or X1	G2 or Y1	G3	G4 or X2	G5 or Y2	G6	[abs(nptu) grids or x, y,z, values]	See Remark 8

		Homo1	Homo2	Homo3	Homo4	etc.		(nptu values)	
		Knot1	Knot2	Knot3	Knot4	Knot5	etc.	(nptu+noru values)	
	"NURBS"	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	See Remark 8
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*nptv values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	etc.			(nptu*nptv vales)		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	etc.			(nptu+noru+ntpv+norv values)		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc.	(NPTUtrim + NORUtrim entries)		

Format: (SOL 700 only)

BCBODY	BID	DIM	BEHAV	BSID					
--------	-----	-----	-------	------	--	--	--	--	--

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

BCBODY	1		DEFORM	101	0	.05			
--------	---	--	--------	-----	---	-----	--	--	--

Example 2 -- Simple 4-node rigid patch (see Remark 4. for rigid bodies)

BCBODY	2		RIGID	102	0	.08			
	PATCH3D	1							
		1	101	102	103	104			

Describer	Meaning
BID	Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)
DIM	Dimension of body. (Character; Default= 3D) (Ignored by SOL 700) DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.
BEHAV	Behavior of curve or surface (Character; Default = DEFORM) (Ignored by SOL 700) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, HEAT indicates body is a heat-rigid body. <i>See Remark 4. for Rigid Bodies.</i>
BSID	Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM. For SOL 700 the BSID may also be the identification number of a BCBOX, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0)
ISTYP	Check of contact conditions. (Integer $\geq$ 0; Default = 0 for all solutions) ISTYP is not necessary in segment-to-segment contact. For a deformable body: =0 check each body, versus the other . =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”).  Note that ISTYP is supported with ISEARCH=0 in <a href="#">BCTABLE</a> or <a href="#">BCONPRG</a> only.
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq$ 0.0 or Integer > 0; Default = 0.0)  When a grid point contacts a rigid body, the coefficient of friction associated with the rigid body is used. When a grid point contacts a deformable body, the average of the coefficients for the two bodies are used.  In general, entering the friction coefficient for a contact body pair via a BCTABLE and BCONNECT/BCONPRP is strongly recommended.
IDSPL	Controls geometric smoothing of boundary of deformable body. (Integer; Default = 0) =0 or blank, discrete geometric representation >0 the surface of the body is smoothed out with splines (2D) or Coons surfaces (3D) and discontinuity edges/corners are being defined by using abs (IDSPL) as the ID of the BLSEG entries. If BLSEG with ID=abs(IDSPL) does not exist, the whole body is smoothed and there are no user-defined discontinuity corners(2D) or edges(3D). (See Remark 7.) <0 Same as IDSPL>0. Furthermore, additional discontinuity edges are being generated automatically if the difference in patch normals exceeds the value of SANGLE.

Descriptor	Meaning
CONTROL	<p>= -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRB<sub>i</sub> in the following line.</p> <p>= 0 for velocity control (default).</p> <p>= positive number for "load control". The positive number entered is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)</p>
NLOAD	<p>Enter a positive number if "load controlled" and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for rotation about x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively).</p> <p>Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation.</p>
ANGVEL	Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)
DCOS <sub>i</sub>	Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.
VELRB <sub>i</sub>	Translation velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID field. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0)
"ADVANCE"	The entries for this continuation line are for advanced options starting with MD Nastran R2.
SANGLE	Threshold for automatic discontinuity detection in degrees. (Real; Default = 60.0) Used for SPLINE option in SOL 400 only. SANGLE is not used when IDSPL ≥ 0.
COPTB	Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)
MIDNOD	Mid-side node projection flag. (Integer ≥ 0; Default = 0) When MIDNOD > 0 and IDSPL ≠ 0, the mid-side grid of quadratic elements are projected onto the selected spline surfaces. This operation is performed before the contact process starts and it may change the location of grids in contact bodies. It may operate in combination with the initial stress-free contact.
"RIGID"	The entries of this continuation line are for the rigid body description. See Remark 4.

Describer	Meaning
CGID (5,i) i=1,2,3	Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied
NENT	Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)
Rigid Body Name	Name associated with the rigid body. (Default is blank, 24-characters maximum)
“APPROV”	The entries of this continuation line are for approaching velocity to establish initial contact.
A	Angular velocity about local axis through center of rotation. (Real, Default = 0.0)
Ni	Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the “RIGID”, CGID entry. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)
Vi	V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)
“GROW”	The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2(growth vs time) or TABL3D (growth vs one or more variables).
GFi	Components of Growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Real, Default = 1.0)
TAB-GFi	Tabled IDs for growth factor of rigid body in the coordinate system of the “RIGID”, CGID entry. (Integer > 0 or blank, Default is blank)
“HEAT”	The entries of this continuation line(s) are for contact in heat transfer in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.
CFILM (9,1)/(10,1)	Heat transfer coefficient (film) to environment. (Real or Integer, Default = 0.0) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the heat transfer coefficient vs temperature or a TABL3D entry specifying the film coefficient vs temperature and possibly other variables.
TSINK (9,2)/(10,2)	Environment sink temperature. (Real or Integer, Default = 0.0). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying temperature vs time or a TABL3D entry specifying the sink temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry.

Descriptor	Meaning
CHEAT (9,3)/(10,3)	Contact heat transfer coefficient. (Real or Integer; Default = 0.0). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables.
TBODY (9,4)/(10,4)	Body temperature. (Real or Integer; Default = 0.0). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLED1 or TABLED2 entry specifying the body temperature vs time or a TABL3D entry specifying the body temperature vs time and possibly other variables. When entered as a negative integer its absolute value is a scalar point identification number. If a scalar point is specified on this entry it need not be defined on an SPOINT entry.
HCV (9,5)/(10,5)	Convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables.
HNC (9,6)/(10,6)	Natural convection coefficient for near field behavior (Real or Integer; Default = 0.0). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables.
ITYPE [4,8]	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink (rigid body) 2 - Deformable body (with heat conduction) 4 - Heat conduction body (heat-rigid body) (Not supported in SOL 400 coupled thermal-mechanical analysis)
BNC (9,7)/(10,7)	Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables.
EMISS (9,8)/(10,8)	Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables.



Describer	Meaning
HBL (7,6)/(8,6)	Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0). If Real, the value entered is the separation distance dependent thermal a convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables.
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 0.0). If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables.
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables.
HNLE	Heat transfer coefficient for nonlinear convective heat flow to the environment. (Real $\geq 0.0$ or Integer; Default = 0.0). If Real, the value entered is the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the environment nonlinear convection coefficient vs temperature and possibly other variables.
BNLE	Exponent associated with the nonlinear convective heat flow to the environment (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment nonlinear convection coefficient vs temperature and possibly other variables.
HNCE	Natural convection coefficient for heat flow to the environment. (Real or Integer, Default = 0.0). If Real the value entered is the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the environment natural convection coefficient vs temperature or a TABL3D entry specifying the environment natural convection coefficient vs temperature and possibly other variables.

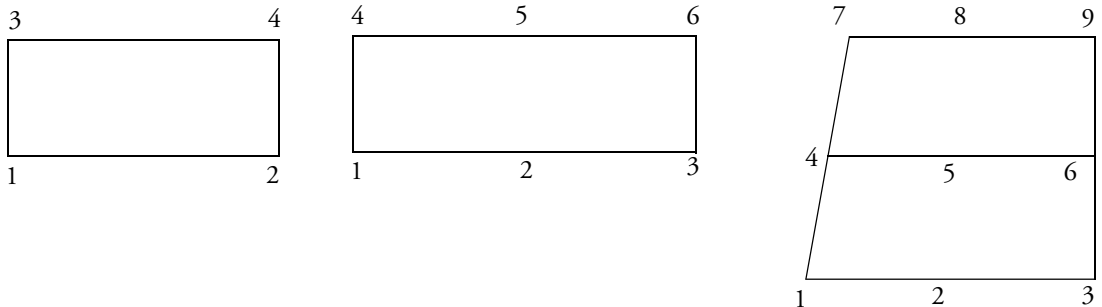
Descriptor	Meaning
BNCE	Exponent associated with natural convection heat flow to the environment. (Real or Integer; Default = 1.0). If Real, the value entered is the exponent associated with the environment natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the environment natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the environment natural convection coefficient vs temperature and possibly other variables.
CMB	Heat capacity of the rigid body, when entered as a geometric entity with an associated scalar point. (Real $\geq 0.0$ ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
CMS	Heat capacity of the environment, when associated with a scalar point. (Real $\geq 0.0$ ; Default = 0.0). For nonzero values a grounded CDAMP4 element with the scalar point on its first side is generated internally which obtains this capacity.
“PATCH3D”	Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch (see Note 5).
“BEZIER”	Entries for this continuation line describe a rigid body made up of Bezier Surfaces.
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
“NURBS2D”	Entries for this continuation line describe a 2D rigid body made up of nurbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer, no Default)
NORU	Order
NSUB	Number of subdivisions
G1, G2, G3, G4	Grid numbers for each of the NPTU control points
X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.

Describer	Meaning
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.
“NURBS”	Entries for this continuation line describe a rigid body made up of nurbs.
NPTU	Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer > 0; Required)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction (Integer > 0; Required)
NTRIM	Number of trimming curves (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space. (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

Remarks:

1. Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
2. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
3. All continuation lines may be omitted if not required.
4. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
5. For BEZIER surfaces, enter np1\*np2 points in the order shown below:

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value.
7. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
8. For hard contact, with HGLUE=1 (see BCTABLE for the meaning of HGLUE):
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.

- b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

9. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = CHEAT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry. It is recommended to enter the CHEAT value on the BCTABLE Bulk Data input, since it generally applies to body pairs and is not a property of a single body.

10. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BNL} - T_B^{BNL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[ CHEAT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry. It is recommended to enter the near contact heat transfer coefficients and the corresponding exponents on the BCTABLE Bulk Data input, since they generally apply to body pairs and are not properties of a single body.

11. For no contact:

$$\begin{aligned}
 q = & CFILM \cdot (T_A - T_{SINK}) + \\
 & HNCE \cdot (T_A - T_{SINK})^{BNCE} + \\
 & HNLE \cdot (T_A^{BNLE} - T_{SINK}^{BNLE}) + \\
 & \sigma \cdot EMISS \cdot (T_A^4 - T_{SINK}^4)
 \end{aligned}$$

12. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
13. The  $T_{SINK}$  and  $T_{BODY}$  temperatures can be time dependent when they are entered as a positive integer value. This integer value is the table ID of a TABLED1, TABLED2 or TABL3D entry (formulas are not supported on TABL3D).
14. Table IDs of tables used on the BCBODY and the BCTABLE entry must be unique.
15. TBODY entries only apply to rigid bodies (i.e., RIGID as the BEHAV value). TSINK entries only apply to deformable bodies (meshed regions with DEFORM or HEAT as the BEHAV value).

**BCBODY****Flexible or Rigid Contact Body in 2D and 3D in SOL 600**

Defines a flexible or rigid contact body in 2D or 3D used in SOL 600 only.

**Important Notes for SOL 600:**

1. The 2nd line is required if any of the subsequent lines are to be entered (if this line is blank, enter a “+” in column 1 or Nastran will ignore the entire line).
2. PARAM,MRCONVER,11 is required if CTYPE, APPROV, or GROW headers are entered.
3. PARAM,MRCONTAB,11 is required if any of the variables are described by tables.

Use only as many forms (i.e., HEAT, PATCH3D, BEZIER, POLY, CYLIND, SPHERE, NURBS2, or NURBS) as necessary to describe the body (if rigid). Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM (only the first line should be entered for deformable bodies). Unless shrink fit its being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

The “RIGID” header may be used with any of the other rigid entries but only once per body. Also, only one of the character entries after RIGID (HEAT, PATCH3D, NURBS, etc.) should be entered for any particular body. See Remark 4. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

**Format:**

1	2	3	4	5	6	7	8	9	10
BCBODY	BID	DIM	BEHAV	BSID	ISTYP	FRIC	IDSPL	CONTROL	
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	“ADVANCE”	SANGLE	COPTB	USER					
	“CTYPE”	ISMAIL	ITYPE	IAUG	PENALT	AUGDIST			
	“RIGID”	CGID	NENT	--- Rigid Body Name ---					
	“APPROV”	A	N1	N2	N3	V1	V2	V3	
	“RTEMP”	G(temp)	Tempr	T(Tempr)					
	“SINK”	G(sink)	Tsink	T(Tsink)					
	“GROW”	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		
	“HEAT”	CFILM	TSINK	CHEAT	TBODY	HCV	HNC	ITYPE	
		BNC	EMISS	HBL					
	“PATCH3D”	NPATCH							
		IDP	G1	G2	G3	G4			
		IDP	G1	G2	G3	G4			
		etc.	(npatch entries)						
	“BEZIER”	NP1	NP2	NSUB1	NSUB2				
		G1	G2	G3	G4	etc		(np1*np2 values)	
	“POLY”	NP1	NP2						
		G1	G2	G3	G4	etc		(np1*np2 values)	

	“CYLIND”	NSUB							
		Gtop	Rtop	Gbottom	Rbottom				
	“SPHERE”	NSUB							
		Gcenter	Radius						
	“LINE”	NPTS	Ix	Iy	Rx	Ry		(2D Contact)	
		G1	G2	G3	G4	etc.		(npts values)	
	“ARC”	NPTS	MethArc	Ix	Iy	Rx	Ry	(2D Contact)	
		G1	G2	G3	G4	etc.		(npts values)	
	“SPLINE”	NPTS	Ix	Iy	Rx	Ry		(2D Contact)	
		G1	G2	G3	G4	etc.		(npts values)	
	“NURBS2D”	NPTU	NORU	NSUB	Rx	Ry		(2D Contact)	
		G1 or X1	G2 or Y1	G3	G4 or X2	G5 or Y2	G6	[abs(nptu) grids or x, y, z, values]	See Remark 8
		Homo1	Homo2	Homo3	Homo4	etc.		(nptu values)	
		Knot1	Knot2	Knot3	Knot4	Knot5	etc.	(nptu+noru values)	
	“NURBS2”	IDN							
	“NURBS”	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	See Remark 8
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*nptv values]	See Remark 8	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	etc.			(nptu*nptv values)		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	etc.			(nptu+noru+nptv+norv values)		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc.	(NPTUtrim + NORUtrim entries)		

Examples (of Deformable and Rigid Contact):

Example 1 -- Typical deformable body

BCBODY	1		DEFOR M	101	0	.05			
--------	---	--	------------	-----	---	-----	--	--	--



## Example 2 -- Simple 4-node rigid patch (see Remark 5 for rigid bodies)

BCBODY	2		RIGID		0				
	PATCH3D	1							
		1	101	102	103	104			

Example 3-- Same as Example 2 except that a user subroutine named motion.f will be employed to specify the rigid body motion.

BCBODY	2		RIGID		0				
	ADVANCE			M					
	PATCH3D	1							
		1	101	102	103	104			

Describer	Meaning
BID (4,1)	Contact body identification number referenced by BCTABLE, BCHANGE, or BCMOVE. (Integer > 0; Required)
DIM	Dimension of body. (Character; Default = 3D) DIM=2D planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D any 3D body composed of rigid surfaces, shell elements or solid elements.
BEHAV (4,8)	Behavior of curve or surface (Character; Default = DEFORM) DEFORM body is deformable, RIGID body is rigid, SYMM body is a symmetry body, ACOUS indicates an acoustic body, WORK indicates body is a workpiece, HEAT indicates body is a heat-rigid body. <i>See Remark 3. for Rigid Bodies.</i>
BSID	Identification number of a BSURE, BCBOX, BCPROP or BCMATL entry if BEHAV=DEFORM. (Integer > 0)
ISTYP (4,3)	Check of contact conditions. (Integer $\geq$ 0; Default = 0) ISTYP is not supported in segment-to-segment contact. For a deformable body: =0 symmetric penetration, double sided contact. =1 unsymmetric penetration, single sided contact. (Integer > 0) =2 double-sided contact with automatic optimization of contact constraint equations (this option is known as “optimized contact”). Notes: single-sided contact (ISTYP=1) with the contact bodies arranged properly using the contact table frequently runs much faster than ISTYP=2. For a rigid body: =0 no symmetry condition on rigid body. =1 rigid body is a symmetry plane.

Describer	Meaning
FRIC (6,7)	Friction coefficient. (Real $\geq 0$ or integer; Default = 0) If the value is an integer it represents the ID of a TABL3Di.
IDSPL (4,5)	Set IDSPL=1 to activate the SPLINE (analytical contact) option for a deformable body and for a rigid contact surface. Set it to zero or leave blank to not have analytical contact. (Integer; Default = 0) <u>SOL 600 Options:</u> =0 or blank, SPLINE option is turned off. =1, The body is smoothed out with splines (2D) or Coons surfaces (3D). If SANGLE is not entered, the default 60.0 degrees is used. IDSPL=1 triggers the Marc SPLINE option with the 3rd field of the 3rd datablock set to 1. If analytical contact changes between increment zero and any subcase, the SANGLE Bulk Data entry is required (see the following SANGLE). SANGLE is placed in the 4th field of the 3rd Mac SPLINE datablock and a value of 1 is placed in the 3rd field of the 3rd datablock. >1, Identification number of a BLSEG entry that lists nodes on edges of the body which are excluded from the SPLINE option. (See Remark 10.) This option may be used for deformable or rigid contact surfaces. See PARAM,MSPLINC0 to enforce C0 continuity.
CONTROL (4,6)	Indicates the type of control for the body. Integer: = -1 for position control, = 0 for velocity control. = positive number for “load control” (the positive number is the grid number which has translational forces or SDCP’s are applied. The position of this grid is at the center of rotation given in the CGID field. For velocity controlled surfaces, the velocity of the body must be specified by the VELRBi fields. For displacement controlled surfaces the final displacement can be specified using SPCD at the grid ID specified by CONTROL. CONTROL > 0 is not available for enforced motion using SPCD’s for 2D contact using SOL 600.
NLOAD (4,7)	Enter a positive number if “load controlled” and rotations are allowed (Integer). The positive number is the grid number where the moments or rotations are applied. The position of this grid is at the center of rotation given in the CGID field. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof’s 1-3 (for rotation about x, y, z respectively), or starting with the MSC Nastran 2007 release by dof’s 4-6 (for rotation about x, y, z respectively). For versions prior to MSC Nastran 2007 only dof’s 1-3 could be used.
ANGVEL (6,1)	Angular velocity or angular position about local axis through center of rotation. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
DCOS1 (6,4)	3D - First component direction cosine of local axis if ANGVEL is nonzero. (Real)  2D - First coordinate of initial position of rotation of rigid body.

Describer	Meaning
DCOS2 (6,5)	3D - Second component direction cosine of local axis if ANGVEL is nonzero. (Real)  2D - Second coordinate of initial position of rotation of rigid body
DCOS3 (6,6)	3D - Third component direction cosine of local axis if ANGVEL is nonzero. (Real)  2D - Not used.
VELRB1 (5,4)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 1st direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
VELRB2 (5,5)	2D & 3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 2nd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.
VELRB3 (5,6)	3D - Velocity or final position (depending on the value of CONTROL) of rigid body in 3rd direction. (Real or Integer; Default = 0.0) If the value is an integer it represents the ID of a TABL3Di.  2D - Not used.
ADVANCE	The entries for this continuation line are for advanced options starting with MSC Nastran 2007.
SANGLE	Threshold for automatic discontinuity detection in degrees. (Real, Default = 60.0) SANGLE values on the BCBODY entry can be overridden using the SANGLE Bulk Data entry. This allows SANGLE to change for different subcases. It also allows analytical contact for this body to be "on" or "off" for different subcases (see the SANGLE Bulk Data entry). [Marc Spline option (3,4)]
COPTB	Flag to indicate how body surfaces may contact. See Remark 9. on the BCTABLE entry. (Integer; Default = 0)

Describer	Meaning
USER	<p>A series of character flags to indicate which (if any) user subroutines are required for this contact body. Enter as many characters as necessary to specify the user subroutines desired, for example to use MOTION and UFIRC enter MF. (Character; Default = blank meaning no user subroutines are required) (SOL 600 Only)</p> <p>M = The MOTION user subroutine is required to describe complex motion of a rigid contact body.</p> <p>F = The UFRIC or UFRICBBC user subroutine is required to describe complex friction behavior of the contact body.</p> <p>C = Film coefficient and sink temperature user subroutine UHTCON is required (coupled structural/heat analysis only)</p> <p>P = 4-Node patch user subroutine DIGEOM is required (the coordinates of the patch are entered in the user subroutine rather than on the BCBODY entry).</p> <p>S = The SEPSTR or SEPFOR user subroutine is required depending on whether stress-based or force-based friction has been specified using BCPARA IBSEP.</p>
“CTYPE”	<p>The entries of this continuation line define whether the original node-to-segment or segment-to-segment contact algorithm is used. Also the selection of general or small sliding contact and associated values when segment-to-segment contact is selected are entered here. These options may be placed on BCPARA instead of BCBODY (which is the recommended option. BCBODY entries override BCPARA entries). For MSC Nastran 2012, these items should be considered beta-test capabilities.</p>
ISMAIL [3,1]	<p>Enter 0 for general contact or 1 for small sliding, small displacement contact. See Remarks 15 and 16. (Integer; Default = 0)</p>
ICTYPE [3,2]	<p>Enter 0 for node-to-segment contact or 1 for segment-to-segment contact. See Remarks 15 and 16. (Integer; Default = 0)</p>
IAUG [3,3]	<p>This entry applies only if segment-to-segment contact is selected. See Remarks 15 and 16. (Integer; Default = 0)</p> <p>0 = No augmentation</p> <p>1 = Augmentation is based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements.</p> <p>2 = Augmentation is based on a constant Lagrange multiplier field.</p> <p>3 = Augmentation is based on a bilinear Lagrange multiplier field.</p>

Describer	Meaning
PENALT [5,1]	Augmented Lagrange penalty factor, used by segment-to-segment contact only. The default is derived from the contact tolerance distance and the stiffness of the deformable contact bodies or average stiffness if two deformable bodies are in contact. See Remark 18. (Real; Default = leave blank)
AUGDIST [5,2]	Penetration distance beyond which an augmentation will be applied, used by segment-to-segment contact only. this default is 1% of the contact tolerance distance. See Remark 18. (Real; Default = leave blank)
“RIGID”	The entries of this continuation line are for the rigid body description. See Remark 3.
CGID (5,i) i=1,2,3 (4,6)	Grid point identification number defining the initial position of the center of rotation for the rigid body or the point where a concentrated force or moment is applied.
NENT (4,2)	Number of geometric entities to describe this rigid surface. A rigid surface can be described by multiple sets of patches, nurbs, etc. For example, if it takes 3 sets of PATCH3D entries to describe a rigid surface, then set NENT=3. (Integer > 0; Default = 1)
Rigid Body Name (4,9)	Name associated with the rigid body. (Default is blank, 24-characters maximum)
“APPROV”	The entries for this continuation line are used for rigid contact body approach velocity and angular velocity. (For SOL 600, APPVEL may be used instead of APPROV if desired.)
A	Angular velocity about local axis through center of rotation. The local axis is defined by DCOS1, DCOS2, DCOS3. (Real; Default = 0.0)
Ni	Not used by SOL 600. The direction cosines for approach velocity in SOL 600 are DCOS1, DCOS2, DCOS3 on the second BCBODY line. (Real; Default = 0.0; Ni values entered on this line are ignored for SOL 600)
Vi	V1, V2 and V3 define the three components of approach velocity. For 2D contact only two should be entered as defined by Rx or Ry (see each 2D rigid contact body description for the meaning of Rx and Ry). (Real; Default = 0.0)
“RTEMP”	The entries of this continuation line describe rigid contact surface temperature variations.
G(Temp)	Grid point identification number defining the location of the temperature. (Integer; no Default)
Tempr	Temperature at the start of the run. (Real; no Default)
T(Temp)	Identification number of a TABL3Di describing the variation of the rigid body temperature during the analysis. (Integer; Default = 0 which means a constant value Ttempr is used during the entire analysis)

Describer	Meaning
<b>“SINK”</b>	The entries of this continuation line describe sink temperatures and their variation for flexible or heat transfer bodies.
G(sink)	Grid point identification number of defining the location of the sink temperature. (Integer; no Default)
Tsink	Sink temperature at the start of the run. (Real; no Default)
T(Tsink)	Identification number of a TABL3Di describing the variation of the sink temperature during the analysis. (Integer; Default = 0 which means a constant value Tsink is used during the entire analysis)
<b>“GROW”</b>	The entries for this continuation line are for rigid body growth - This line is rarely used. If entered PARAM,MRCONVER,11 must be used. If tables are used for growth, they should either be TABLED1 (growth vs time) or TABL3Di (growth vs one or more variables) and PARAM,MRCONTAB,11 must be entered.
GF1 (7,5)	Growth factor of rigid body in first coordinate direction. (Real; Default = 1.0)
GF2 (7,6)	Growth factor of rigid body in second coordinate direction. (Real; Default = 1.0)
GF3 (7,7)	Growth factor of rigid body in third coordinate direction. (Real; Default = 1.0)
TAB-GF1 (8,5)	Table for growth factor of rigid body in first coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
TAB-GF2 (8,6)	Table for growth factor of rigid body in second coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
TAB-GF3 (8,7)	Table for growth factor of rigid body in third coordinate direction. (Integer or blank; Default is blank which means no table and growth factor varies from 0.0 to 1.0 over the subcase being analyzed.)
<b>“HEAT”</b>	The entries of this continuation line(s) are for contact in heat transfer. Do not enter these line(s) for structural analyses.
CFILM (9,1)/(10,1)	Heat transfer coefficient (film) to environment. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem) If Real, the value entered is the film coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the heat transfer coefficient vs temperature. This is usually called HCVE in the Marc documentation.

Describer	Meaning
TSINK (9,2)/(10,2)	Environment sink temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the sink temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying temperature vs time. At present, this variable should not be a function of temperature.
CHEAT (9,3)/(10,3)	Contact heat transfer coefficient. (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the contact heat transfer coefficient vs temperature.
TBODY (9,4)/(10,4)	Body temperature. (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the body temperature. If Integer, the value entered is the ID of a TABLEM1 entry specifying the body temperature vs time. At present, this variable should not be a function of temperature.
HCV (9,5)/(10,5)	Convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field convection coefficient vs temperature.
HNC (9,6)/(10,6)	Natural convection coefficient for near field behavior (Real or Integer; Default = 1.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the near field natural convection coefficient vs temperature.
ITYPE [4,8]	An option entry for heat transfer only (Integer; no Default) 1 - Heat sink 4 - Heat conduction body
BNC (9,7)/(10,7)	Exponent associated with the natural convection coefficient for near field behavior (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is the exponent associated with the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the exponent associated with the near field natural convection coefficient vs temperature. At present, this variable should not be a function of temperature.
EMISS (9,8)/(10,8)	Emissivity for radiation to the environment or near thermal radiation (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 entry specifying the emissivity vs temperature.

Describer	Meaning
HBL (11,1)[12,1]	Separation distance dependent thermal convection coefficient (Real or Integer; Default = 0.0 for a heat transfer problem, omit for a structural problem). If Real, the value entered is a convection coefficient. If Integer, the value entered is the ID of a TABLEM1 entry specifying the convection coefficient vs temperature.
<b>“PATCH3D”</b>	Entries for this continuation line describe a rigid body made up of as many 4-node patches as desired. (Triangular patches are not available.)
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch (see Note 5).
<b>“BEZIER”</b>	Entries for this continuation line describe a rigid body made up of Bezier Surfaces.
NP1	Number of points in 1st direction. (Integer > 0)
NP2	Number of points in 2nd direction. (Integer > 0)
NSUB1	Number of subdivisions in 1st direction. (Integer > 0)
NSUB2	Number of subdivisions in 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
<b>“POLY”</b>	Entries for this continuation line describe a rigid body made up of Poly Surfaces.
NP1	Number of points in the 1st direction. (Integer > 0)
NP2	Number of points in the 2nd direction. (Integer > 0)
G1, G2, G3, etc	Grid numbers of each point (must be in order). There must be NP1*NP2 grid points defined. Enter NP1 points for NP2=1, then NP2 points for NP2=2, etc. (Integer)
<b>“CYLIND”</b>	Entries for this continuation line describe a cylindrical rigid body.
NSUB	Number of subdivisions. (Integer > 0)
Gtop	Grid point ID of a grid in the center of the top of the cylinder. (Integer > 0)
Rtop	Radius of the top of the cylinder. (Real > 0.0)
Gbottom	Grid point ID of a grid in the center of the bottom of the cylinder. (Integer > 0)
Rbottom	Radius of the bottom of the cylinder. (Real > 0.0)
<b>“SPHERE”</b>	Entries for this continuation line describe a spherical rigid body.
NSUB	Number of subdivisions. (Integer > 0)
Gcenter	Grid point ID of a grid in the center of the sphere. (Integer > 0)
Radius	Radius of the sphere. (Real > 0.0)
<b>“LINE”</b>	Entries for this continuation line describe a 2D rigid body made up of as many line segments as desired.



Describer	Meaning
NPTS	Number of points in the line segment.
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the line segment.
"ARC"	Entries for this continuation line describe a 2D rigid body made up of as many segments as desired describing an arc.
NPTS	Number of points in the arc. NPTS must be 4 for an ARC.
MethArc	Method to generate arc (Integer 0 to 4) (see Marc volume C Contact description (SOL 400 chapter 9) Figures 3-3 and 3-4).
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the 4 points as described by method.
"SPLINE"	Entries for this continuation line describe a 2D rigid body made up of as many spline segments as desired.
NPTS	Number of points for the spline.
Ix	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the first coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 1).
Iy	Grid direction (1, 2, or 3) of the G1, G2, ... list to be used as the second coordinate in Marc's 2D line description. See Remark 12. (Integer; Default = 2).
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc's 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc's 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTS points on the spline.

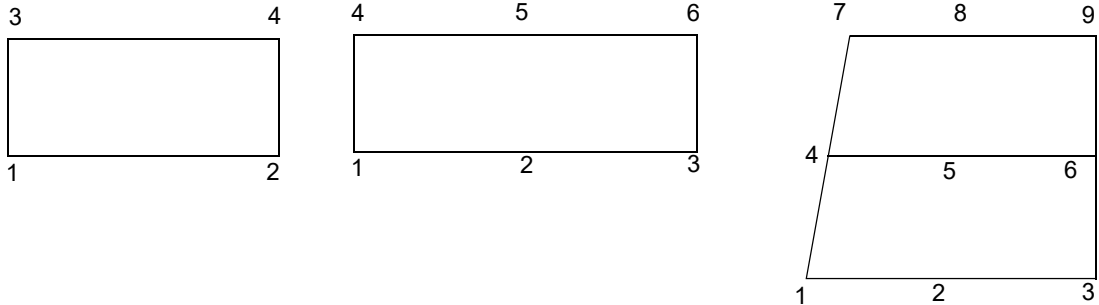
Describer	Meaning
“NURBS2D”	Entries for this continuation line describe a 2D rigid body made up of nurmbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default)
NORU	Order
NSUB	Number of subdivisions
Rx	Direction (1, 2, or 3) used as the first VELRBi in Marc’s 2D contact description. (Integer; Default = 1)
Ry	Direction (1, 2, or 3) used as the second VELRBi in Marc’s 2D contact description. (Integer; Default = 2)
G1, G2, G3, G4	Grid numbers for each of the NPTU control points
X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) \cdot \text{NPTV}$ (x,y,z) entries.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0) (Real). There must be NPTU entries.
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0) (Real). There must be (NPTU+NORU) entries.
“NURBS2”	Entries for this continuation line describe a rigid body made up of nurbs.
IDN	ID of a matching GMNURB entry. The GMNURB is an entry that contains the same information as at shown for the NURBS option. (Integer > 0)
“NURBS”	Entries for this continuation line describe a rigid body made up of nurbs.
NPTU	Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU may be set to a negative value whose absolute value is the number of xyz coordinates, but that is not required. (Integer; no Default)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction (Integer > 0; Required)
NTRIM	Number of trimming curves (Integer $\geq$ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0). There must be $\text{NPTU} \cdot \text{NPTV}$ entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be $\text{abs}(\text{NPTU}) \cdot \text{NPTV}$ (x,y,z) entries.

Describer	Meaning
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0). There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space. (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

#### Remarks:

1. Named continuation entries are ignored for a deformable curve or surface (BEHAV=DEFO), except for "HEAT".
2. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID.
3. WARNING: For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
4. (i,j) refers to data block i and field j of Marc's CONTACT model definition entry. IDSPL covers the SPLINE history definition in Marc. For structural analysis (i,j) refers to contact without tables. For heat transfer (i,j) refers to contact with tables.
5. For BEZIER surfaces, enter np1\*np2 points in the order shown:

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



6. For NURBS, enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for abs(NPTU) points and set NPTU to a negative value.
7. The heat transfer options are available for SOL 600 starting with MSC Nastran 2005 r2 and must use Marc 2005 or later.
8. For heat transfer items described using a TABLEM1 ID, the smallest value in the table will be entered into Marc's 9th contact (with tables) datablock. The table ID will be translated directly to Marc's 10th contact (with tables) datablock.
9. All flexible surfaces must have smaller BID values then the rigid surfaces so that in the Marc file all flexible surfaces are defined prior to all rigid surfaces. This is a Marc limitation.
10. When IDSPL is greater than 1, these nodes are entered in pairs. For a quad surface (for example, CQUAD4 or edge of a CHEXA) usually 4 sets of nodal pairs are needed to describe the surface. For example, a CQUAD4 with grid numbering 1,2,4,3 would need pairs of nodes, 1,2 2,4 4,3 3,1. The nodal pairs may be entered in any order. See Marc Volume C SPLINE (SOL 400 chapter 9) option documentation for more details.
11. With NURBS2D, the coordinates x1, x2, y1, y2 etc. or those given by G1, G2, etc. will be reversed (Y becomes X and X becomes Y) if any CQUADX, CTRIAX or CTRIAX6 entries are found in the bulk data. It is up to the user to ensure that the rigid contact surface is orientated in the correct direction after this reversal. If the user does not want the reversal to happen, enter PARAM,MARCREVX,-1 in the bulk data.
12. Ix and Iy for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIAX6 is defined in the Z-X plane, set Ix = 3 and Iy = 1 for all 2D rigid surfaces in the model. For CTRIAX and CQUADX, normally Ix=2 and Iy=1.
13. Rx and Ry for LINE, ARC, SPLINE and NURBS2D must be the same for all entries in the model. An example would be if CTRIAX6 is defined in the Z-X plane, set Rx = 3 and Ry = 1 for all 2D rigid surfaces in the model. For CTRIAX and CQUADX, normally Rx=2 and Ry=1.
14. For SOL 600, tables using integers for ANGVEL, VELRB1, VELRB2 and/or VELRB3 require PARAM,MRCONVER,11 and PARAM,MARCTOTT,1 to be set. the TABLE3D's used must define the entire motion/velocity desired.
15. If any CTYPE entries are used, PARAM,MRCONVER,11 and PARAM,MRCONVER,11 are required.

16. ISMALL, ITYPE, IAUG should be the same for all BCTABLE entries. The first entry “CTYPE” entry will be used for all bodies. PENALT and AUGDIST may vary from body to body.
17. 2D and 3D rigid bodies may not be mixed in the same input file.
18. The entries on the CTYPE header may optionally be specified on the BCPARA entry.

Correlation between BCBODY “CTYPE” entries and equivalent BCPARA entries:

BCPARA	BCPARA	CTYPE value(s)
METHOD	NODESURF	ictype=0, ismall=0
METHOD	NODSMALL*	ictype=0, ismall=1
METHOD	SEGSMALL	ictype=1, ismall=1
METHOD	SEGMENT*	ictype=1, ismall=1
METHOD	SEGLARGE*	ictype=1, ismall=0
AUTMENT	Integer Value	IAUG=Integer Value
PENALT	Real Value	PENALT=Real Value
AUGDIST	Real Value	AUTDIST=Real Value
* Not available in SOL 600.		

19. Most continuation lines may be omitted if not required except that the second line is required if the third or subsequent lines are to be entered.

## BCBODY1 Flexible or Rigid Contact Body in 2D and 3D (New Form)

Defines a flexible or rigid contact body in 2D or 3D used in SOLs 101, 400, and 700 only.

If the body is rigid, this entry may refer to BCRIGID entry by the BCRGID field to describe the body. See Remark 1. for an important note regarding how to define the outward direction of rigid bodies (which must point towards a flexible body for contact to occur).

Deformable bodies are described using as many standard elements as necessary and are specified by the BSID field with BEHAV=DEFORM. Unless shrink fit or interference fit is being analyzed, deformable bodies should not be inside other deformable bodies when the thickness of each body is taken into account.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCBODY1	BID	BPID	DIM	BEHAV	BSID	BCRGID	BCGOUT		

Format: (SOL 700 only)

1	2	3	4	5	6	7	8	9	10
BCBODY1	BID				BSID				

Examples of Deformable and Rigid Contact:

Example 1 -- Typical deformable body

BCBODY1	94	3001		DEFORM	103				
---------	----	------	--	--------	-----	--	--	--	--

Example 2 -- Simple rigid patch (see Remark 1. for rigid bodies)

BCBODY1	6	3001		RIGID	206	115			
---------	---	------	--	-------	-----	-----	--	--	--

Describer	Meaning
BID	Unique contact body identification number referenced by BCONECT, BCHANGE, or BCMOVE. (Integer > 0; Required)
BPID	Parameter identification number of a BCBDPRP entry. (Integer > 0 or blank) Ignored in SOL 700. See Remark 2.
DIM	Dimension of body. (Character; Default= 3D) Ignored in SOL 700. DIM=2D: planar body in x-y plane of the basic coordinate system, composed of 2D elements or curves. DIM=3D: any 3D body composed of rigid surfaces, shell elements or solid elements.
BEHAV	Behavior of curve or surface (Character; Default = DEFORM) Ignored in SOL 700. DEFORM body is deformable; RIGID body is rigid (See Remark 1.); SYMM body is a symmetry rigid body; HEAT indicates body is a heat-rigid body (See Remark 3..).

BSID	For SOLs 101 and 400: Identification number of a BSURF or BCPROP entry if BEHAV=DEFORM or HEAT, or identification number of a BCRGSRF, BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry if BEHAV=RIGID or SYMM (See Remark 4.).
	For SOL 700: Identification number (RBID) of a BSURF, BCBOX, BCPROP, BCMATL, BCSEG, BCGRID or BCELIPS entry. (Integer > 0).
BCRGID	For SOLs 101 and 400: Identification number of a BCRIGID entry if BEHAV=RIGID or SYMM. Ignored in SOL 700. (Integer > 0)
BCGOUT	Reference point in basic coordinate system to calculate the global resultant contact force/moment (integer)
	-1      Origin
	0        Estimated centroid of deformable body (default)

#### Remarks:

1. **WARNING:** For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e., in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches
2. If BPID field is blank, then default values are set for the parameters defined in BCBDPRP entry.
3. For pure thermal analysis, BEHAV=DEFORM and BEHAV=HEAT are identical. However, in coupled thermal-mechanical analysis, BEHAV=DEFORM specifies a deformable in mechanical analysis part, while BEHAV=HEAT defines a rigid meshed body in mechanical analysis part.
4. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2, or BCNURBS entry directly (RBID).

## BCBOX

## 3D Contact Region

Defines a 3D contact region - all elements within the region define a contact body in SOL 600 and 700.

Format (Form 1):

1	2	3	4	5	6	7	8	9	10
BCBOX	ID		HOW						
	N1	N2	N3	N4	N5	N6	N7	N8	

Form 2:

BCBOX	ID	COORD	HOW						
			X1	Y1	Z1	X2	Y2	Z2	
			X3	Y3	Z3	X4	Y4	Z4	
			X5	Y5	Z5	X6	Y6	Z6	
			X7	Y7	Z7	X8	Y8	Z8	

Example (for Form 1):

BCBOX	101		0						
	1001	1002	1003	1004	1005	1006	1007	1008	

Describer	Meaning	
-----------	---------	--

**ID** Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control BCONTACT=BCBOX is specified. All elements corresponding to the designated box may potentially come into contact. See Remark 1. (Integer > 0)

**COORD** Enter COORD in field 3 if x,y,z coordinates of the box are to be specified rather than grid IDs. (Character)

**HOW** A flag indicating whether an element is in the defined box or not. (Integer; Default = 0)

0 If only one grid point of an element is in the box, the entire element is considered to be in the box.

1 All grid points comprising the element must be within the box, otherwise the element is considered outside of the box.

**N1-N8** Enter 8 Grid IDs defining a box (hexa-like region) if the third field is blank. (Integer; Required if COORD is blank)

**Xi, Yi, Zi** Enter eight x,y,z values in the basic coordinate system if the third field is COORD. See Remark 6. (Real; Required if "COORD" is entered in field 3 of line 1)



**Remarks:**

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCPROP, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. The model is searched to determine whether each element lies within the specified box region as specified by the HOW criteria option.
6. For SOL 700, the BCBOX allows a general box and edges do not need to be aligned with the coordinates axis. The eight grid points define the box identical to the CHEXA grid point numbering.
7. BCBOX is not supported in SOLs 101 and 400.

**BCBZIER**

Defines a Rigid Contact Body Made up of Bezier Surfaces in SOLs 101 and 400

Defines a rigid contact body made up of Bezier surfaces used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCBZIER	RBID	NP1	NP2	NSUB1	NSUB2				
	G1	G2	G3	G4	-etc.-				

Example:

BCBZIER	98	3	3	25	15				
	21	11	1	84	74	64	147	137	
	127								

Descriptor    Meaning

RBID            Unique identification number referenced by a BCRGSRF or BCBODY1 entry. (Integer > 0) See Remark 1.

NP1            Number of points in 1st direction. (Integer > 0)

NP2            Number of points in 2nd direction. (Integer > 0)

NSUB1        Number of subdivisions in 1st direction. (Integer > 0)

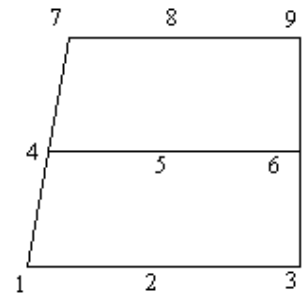
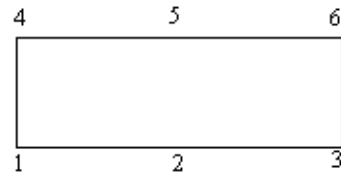
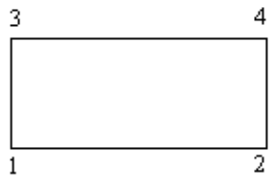
NSUB2        Number of subdivisions in 2nd direction. (Integer > 0)

G1, G2, G3, etc.    Grid numbers of each point (must be in order). There must be NP1\*NP2 grid points defined. Enter NP1 points for NP2=1, then NP1 points for NP2=2, etc. (Integer > 0)

Remarks:

1. If BCRGSRF entry does not exist, BCBZIER entry will be referenced by the **BCBODY1** entry directly.
2. For BEZIER surfaces, enter np1\*np2 points in the order shown below

Mesh	Normal Order	Reversed Order
2x2	1,2,3,4	2,1,4,3
3x2	1,2,3,4,5,6	3,2,1,6,5,4
3x3	1,2,3,4,5,6,7,8,9	3,2,1,6,5,4,9,8,7



## BCELIPS

### Contact Ellipsoid List- SOL 700

Defines a list of ellipsoid names (character strings) for use of contact analysis. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCELIPS	CID	NAME1	NAME2	NAME3	NAME4	NAMEi	etc		

Example:

BCELIPS	1	HUB	RIM	HEAD	CHEST				
---------	---	-----	-----	------	-------	--	--	--	--

Field	Content
CID	Unique identification number of ellipsoid names which can be used for contact. (Integer > 0; required)
NAME	Name of an ellipsoid. (Character; required)

Remarks:

Use as many continuations as required to define the complete list of names. A blank field terminates the list..

## BCGRID (SOL 700 only)

## Contact Node Region

Grids to be included in contact analyses in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCGRID	CID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
	GID8	GID9	GID10	GID11	GID12	GID13	-etc.-		

Example:

BCGRID	100	12	14	17	121	234	235	270	
	309	1001	THRU	2000	BY	2			

Describer	Meaning
ID	Unique identification number of a “cloud” of grid points which can be used for contact or RCONN, BJOIN or WALL. (Integer > 0; Required)
GID1, GID2,...	Gridpoint ID. THRU indicates a range to be used. BY is the increment to be used within this range. (Integer > 0; Required)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, BCMATL, or BCSEG entries.
2. BCGRID may only be used for SECNDRY body definitions on the BCTABLE entry.
3. As many continuation lines as necessary may be used to define all GRID points used in the definition.

## BCGRID

## Contact region by a list of grid points.

Define a contact region by a grid point list, for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCGRID	BID	BPID	DIM						
	GID1	GID2	GID3	"THRU"	GID4	"BY"	GID5	GID6	
	GID7								

Example:

BCGRID	2	3	3D						
	12	21	THRU	101	3	6			

Describer	Meaning
BID	Unique contact face identification number referenced by BCONECT (Integer > 0; Required). See remark 1.
BPID	Parameter identification number of a BCBDPRP entry. (Integer > 0 or blank)
DIM	Dimension of the contact entry. (Character; Default= "3D"), "3D" or "2D".
GIDi	Grid point ids. (Integer > 0)

Remarks:

1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.
2. BCGRID entries are able to coexist with BCBODY1 entries. A BCGRID entry can be used with a BCSURF or a BCBODY1 entry to construct a contact pair. The BCGRID entry must be a secondary, the BCSURF or BCBODY must be a primary.
3. This entry only works in node to segment method.
4. The identification of BCGRID cannot be specified in the continuations of a BCONECT entry.
5. If a BCRIGID is referenced by a BCONECT, COPTS and COPTM in the BCONPRG referenced by the BCONECT will be ignored.

## BCHANGE

## Changes Definitions of Contact Bodies

Changes definitions of contact bodies used in SOL 101 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCHANGE	ID	TYPE			IDBOD1	N1	N2	INC	
	IDBOD2	N1	N2	INC	IDBOD3	etc.			

Example:

BCHANGE	201	NODE			1	1001	1010	1	
	2	2001	2021	2					

Describer	Meaning	
ID	Identification number referenced by a BCONTACT or BCHANGE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer $\geq 0$ ; Required)	
Type	Type of modification (Character; Required)	
	NODE	Defines nodes of a contact body which may come into contact.
	EXCLUDE	Excludes 2 node segments in 2D or 4 node patches in 3D.
IDBODi	Identification number of a contact body, BCBODY or BCBODY1 entry. (Integer $> 0$ )	
N1	Starting grid ID. More than one N1-N2-INC range may be entered for each body, see Remark 7. (Integer $> 0$ ; Default = 1)	
N2	Ending grid ID.	
INC	Grid ID increment.	

Remarks:

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCHANGE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCHANGE=ID. Note that if BCHANGE Case Control command exists, it always dominates the selection of BCHANGE Bulk Data entries.
2. The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
3. BCHANGE is useful only for saving computer time and is not recommended for general usage.
4. Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.
5. BCHANGE with the NODE option is not supported in Segment-to-segment contact.

6. NODE and EXCLUDE may not be used simultaneously in the same BCHANGE entry.
7. If TYPE=NODE, the form of N1-N2-INC range has the following rules:
  - a. The format of INC is either blank or integer ( $\geq 0$ ).
  - b. N1-N2-0 or N1-N2-blank represents 2 nodes (N1,N2) where  $0 < N1$  and  $0 < N2$ ,
  - c. N1-N2-INC,  $INC > 0$ , represent a range input but  $0 < N1 < N2$ .
8. If TYPE=EXCLUDE, the form of N1-N2-INC range has the following rules:
  - a. For node segment, N1-N2-INC is used. If  $0 < N1 < N2$  and  $INC > 0$ , the range is applied. If INC-blank or 0, it is 2 nodes (N1,N2) input that can be in any order.
  - b. For 4 nodes patch, (N1,N2,N3,N4), 2 sets of range, which have to input in sequence, are required.  
(IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N4,INC2) where  $IDBOD1 = IDBOD2$ ,  $N_i > 0$ , INC1 and INC2 are ignored.

The following example is for a 4-node patch (100,110,300,200)

1	2	3	4	5	6	7	8	9	10
BCHANGE	1	EXCLUDE			1	-100	110		
	1	300	200						

- c. For 3 nodes patch, (N1,N2,N3), 2 sets of range, which have to input in sequence, are required (IDBOD1,-N1,N2,INC1) and (IDBOD2,N3,N3,INC2) where  $IDBOD1 = IDBOD2$ ,  $N_i > 0$ , INC1 and INC2 are ignored.

The following example is for a 3-noded patch (132,97,95)

1	2	3	4	5	6	7	8	9	10
BCHANGE	1	EXCLUDE			1	-132	97		
	1	95	95						



## BCHANGE

## Changes Definitions of Contact Bodies in SOL 600

Changes definitions of contact bodies used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCHANGE	ID	TYPE	NBOD		IDBOD1	N1	N2	INC	
	IDBOD2	N1	N2	INC	IDBOD3	etc.			

Example:

BCHANGE	201	NODE	2		1	1001	1010	1	
	2	2001	2021	2					

Descriptor	Meaning
ID	Identification number referenced by a SUBCASE Case Control command. See Remark 2. (Integer $\geq 0$ ; Required)
Type	Type of modification (Character; Required) NODE Defines nodes of a contact body which may come into contact. EXCLUDE Excludes 2 node segments in 2D or 4 node patches in 3D.
NBOD	Number of bodies to be modified -- must match number of bodies actually entered. More than one N1-N2-INC range may be entered for each body, see Remark 7. See N1 below. (Integer $> 0$ ; Default = 1)
IDBODi	Identification number of a contact body, BCBODY entry. (Integer $> 0$ )
N1	Starting grid ID.
N2	Ending grid ID.
INC	Grid ID increment.

Remarks:

- To place an entry in Marc's phase 0, set ID=0. To activate the entry for the first SUBCASE, SET ID=1, for the 2nd, set ID=2.
- The BCHANGE entry does not apply to rigid bodies. Multiple BCHANGE entries are allowed. A body may be entered more than once with different grid IDs.
- The BCHANGE entry covers Marc's history definitions CONTACT NODE and EXCLUDE.
- BCHANGE is useful only for saving computer time and is not recommended for general usage.
- Warning -- For the NODE option, if some nodes in a body are inadvertently omitted, they may penetrate other bodies.

6. If more than one N1-N2-INC range is required for a body, enter N1 as a negative value for all ranges except for the last range for which N1 is entered as a positive value.
7. If more than one N1-N2-INC range is required, all ranges with IBOD1 must come first, followed by all with IBOD2, etc.
8. The EXCLUDE option is obsolete. The reasons why it was added in the past have been alleviated by better improved contact algorithms. If the EXCLUDE option is entered for 3D shapes two N1-N2-INC ranges are normally required to define all 4 nodes of a patch. If the patch is triangular, the last two nodes must be repeated. The following are examples of how data are entered for one element using the exclude option. The first example is for a 4-node patch (nodes 100, 110, 200, 300) and the second is for a 3-node patch (nodes 132, 97, 95, 95).

BCHANGE	1	EXCLUDE	1		1	-100	110	10	
	1	200	300	100					

BCHANGE	1	EXCLUDE	1		1	-132	132	0	
	1	-97	97	0	1	-95	95	0	
	1	95	95	0					

## BCMATL

## 3D Contact Region by Element Materials

Defines a 3D contact region by element material. All elements with the specified materials define a contact body used in SOL 600 and SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
BCMATL	ID	IM1	IM2	IM3	IM4	IM5	IM6	IM7	
	IM8	IM9	etc.						

Example:

BCMATL	1001	101	201	301					
--------	------	-----	-----	-----	--	--	--	--	--

Describer	Meaning
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry if the Case Control command, BCONTACT=BCMATL is specified. All elements corresponding to the material IDs specified may potentially come into contact. See Remark 1. (Integer > 0)
IMi	Material ID. A minimum of one entry is required. (Integer)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCPROP entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all material IDs associated with a particular deformable body.
6. BCMATL may not be used to define contact regions made up of composite elements.

**BCMOVE****Movement of Bodies in Contact**

Defines movement of bodies in contact used in SOL 101 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
BCMOVE	ID	MTYPE							
	IDRBOD1	IDRBOD2	IDRBOD3	etc.					

Examples:

BCMOVE	33	RELEASE							
	1	3	5	7					
BCMOVE	1	approach							

Describer	Meaning
ID	Identification number referenced by a BCONTACT or BCMOVE Case Control command within a SUBCASE or STEP. See Remark 1. (Integer $\geq 0$ ; Required)
MTYPE	Movement type. (Character; Default = APPROACH)
	APPROACH      All rigid bodies are moved so that they all make contact with deformable bodies.
	RELEASE        The contact condition is released for selected bodies.
	SYNCHRON      All rigid bodies are moved until the first rigid body makes contact with a deformable body.
IDRBODi	Identification numbers of contact bodies to be released, for option RELEASE only. Points to BCBODY or BCBODY1 Bulk Data entries.

Remarks:

1. To place an entry in the loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or BCMOVE=0, and it is always executed automatically. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID or BCMOVE=ID. Note that if BCMOVE Case Control command exists, it always dominates the selection of BCMOVE Bulk Data entries. ID must be unique (only one BCMOVE per SUBCASE).
2. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
3. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
4. You may release a deformable body from contact with either a deformable or rigid body.

## BCMOVE

### Movement of Bodies in Contact in SOL 600

Defines movement of bodies in contact to be used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCMOVE	ID	MTYPE	IREL						
	IDRBOD1	IDRBOD2	IDRBOD3	etc.					

Examples:

BCMOVE	33	RELEASE	20						
	1	3	5	7					
BCMOVE	1	approach							

Describer	Meaning
ID	Identification number referenced by a SUBCASE or STEP Case Control command. See Remark 1. (Integer $\geq 0$ ; Required)
MTYPE	Movement type. (Character; Default = APPROACH) APPROACH      All rigid bodies are moved so that they all make contact with deformable bodies. RELEASE        The contact condition is released for selected bodies. SYNCHRON      All rigid bodies are moved until the first rigid body makes contact with a deformable body.
IREL	Flag to indicate how contact forces are removed, for option RELEASE only. (Integer) = 0              Contact forces are immediately removed. (Default) > 0              Contact forces are reduced to zero over the number of increments specified in this load period. See NLPARM and TSTEP1 for the number of increments.
IDRBODi	Identification numbers of rigid bodies to be released, for option RELEASE only. Points to BCBODY Bulk Data entries.

Remarks:

1. To place an entry in Marc's phase 0, set ID=0. To activate the entry for the 1st SUBCASE, SET ID=1, for the 2nd, set ID=2. ID must be unique (only one BCMOVE per SUBCASE).
2. This entry matches Marc's history definitions RELEASE, APPROACH, and SYNCHRONIZED. Note that Marc's history definition MOTION CHANGE is done in MSC Nastran by describing the enforced motion for the grid which is defined to be the center of rotation of the rigid body, see CGID of the BCBODY entry.

3. For MTYPE=APPROACH and MTYPE=SYNCHRON leave all following fields blank.
4. The APPROACH and SYNCHRON options apply to rigid contact surfaces only.
5. You may release a deformable body from contact with either a deformable or rigid body.

**BCNURB2**

Defines a 2D Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a 2D rigid contact body made up of NURBS used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCNURB2	RBID	NPTU	NORU	NSUB					
	“GRID” or “COORD”	G1 or X1	G2 or Y1	G3 or X2	G4 or Y2	-etc.-			
	“HOMO”	Homo1	Homo2	Homo3	Homo4	-etc.-			
	“KNOT”	Knot1	Knot2	Knot3	Knot4	Knot5	-etc.-		

Examples:

BCNURB2	102	4	1	1					
	GRID	237	101	104	235				
	HOMO	0.0	0.5	0.5	1.0				
	KNOT	0.0	0.2	0.4	0.8	1.0			

BCNURB2	3001	-5	3	50					
	COORD	-.1	.14	-.1	.04	-1.4-16	.04		
		.1	.04	.1	.14				
	HOMO	1.	.707107	1.	.707107	1.			
	KNOT	0.0	0.0	0.0	0.5	0.5	1.0	1.0	
		1.0							

Describer

Meaning

RBID

Unique identification number referenced by a BCRGSRF or BCBODY1 entry. (Integer &gt; 0) See Remark 1.

NPTU

Number of control points. If the control points are entered as coordinates rather than grid IDs NPTU must be set to a negative value whose absolute value is the number of xy coordinates. (Integer; no Default)

NORU

Order along U direction. (Integer &gt; 0)

NSUB

Number of subdivisions. (Integer &gt; 0)

“GRID”

Indicate the start of the list of grid numbers. See Remark 2.

G1, G2, G3, G4,  
etc.

Grid numbers for each of the NPTU control points. (Integer &gt; 0)

“COORD”

Indicate the start of the list of xy coordinates. See Remark 2.

X1, Y1, X2, Y2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU) set of (x,y) entries. (Real)
“HOMO”	Indicate the start of the list of homogeneous coordinates.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates. (Real; 0.0 to 1.0) There must be NPTU entries.
“KNOT”	Indicate the start of the list of knot vectors.
Knot1, Knot2, Knot3, etc.	Knot vectors. (Real; 0.0 to 1.0) There must be (NPTU+NORU) entries.

#### Remarks:

1. If BCRGSRF entry does not exist, BCNURB2 entry will be referenced by the [BCBODY1](#) entry directly.
2. The list of grid IDs (“GRID”) and the list of xy coordinates (“COORD”) cannot coexist.



## BCNURBS Defines a Rigid Contact Body Made up of NURBS in SOLs 101 and 400

Defines a rigid contact body made up of NURBS used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCNURBS	RBID	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV		
	“GRID”	G1	G2	G3	G4	G5	G6		
	or	or	or	or	or	or	or	-etc.-	
	“COORD”	X1	Y1	Z1	X2	Y2	Z2		
	“HOMO”	Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	-etc.-					
	“KNOT”	Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	-etc.-					
	“TRIM”	IDtrim1	IDtrim2	IDtrim3	-etc.-				

Examples:

BCNURBS	48	2	1	2	1	50	50		
	GRID	3005	102						
	HOMO	1.	.3333						
	KNOT	0.0	0.0	0.5	0.5	1.	1.		

BCNURBS	63	-2	2	2	2	50	50		
	COORD	.3	-.6	0.	.3	-.1	0.		
	HOMO	1.	1.	1.	1.				
	KNOT	0.0	0.0	1.	1.	0.	0.		
		1.	1.						
	TRIM	511	2002	87	704				

Describer	Meaning
RBID	Unique identification number referenced by a BCRGSRF or BCBODY1 entry. (Integer > 0) See Remark 1.
NPTU	Number of control points in U direction. If the control points are entered as coordinates rather than grid IDs, NPTU must be set to a negative value whose absolute value is the number of xyz coordinates. See Remark 2. (Integer > 0; Required)
NPTV	Number of control points in V directions. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)

NORV	Order along V direction. (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction. (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction. (Integer > 0; Required)
“GRID”	Indicate the start of the list of grid numbers. See Remark 3.
G1, G2, G3, etc.	Grid point IDs defining control points. (Integer > 0) There must be $NPTU \cdot NPTV$ entries.
“COORD”	Indicate the start of the list of xyz coordinates. See Remark 3.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be $abs(NPTU) \cdot NPTV$ sets of (x,y,z) entries.
“HOMO”	Indicate the start of the list of homogeneous coordinates.
Homo1, Homo2, Homo3, etc.	Homogeneous coordinates (0.0 to 1.0). There must be $abs(NPTU) \cdot NPTV$ entries. (Real)
“KNOT”	Indicate the start of the list of knot vectors.
Knot1, Knot2, Knot3, etc.	Knot vectors (0.0 to 1.0). There must be $(abs(NPTU) + NORU) + (NPTV + NORV)$ entries. (Real)
“TRIM”	Indicate the start of the list of trimming vector IDs.
IDtrimi	Trimming vector identification number of a BCTRIM entry. (Integer > 0)
Homot1, Homot2, Homot3, etc.	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be $NPTU_{trim}$ entries. (Real)
Knott1, Knott2, Knott3, etc.	Knot vectors (0.0 to 1.0) of this trimming vector. There must be $NPTU_{trim} + NORU_{trim}$ entries. (Real)

#### Remarks:

1. If BCRGSRF entry does not exist, BCNURBS entry will be referenced by the BCBODY1 entry directly.
2. Enter NPTU grid points G1, G2, G3, etc. (set NPTU to a positive value equal to the number of grid points) or enter X1, Y1, Z1, X2, Y2, Z2, etc. coordinates for  $abs(NPTU)$  points and set NPTU to a negative value.
3. The list of grid IDs (“GRID”) and the list of xyz coordinates (“COORD”) cannot coexist.

## BCONECT Defines a Contact Pair

Defines a contact pair used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types](#) of *Nastran SOL 400 Getting Started Guide*). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103-112 and standard 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types.

Format:

1	2	3	4	5	6	7	8	9	10
BCONECT	ID	BCGPID	BCPPID	IDSCND	IDPRIM				
	“SECNDRY”	IDSEC1	IDSEC2	IDSEC3	IDSEC4	IDSEC5	IDSEC6	IDSEC7	
		IDSEC8	IDSEC9	-etc.-					
	“PRIMARY”	IDPRIM1	IDPRIM2	IDPRIM3	IDPRIM4	IDPRIM5	IDPRIM6	IDPRIM7	
		IDPRIM8	IDPRIM9	-etc.-					

Examples:

BCONECT	57	306		2	1002				
BCONECT	9		108						
	SECNDRY	30	26						
	PRIMARY	294	135	528					

Describer	Meaning
ID	Unique identification number referenced by a BCTABL1 entry (Integer ≥ 0). See Remark <a href="#">1</a> .
BCGPID	Parameter identification number of a BCONPRG entry (Integer ≥ 0 or blank). See Remark <a href="#">2</a> .
BCPPID	Parameter identification number of a BCONPRP entry (Integer ≥ 0 or blank). See Remark <a href="#">2</a> .
IDSCND	Identification number of BCBODY1, BCSURF and BCGRID entry defining the touching body (Integer ≥ 0 or blank). See Remarks <a href="#">3</a> . and <a href="#">4</a> .
IDPRIM	Identification number of BCBODY1 and BCSURF entry defining the touched body (Integer ≥ 0 or blank). See Remarks <a href="#">5</a> . and <a href="#">6</a> .
“SECNDRY”	Indicates the start of the list of the touching bodies. See Remark <a href="#">4</a> .

IDSECI	Identification number of BCBODY1 entry defining the touching bodies (Integer $\geq 0$ or blank).  For SOL 700, leaving IDSEC1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.
“PRIMARY”	Indicates the start of the list of bodies touched by touching bodies. See Remark 6.
IDPRIMI	Identification number of BCBODY1 entry defining touched bodies (Integer $\geq 0$ or blank).

#### Remarks:

1. BCONNECT can be selected by the Case Control command BCONTACT=ID to define surface contact if BCTABL1 entry does not exist. See Remarks 2. and 3. of BCTABL1 entry.
2. If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
3. A short input to define a single touching body exists if the user provides IDSCND. On the other hand, if the user leaves IDSCND blank, then “SECNDRY” descriptor is required and IDSEC1 must be specified. Exceptions are for SOL 700 self-contact, which may use a secondary IDSEC1 of zero and no “PRIMARY” entry.
4. “SECNDRY” and IDSECI fields will be ignored if IDSCND exists. If IDSCND field is blank, then “SECNDRY” and IDSECI must be specified. In this case, each IDSECI will be processed separately.
5. A short input to define a single touched body exists if the user provides IDPRIM. On the other hand, if the user leaves IDPRIM blank, then “PRIMARY” descriptor is required and IDPRIM1 must be specified.
6. “PRIMARY” and IDPRIMI fields will be ignored if IDPRIM exists. If IDPRIM field is blank, then “PRIMARY” and IDPRIMI must be specified.
7. The concept of Secondary and Primary relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
8. If all the BCONPRG that are referenced by a BCONNECT (which is referenced by BCTABL1) have a value of IGLUE  $> 0$ , and this BCONNECT is referenced in the 1st Loadcase SOL 101 to 112 and SOL 200 or the 1st Step (SOL 400) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

## BCONP Contact Parameters

Defines the parameters for a contact region and its properties for slideline contact in SOL 106 or SOL 129. SOL 400 is the preferred method for contact analysis, see BCBODY1, BCONNECT, BCONPRG, BCONPRP, BCPARA and BCTABL1.

Format:

1	2	3	4	5	6	7	8	9	10
BCONP	ID	SECNDRY	PRIMARY		SFAC	FRICID	PTYPE	CID	

Example:

BCONP	95	10	15		1.0	33	1		
-------	----	----	----	--	-----	----	---	--	--

Descriptor	Meaning
ID	Contact region identification number. See Remark 1. (Integer > 0)
SECNDRY	Secondary region identification number. See Remark 2. (Integer > 0)
PRIMARY	Primary region identification number. See Remark 3. (Integer > 0)
SFAC	Stiffness scaling factor. SFAC is used to scale the penalty values automatically calculated by the program. See Remark 4. (Real > 0.0; Default = 1.0)
FRICID	Contact friction identification number. See Remark 5. (Integer > 0 or blank)
PTYPE	Penetration type. See Remark 6. (Integer 1 or 2; Default = 1) 1            Unsymmetrical (secondary penetration only--Default) 2            Symmetrical
CID	Coordinate system identification number to define the slideline plane vector and the slideline plane of contact. See Remark 7. (Integer > 0; Default = 0, which means the basic coordinate system)

Remarks:

1. ID field must be unique with respect to all other BCONP identification numbers.
2. The referenced SECNDRY is the identification number in the BLSEG Bulk Data entry. This is the secondary line. The width of each secondary segment must also be defined to get proper contact stresses. See the Bulk Data entry, [BWIDTH](#) for the details of specifying widths.
3. The referenced PRIMARY is the identification number in the BLSEG Bulk Data entry. This is the primary line. For symmetrical penetration, the width of each primary segment must also be defined. See the Bulk Data entry, [BWIDTH](#) for the details of specifying widths.

4. SFAC may be used to scale the penalty values automatically calculated by the program. The program calculates the penalty value as a function of the diagonal stiffness matrix coefficients that are in the contact region. In addition to SFAC, penalty values calculated by the program may be further scaled by the ADPCON parameter (see description of ADPCON parameter for more details). The penalty value is then equal to  $k \cdot \text{SFAC} \cdot |\text{ADPCON}|$ , where  $k$  is a function of the local stiffness. It should be noted that the value in SFAC applies to only one contact region, whereas the ADPCON parameter applies to all the contact regions in the model.
5. The referenced FRICID is the identification number of the BFRIC Bulk Data entry. The BFRLC defines the frictional properties for the contact region.
6. In an unsymmetrical contact algorithm only secondary nodes are checked for penetration into primary segments. This may result in primary nodes penetrating the secondary line. However, the error depends only on the mesh discretization. In symmetric penetration both secondary and primary nodes are checked for penetration. Thus, no distinction is made between secondary and primary. Symmetric penetration may be up to thirty percent more expensive than the unsymmetric penetration.
7. In Figure 9-2, the unit vector in the Z-axis of the coordinate system defines the slideline plane vector. The slideline plane vector is normal to the slideline plane. Relative motions outside the slideline plane are ignored, and therefore must be small compared to a typical primary segment. For a primary segment the direction from primary node 1 to primary node 2 gives the tangential direction ( $t$ ). The normal direction for a primary segment is obtained by the cross product of the slideline plane vector with the unit tangent vector (i.e.,  $n = z \times t$ ). The definition of the coordinate system should be such that the normal direction must point toward the secondary region. For symmetric, penetration, the normals of primary segments and secondary segments must face each other. This is generally accomplished by traversing from primary line to secondary line in a counterclockwise or clockwise fashion depending on whether the slideline plane vector forms a right-hand or left-hand coordinate system with the slideline plane.

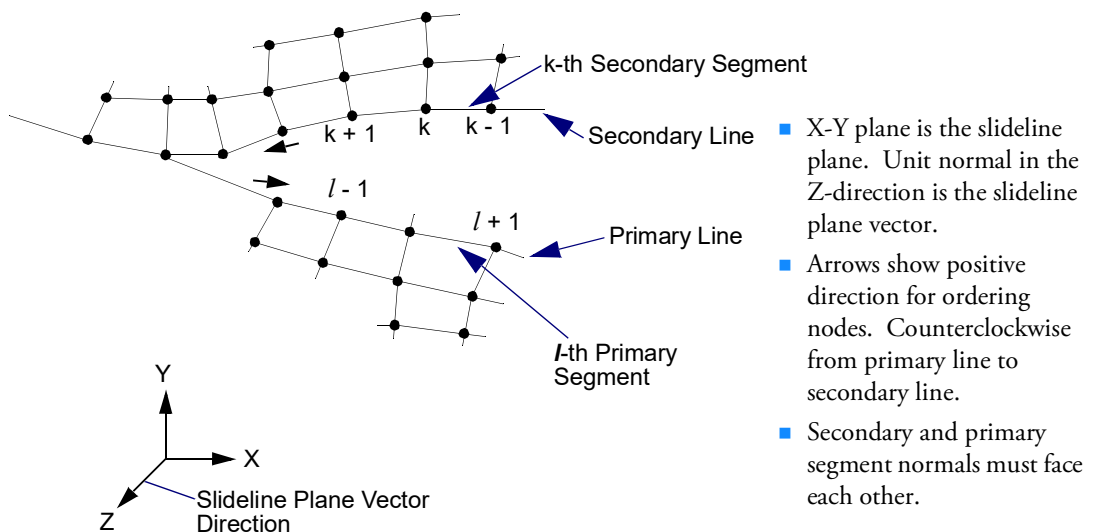


Figure 9-2 A Typical Finite Element Slideline Contact Region

## BCONPRG      Geometric Contact Parameters of Touching Bodies

Defines geometric contact parameters used to determine if contact occurs between bodies. For segment-to-segment contact it also defines additional numerical parameters used to apply the constraints. The parameters defined here are referenced by the BCONNECT entry. This entry is used in conjunction with the BCONPRP entry.

Geometric Contact Parameters of Touching Bodies in SOLs 101 and 400 for General Contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the Permanently Glued or Tied Contact.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRG	BCGPID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRG	90		ICOORD	1	IGLUE	1			
---------	----	--	--------	---	-------	---	--	--	--

Describer	Meaning
BCGPID	Identification number for geometric contact parameters (Integer > 0).
PARAMi	Name of a parameter. Allowable names are given in <a href="#">Table 9-4</a> (Character).
VALi	Value of a parameter. See <a href="#">Table 9-4</a> (Real or Integer).

Table 9-4 Geometric Contact Parameters of Touching Bodies in SOLs 101, 103, 105, 107-112, 200 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real $\geq 0.0$ , see Remark 6. for default)
BIAS	Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0 $\leq$ Real $\leq$ 1.0)
CINTERF	The definition varies depending on OPINTRF. (Real; Default = 0.0) When OPINTRF is set to 0 (default), Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies. For CINTERF < 0.0, gap between bodies. When OPINTRF is set to 1, interference closure magnitude. To clear penetration, set a negative value. When OPINTRF is set to 2, magnitude of translation vector defining in VXINTRF, VYINTRF and VZINTRF. When OPINTRF is set to 3, not used. When OPINTRF is set to 4, not used.
COPTS, COPTM	Flag to indicate how secondary and primary surfaces may contact. See Remark 2. (Integer; Default = 1011). COPTS and COPTM are ignored if secondary or primary in BCONECT is a BCGRID entry but honored if it is a BCSURF entry.
ERROR	Distance below which a node is considered touching a body. Default = blank, automatic calculation. See BCPARA Bulk Data entry for more details. (Real)
HARDS	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, i.e. ISTYP=2 on the BCBDPRP entry. The hard-soft ratio can be used by the program if there is a significant difference in the average stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger than the hard-soft ratio, the nodes of the softest body are the preferred secondary nodes. (Real; Default = 2.0) This parameter is ignored for the permanently glued contact.
FGCFLG	Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.) 0 – no Cohesive (Flexible) Glued Contact 1 – activate Cohesive (Flexible) Glued Contact
FGCNST	Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCTST	Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCNSTR	ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCTSTR	ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.



FGCNSTI	ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCTSTI	ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGRCEN	Reference Grid at which resultant contact force of secondary body to primary body is evaluated. See Remark 12.
FGRCN1	Reference Grid at which resultant contact force of primary body to secondary body is evaluated.  Default of FGRCN1 is FGRCEN
ICOORD	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)
IGLUE	Flag to activate glue options (Integer $\geq 0$ ). Default is 0, no glue option. See JGLUE option for controlling a glue contact type in details. This field must be specified for the permanently glued contact. See Remarks 4. and 5. A negative value of IGLUE indicates that this method of glue will be used for the entire Step in SOL 400. See Remark 9.
1 or -1	Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body. For the permanently glued contact, this option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.
2 or -2	Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.
3 or -3	Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body. For the permanently glued contact, this option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified.
4 or -4	Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.

ISEARCH	<p>Option for contact searching order, from Secondary to Primary or from Primary to Secondary, for deformable contact bodies. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0)</p> <p>0 (Double orders search) the search order is from lower BCBODY1 ID's to higher ones first. Then it searches the opposite order. See Remark 3.</p> <p>1 (Single order search) the searching order is from Secondary to Primary.</p> <p>2 (Single order search) let the program decide which search order. See Remark 3.</p> <p>Note that ISTYP in BCB DPRP or BCBODY is supported with ISEARCH=0 only.</p>
JGLUE	<p>This option is only relevant if the glue option is invoked (JGLUE &gt; 0). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force (related to FNTOL parameter of BCPARA or BCONPRP entry). Enter 2 to activate breaking glue (related to BGM, BGN, BGSN and BGST parameters of BCONPRP entry). (Integer; Default = 0) This parameter is ignored for the permanently glued contact or step glued contact, see Remark 9.</p>
PENALT	<p>Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real <math>\geq 0.0</math>; see Remark 7. for default)</p>
SLIDE	<p>Delayed slide off distance. This entry should not be made unless ICOORD <math>\geq 2</math> (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default=0.0)</p>
STKSLP	<p>Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real <math>\geq 0.0</math>; Default = 0.0) See Remark 8.</p>
TPENALT	<p>Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real &gt; 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.</p>
OPINTRF	<p>The method for interference fit. (Integer<math>\geq 0</math>; Default=0) See Remark 10.</p> <p>0 small interference fit (solved in one increment)</p> <p>1 interference fit varying with time; resolved in the normal direction</p> <p>2 interference fit varying with time in a user specified direction</p> <p>3 interference fit varying with time using scale factor</p> <p>4 automatic interference fit varying with time and node location</p>

TBINTRF	TABLED1 id used by interference fit. (Integer>0; Required only when OPINTRF > 0) When OPINTRF=1, TABLED1 id giving variation of interference closure with respect to time When OPINTRF=2, TABLED1 id giving variation of magnitude of translation vector with respect to time When OPINTRF=3, TABLED1 id giving variation of magnitude of scale factor with respect to time When OPINTRF=4, TABLED1 id giving variation of magnitude of penetration vector projected node position on the primary segment with respect to time
CBINTRF	Contact body selection for interference fit. Only required when OPINTRF=2, 3 or 4. (Integer=0 or 1; Default=0) 0 is secondary body, 1 is primary body When OPINTRF=2, secondary or primary body for which translation vector to be used When OPINTRF=3, secondary or primary body for which scale factors to be used. When OPINTRF=4, secondary or primary body of which penetrations to be determined.
CDINTRF	Coordinate system ID of VXINTRF, VYINTRF and VZINTRF for interference fit. Only required when OPINTRF=2, 3. (Integer≥0; Default=0) When OPINTRF=2, Coordinate system for direction cosines for the translation vector. When OPINTRF=3, Coordinate system for scale factor vector.
VXINTRF	X component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector
VYINTRF	Y component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector.
VZINTRF	Z component of vector for interference fit. Only required when OPINTRF=2, 3. (Real; default=0.0) When OPINTRF=2, Direction cosines for the translation vector. When OPINTRF=3, scale factor vector.
PTINTRF	Penetration search tolerance for interference fit. Only required when OPINTRF=4. (Real≥0.0; default=error tolerance*100)
XCINTRF	X component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).
YCINTRF	Y component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).
ZCINTRF	Z component of center of scaling. Only required when OPINTRF=3. (Real; default=0.0) Center is described in global coordinate (Coord0).
OPINGP	Initial gap or overlap option. See Remark 1. (Integer ≥0; default=0) 0 not used 1 initial gap or overlap

TOLINGP	Search tolerance of initial overlap. (Real $\geq 0.0$ ; default= average edge length of all contact segments, required only when OPINGP =1)
TOLING1	Search tolerance of initial gap. (Real $\geq 0.0$ ; default=average edge length of all contact segments, required only when OPINGP =1) See Remark 14.
CDINGP	Contact body to be adjusted. (Integer $\geq 0$ ; default=0, required only when OPINGP =1) 0 secondary body 1 primary body
MGINGP	Gap or overlap magnitude. (Real ; default=0.0, required only when OPINGP =1) > 0 gap =0 Preserve initial clearance distance. See Remark 11. < 0 overlap
SFNPNTL	Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFNPNTL*PENALT is used in analysis. If this field is left default, value of SFNPNTL entered in BCPARA will be used. A nonblank value entered in SFNPNTL will override the value of SFNPNTL in BCPARA.
SFTPNTL	Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFTPNTL*TPENALT is used in analysis. If this field is left default, value of SFTPNTL entered in BCPARA will be used. A nonblank value entered in SFTPNTL will override the value of SFTPNTL in BCPARA.
TCNTCTL	Flag to define the touching contact status in linear perturbation and ignoring the touching Contact status determined by the contact analysis in NLSTATIC analysis at the load point assigned by NLIC. (Integer; Default = 0). Value of TCNTCTL defined in BCONPRG will override one defined in BCPARA. 0 - Skip this option, keep the original contact status obtained in NLSTATIC 1 - Assuming sliding status of touching contact for contact and frictional stiffness matrix calculation. 2 - Assuming sticking status of touching contact for contact and frictional stiffness matrix calculation. 3 - Assuming glued contact status for touching contact for contact stiffness matrix calculation.

Remarks:

1. The multipoint constraint equations (MPC equations) internally created from general contact or glued contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH.
2. COPTS and COPTM are packed numbers designating how the surfaces may contact using the formula

$$COPT_x = A + 10 * B + 1000 * C$$

where the following codes apply:

A: the outside of the solid elements in the body.

- = 1: the outside will be in the contact description (Default).

B: (flexible bodies): the outside of the shell elements in the body.

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default).
- = 2: only bottom faces will be in the contact description, thickness offset will be included.
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored.
- = 4: only top faces will be in the contact description, thickness offset will be included.
- = 5: only top faces will be in the contact description, shell thickness will be ignored.
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored.

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

Note: for segment-to-segment contact only use B=1 or B-6.

Note: For thermal contact with 2D shell elements, i.e. uniform temperature gradient across thickness, do not use B=2 or B=3.

B: (rigid bodies): the rigid surface.

- = 1: the rigid surface should be in the contact description (Default).

C: (flexible bodies): the edges of the body.

- = 1: only the beam/bar edges are included in the contact description (Default).
- = 10: only the free and hard shell edges are included in the contact description.
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description.

Note that C has no effect if beam-to-beam contact is not switched on. (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

3. When ISEARCH=0 (and ISTYP=0 in default on BCB DPRP), the search order is from lower BCBODOY1 ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY1 ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY1. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2.

CL1= Min(1/20 of the smallest edge, 1/4 of the smallest thickness) of BCBODY1 ID1

CL2= Min(1/20 of the smallest edge, 1/4 of the smallest thickness) of BCBODY1 ID2

The search order is from lower BCBODY1 ID1 to higher BCBODY1 ID2 if  $CL1 \leq 1.05 * CL2$ . Otherwise, if  $CL1 > 1.05 * CL2$  the search order is from BCBODY1 ID2 to BCBODY1 ID1.

4. For the permanently glued contact, IGLUE = ±2 or ±4 is favorable to pass the GROUND CHECK. The initial stress free contact is also available to preserve the six rigid body modes with IGLUE = ±1 or ±3.

5. A permanently glued contact for small rotation cannot be used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the permanently glued contact for large deformation and rotation (IGLUE < 0) or general contact (IGLUE > 0) with glued option must be performed.
6. By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

$$\text{AUGDIST} = 0.05 L$$

7. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$\text{PENALT} = \frac{0.5(S_i + S_j)}{L}$$

The body stiffness ( $S_i$  and  $S_j$ ), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length ( $L$ ) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and is given by

$$\text{PENALT} = \frac{1000S_i}{L}$$

8. If STKSLP is set to 0.0 (Default), the sticking stiffness  $K1$  is equal to the maximum friction force ( $\mu \cdot F_{\text{normal}}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise,  $K1$  is equal to the maximum friction force divided by the value of STKSLP.
9. A negative value of IGLUE enforces the gluing of the pair of bodies in the BCONNECT that reference this BCONPRG is based upon the geometric conditions at the beginning of the step and will not change over this step. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.
10. For interference fit contact, the following options are available.  
CINTERF, OPINTRF, TBINTRF, CBINTRF, CDINTRF, VXINTRF, VYINTRF, VZINTRF, PTINTRF, XCINTRF, YCINTRF and ZCINTRF
11. For setting the initial gap or overlap, the following options are available. OPINGP, TOLINGP, CDINGP and MGINGP.

When OPINGP is 1 in means that there it will try to preserve a distance between the two bodies often called the clearance distance.

When MGINGP is zero, the clearance is not changed from the distance between the original coordinate positions of the nodes. This is call the distance  $D0$

When  $MINGP$  is  $> 0$  it means the distance between the bodies can shrink (i.e. the bodies can get closer together), but when the distance reaches  $D0 - MINGP$ . The clearance condition is satisfied and this clearance distance ( $D0 - MINGP$ ) is preserved.

When  $MINGP$  is  $< 0$  it means that it considers the bodies to be overlapping, so it first tries to separate them by a distance =  $|MINGP|$ ; then it imposes the clearance condition such that the clearance distance is  $D0 + |MINGP|$

12. Please note that
  - Cohesive Contact is supported for only for segment-to-segment contact.
  - All limitations of segment-to-segment contact is applicable for cohesive contact feature also.
  - Breaking Glue feature is not supported with Cohesive contact.
  - $FGCNST/FGCNSTI/FGCNSTR$  are mutually exclusive;  $FGCTST/FGCTSTR/FGCRSTI$  are mutually exclusive.
  - $FGRCEN/FGRCN1$  applies to all contact pairs, not limited to cohesive contact ( $FGFLG=1$ ) pairs .
13. When the initial stress-free( $ICOORD=1$ ), Node-to-Segment updates model geometry to close the gap, nodes are projected onto the contact body; but Segment-to-Segment treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
14.  $TOLINGP$  and  $TOLING1$  follows the rules below.
  - a. When both of  $TOLINGP$  and  $TOLING1$  are not defined, the default value of average edge length of all contact segments will be used for both.
  - b. When  $TOLINGP$  is only defined,  $TOLINGP$  value will be used for both of  $TOLINGP$  and  $TOLING1$ .
  - c. When  $TOLING1$  is only defined,  $TOLING1$  value will be used for  $TOLING1$  only, and the default value of average edge length of all contact segments will be used for  $TOLINGP$ .
  - d. When both of  $TOLINGP$  and  $TOLING1$  are defined, each values are used for each case

## BCONPRG-700

## Geometric Contact Parameters of Touching Bodies in SOL 700

Defines geometric contact parameters of touching bodies used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10	11
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3		
	PARAM4	VAL4	PARAM5	VAL5	-etc.-					

Example:

BCONPRP	90		IGLUE	1						
---------	----	--	-------	---	--	--	--	--	--	--

Describer      Meaning

PID              Parameter identification number (Integer > 0).

PARAMi        Name of a parameter. Allowable names are given in [Table 9-5](#) (Character).

VALi            Value of a parameter. See [Table 9-5](#) (Real or Integer).



Table 9-5 Geometric Contact Parameters of Touching Bodies in SOL 700

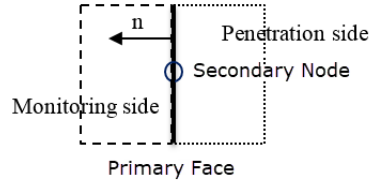
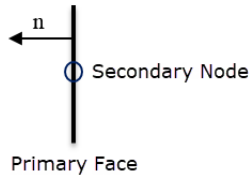
Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
IGLUE	<p>Flag to activate glue option (Integer = 0 or 1). Default is 0, no glue option. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, tied contact methods (METHOD=TIEDxxx) can assign glue contact types as well.</p> <p>0. No glue contact.</p> <p>1. Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.</p>
JGLUE	<p>This option is only relevant if the glue option is invoked (IGLUE &gt; 0). See METHOD=TIEDxxx. (Integer ≥ 0; Default = 0) The following options are available:</p> <p>0 Secondary nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.</p>
METHOD	<p>Character, Influences the contact type used. See Remark 18. (Character, Default = FULL)</p> <p>Options are:</p> <p>FULL: Regular Contact</p> <p>AIRBAG: Single Surface Contact</p> <p>SS1WAY: Surface To Surface One Way</p> <p>SS2WAY: Surface To Surface Two Way</p> <p>RB1WAY: Rigid Body One Way To Rigid Body</p> <p>RB2WAY: Rigid Body Two Way To Rigid Body</p> <p>RNRB: Rigid Nodes To Rigid Body</p> <p>TIEDNS: Tied Nodes to Surface</p> <p>TIEDNSO: Tied Nodes to Surface with Offset</p> <p>RELLIPS: Tied contact between grid points or rigid materials to ATB dummies. See Remark 4.</p> <p>BELT: Suited for modeling contact between a belt element and a rigid structure. Primary secondary contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the secondary nodes exactly on top of the primary face. The secondary node does not slide relative to the primary face when the friction coefficient (FS) is set to 1E20</p>

	<p>BELT1: Identical to BELT algorithm, except that the secondary nodes are initially repositioned on top of the closest primary face. All secondary nodes initially penetrated or within a distance of INITMON from a primary face, are repositioned.</p> <p>DRAWBEAD: Suited for modeling a drawbead. STYPE Must be GRID. The list of secondary nodes must be ordered along the drawbead line. MTYPE Must be SURF. The restraining force per unit drawbead length is specified in the field DRWBEADF. It is V2 version contact.</p> <p>DRAWBDV4: Suited for modeling a drawbead. STYPE Must be GRID. The list of secondary nodes must be ordered along the drawbead line. MTYPE Must be SURF. The restraining force per unit drawbead length is specified in the field DRWBEADF. It is V4 version contact.</p>
ADAPT	<p>Character, influences the contact type used. (Character, Default = NO)</p> <p>Options are NO or YES.</p> <p>When ADAPT=YES, the BCBODY1 entries IDPRIMi must be defined as:</p> <p style="padding-left: 40px;">behav=DEFORM</p> <p style="padding-left: 40px;">bsid references a BCPROP</p>
THICK	Shell thickness scale factor. (Real > 0.0; Default = same as the value in DYPARAM,CONTACT,THICK)
THICKOF	Artificial contact thickness offset. (Real > 0.0; Default = 0.0)
PENV	Overwrites the default maximum penetration distance factor. (Real > 0.0;Default = 1.E20)
MAXPAR	Maximum parametric coordinate in segment. (Real > 0.0; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.
SOFT	<p>Soft constraint option: (Integer ≥ 0; Default = 1)</p> <p style="padding-left: 40px;">1 Soft constraint formulation (same as original Dytran method).</p> <p>In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. This method is more suited for contact between two materials where the elastic moduli vary greatly.</p>
IGNORE	<p>Ignore initial penetrations. (Integer, Default = 1)</p> <p style="padding-left: 40px;">0 Take default from PARAM,DYCONIGNORE*,&lt;value&gt; (Default)</p> <p style="padding-left: 40px;">1 Allow initial penetrations to exist by tracking it</p> <p style="padding-left: 40px;">2 Move nodes to eliminate initial penetrations</p>

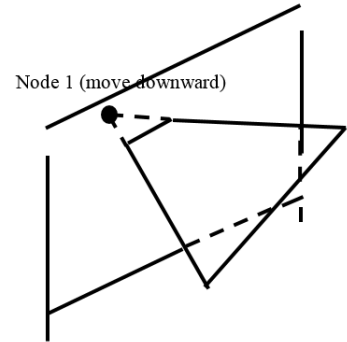
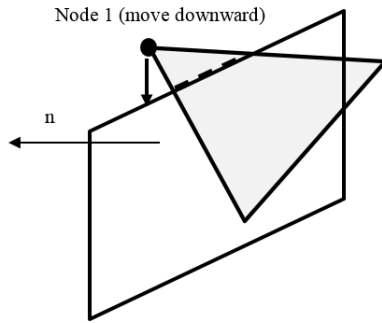
AUTO	<p>Activation of automatic contact (Character, Default = Yes.)</p> <p>Options are:</p> <p style="padding-left: 40px;">YES Automatic Contacts Activated</p> <p style="padding-left: 40px;">NO Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.</p>
SIDE	<p>Defines which side will be the monitoring side of a primary face. The opposite side of the primary face will be the penetration side. See Remark 1. (Characters; default=BOTH)</p> <p>BOTH: The side from which a secondary node approaches the primary face will become the monitoring side.</p> <p>TOP: The monitoring side will always be on the side of the primary face that its normal is pointing at.</p> <p>BOTTOM: The monitoring side is always on the opposite side of the primary face that its normal is pointing at.</p>
WEIGHT	<p>The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH)</p> <p>BOTH: <math>M\_scale = (M\_secondary \cdot M\_primary) / (M\_secondary + M\_primary)</math></p> <p>SECNDRY: <math>M\_scale = M\_secondary</math></p> <p>PRIMARY: <math>M\_scale = M\_primary</math></p> <p>NONE: <math>M\_scale = 1.0</math></p>
MONDIS	<p>Defines the fixed part of the monitoring distance. When the normal distance of a secondary node to a primary face becomes smaller than the monitoring distance the secondary node will tag itself to the primary face. The side from which the secondary node is moving towards the primary face becomes the monitoring region. (Character; default=FACTOR)</p> <p>FACTOR: The monitoring distance is equal to a factor times a characteristic length of the primary faces. The factor is specified in MONDISV.</p> <p>DISTANCE: The monitoring distance is specified in MONDISV.</p>
MONDISV	<p>Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)</p>
INITMON	<p>Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real &gt; 0.0; default=MONDIS)</p>

Remarks:

1. When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SECNDRY body and BCELIPS is only available for PRIMARY body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
2. The options TOP/BOTTOM are useful in the following cases:
  - a. When a secondary node initially is located on the primary face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the primary surface is defined:  
SIDE=TOP case:



- b. When hooking of secondary nodes on the wrong side of a primary face might occur. This often is the case when the primary face is at the edge of a shell element structure:  
SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



3. Recommended usage of WEIGHT:

SECNDRY	PRIMARY	WEIGHT
Deformable	Deformable	BOTH
Deformable	Rigid	SECNDRY
Rigid	Deformable	PRIMARY
Rigid	Rigid	NONE

Default setting when BCELIPS is used in SECNDRY or PRIMARY:

SECNDRY	PRIMARY	WEIGHT
Non-BCELIPS	Non-BCELIPS	NONE
	BCELIPS	

- When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SECNDRY body and BCELIPS is only available for PRIMARY body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable.

## BCONPRP Physical Contact Parameters of Touching Bodies in SOLs 101 and 400

Defines physical contact parameters of touching bodies used in SOLs 101 and 400 only. The parameters defined here are referenced by the BCONNECT entry. This entry is used in conjunction with the BCONPRG entry.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRP	90		FRIC	0.3					
---------	----	--	------	-----	--	--	--	--	--

Descriptor      Meaning

PID              Parameter identification number (Integer > 0).

PARAMi        Name of a parameter. Allowable names are given in [Table 9-6](#) (Character).

VALi            Value of a parameter. See [Table 9-6](#) (Real or Integer).

Table 9-6 Physical Contact Parameters of Touching Bodies in SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
BGM	Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)
BGN	Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 1. (Real; Default = 2.0)
BGSN	Maximum normal stress for breaking glue. See Remark 1. (Real; Default = 0.0)
BGST	Maximum tangential stress for breaking glue. See Remark 1. (Real; Default = 0.0)
BNC	Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0.)
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 1.0)
DQNEAR	Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. See Remark 2. (Real; Default = 0.0; which means near contact does not occur)
EMISS	Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs. temperature or a TABL3D entry specifying the emissivity vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
FNTOL	Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. Its default value is dependent on the IBSEP value. (Real)
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real $\geq$ 0.0 or Integer $>$ 0; Default = 0.0)
FRLIM	Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction stress will be reduced so that the maximum friction stress is given by $\min(\mu\sigma_n, \sigma_{\text{limit}})$ , with $\mu$ the friction coefficient and $\sigma_n$ the contact normal stress. (Real; Default = 1.0E20)

HBL	Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs. temperature or a TABL3D entry specifying the convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
HCT	Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs. temperature or a TABL3D entry specifying the contact heat transfer coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.e., when IGLUE > 0.)
HCV	Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs. temperature or a TABL3D entry specifying the near field convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)
HGLUE	Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. Ignored in a pure structural analysis. See Remarks 3. , 4. and 5. ( $0 \leq \text{Integer} \leq 1$ ; Default = 0)
HNC	Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs. temperature or a TABL3D entry specifying the near field natural convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0).
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs. temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs. temperature and possibly other variables. Ignored in a pure structural analysis. See Remark 5. (Real or Integer; Default = 0.0)

#### Remarks:

1. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/\text{BGSN})^{**}\text{bgn} + (\sigma_t/\text{BGST})^{**}\text{bgm} > 1.0$$



When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If  $BGSN = 0.0$  the first term is ignored. Similarly, the second term is ignored if  $BGST = 0.0$ . For SOL 400, parameters  $BGSN$ ,  $BGST$ ,  $BGM$  and  $BGN$  are only required if  $JGLUE = 2$ . If both  $BGSN$  and  $BGST$  are equal to 0.0,  $JGLUE$  will be set to 0 internally. If Step Glue is used, then Breaking Glue is inactive in this step. If Permanent Glue is active, then Breaking Glue is inactive.

2. For near contact the following convective heat flow law describes the heat exchange per unit area between the body areas (“near” contact is when bodies are not touching each other – but are close enough for convection and radiation.):

$$q = HCV \cdot (T_A - T_B) +$$

$$HNC \cdot (T_A - T_B)^{BNC} +$$

$$HNL \cdot (T_A^{BNL} - T_B^{BNL}) +$$

$$\sigma \cdot EMISS \cdot (T_A^4 - T_B^4) +$$

$$\left[ HCT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A - T_B)$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

3. For hard contact (i.e., two bodies are actually touching each other – heat transfer mode between the bodies is conduction), with  $HGLUE=1$ :
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
  - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures i.e. Gibbs Phenomenon or the temperature oscillations at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

4. For hard contact, with  $HGLUE=0$ :

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

5. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D). Table IDs of tables used on the BCBDPRP entry and the BCONPRP entry must be unique.
6. Parameters BGM, BGN, BGSN, BGST and DQNEAR are not supported in segment to segment contact analysis.

**BCONPRP-700****Physical Contact Parameters of Touching Bodies in SOL 700**

Defines physical contact parameters of touching bodies used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCONPRP	PID		PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	
	PARAM4	VAL4	PARAM5	VAL5	-etc.-				

Example:

BCONPRP	90		FRIC	0.3					
---------	----	--	------	-----	--	--	--	--	--

Describer      Meaning

PID              Parameter identification number (Integer > 0).

PARAMi        Name of a parameter. Allowable names are given in [Table 9-7](#) (Character).

VALi            Value of a parameter. See [Table 9-7](#) (Real or Integer).

**Table 9-7      Physical Contact Parameters of Touching Bodies in SOL 700**

Name	Description, Type and Value (Default is 0 for integer, 0.0 for Real Unless Otherwise Indicated)
FRIC	Static friction coefficient. When it is set to -1.0, different static friction coefficients on properties can be assigned using BCPROP1. (Real $\geq$ -1.0; Default=0.0 for SOL700)
FK	Kinetic coefficient of friction. (Real > 0.0; Default = 0.0)
EXP	Exponential decay coefficient. (Real > 0.0; Default = 0.0)
FACT	Scale factor for the contact forces. (Real > 0.0; Default = 0.1)

TSTART	Time at which the contact is activated. (Real > 0.0; Default = 0.0)
TEND	Time at which the contact is deactivated. (Real > 0.0; Default = 1.e20)
IADJ	<p>Adjacent material treatment option for solid elements. (Integer <math>\geq 0</math>; Default =1)</p> <p>0 Solid element faces are included only for free boundaries.</p> <p>1 Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.</p>
DAMPING	<p>Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a secondary node with respect to a primary face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES).</p> <p>YES damping is activated</p> <p>NO damping is not activated.</p>

## BCONUDS

## Contact Body User Defined Service

Allows the user to provide contact routines for use with enhanced SOL 400 contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BCONUDS	BID	BTYPE	GROUP	UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	“CHAR”	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

Example:

*In FMS Section of MSC Nastran input stream:*

```
CONNECT SERVICE CONTACT 'SCA.MDSolver.Util.Ums'
```

*In Bulk Data:*

BCONUDS	17	BCBODY	CONTACT	UFRIC					
BCONUDS	17	BCBODY	CONTACT	SEPFOR					
	REAL	.00134	1.467+4	.03					
BCONUDS	17	BCBODY	CONTACT	MOTION					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Descriptor	Meaning
BID	Contact body identification number defined by BCBODY or BCBODY1. (Integer > 0; Required)
BTYPE	The name of the contact entry. BCBODY/BCBODY1 (Character; no Default)
GROUP	The group name used for the FMS Section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. See Remark 6. and 7. (Character)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified contact property entry. (Integer, no Default)
“REAL”	Keyword indicating that the following data is real. (Character)

Describer	Meaning
RDATAi	Additional user supplied Real data not already existing on the specified contact property entry. (Real; no Defaults)
“CHAR”	Keyword indicating that the following data is Character. (Character)
CDATAi	Additional user supplied Character data not already existing on the specified contact property entry. (Character; no Default)

**Remarks:**

1. This entry triggers the call to a user contact subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The BID must match an existing BID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
5. Certain user subroutines may require integer or real data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. The following user subroutines are currently available for user convenience. See the User Defined Services manual for details.

Type	Uname	Purpose
BCBODY	motion	Define velocity of rigid surfaces
BCBODY	ufric	Define friction coefficients
BCBODY	sepfor	Define separation forces
BCBODY	sepstr	Define separations stresses
BCBODY	ubsqueal	Define friction coefficient and effective stiffness in brake squeal analysis

## BCONUDS

## Contact Body User Subroutines in SOL 600

Allows the user to provide contact routines for use with SOL 600 contact analysis.

## Format:

1	2	3	4	5	6	7	8	9	10
BCONUDS	BID	BTYPE		UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			

## Example:

BCONUDS	17	BCBODY		RTN1					
BCONUDS	17	BCBODY		RTN2					
	REAL	.00134	1.467+4	.03					
BCONUDS	17	BCBODY		RTN3					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Describer	Meaning
BID	Contact body identification number defined by BCBODY. (Integer > 0; Required)
BTYPE	The name of the contact entry. BCBODY (Character; no Default)
UNAME	The Marc user subroutine name. See Remarks 2. and 5. (Character; Required)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified MAT entry. See Remark 6. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATA <sub>i</sub>	Additional user supplied Real data. See Remark 6. (Real; no Defaults)

## Remarks:

1. The BID must match an existing BID.
2. Certain user subroutines may require integer or real data input as specified in the UserSubroutine interface document.
3. The following user subroutines are currently available for user convenience. See the User Subroutine interface document for details.

UNAME	Purpose
MOTION	Define velocity of rigid surfaces
UFRIC	Define friction coefficients
DIGEOM	Define rigid surfaces
SEPFOR	Define separation forces
SEPFORBBC	Define separation forces for beam-beam contact
SEPSTR	Define separations stresses
UFRICBBC	Define the friction coefficients for beam-to-beam contact with node-to-segment contact.  This user subroutine is not used for segment-to-segment contact; use UFRIC instead.
UHTNRC	Define thermal near contact film coefficients
UMDCOE	Define variable mass and diffusion coefficients and sink pressures on a free surface
UMDCON	Define variable mass diffusion coefficients on surfaces in contact with other surfaces
UMDNRC	Define mass diffusion coefficients of surfaces in contact with other surfaces
UNDRST	Define normal stresses for elements in contact

4. The BCONUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character\*16 name in common block /userch/
5. UNAME identifies the user subroutine name to be called.
6. IDATA and RDATA are not normally required. They are available should additional arguments beyond those described in Marc Vol D be required. Normally only the first line is entered.



**BCPARA****Contact Parameters -SOLs 101 and 400**

Defines contact parameters used in SOL 101 and SOL 400. This entry is not available in SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc.				

Example:

BCPARA		ERROR	0.1	BIAS	0.5				
--------	--	-------	-----	------	-----	--	--	--	--

Describer	Meaning
ID	Subcase to which the defined parameters belong. If ID is zero, the parameters belong to all subcases. (Integer $\geq 0$ ; no Default)
Param(i)	Name of a parameter. Allowable names are given in <a href="#">Table 9-7</a> . (Character)
Value(i)	Value of the parameter. See <a href="#">Table 9-7</a> . (Real or Integer)

Table 9-8 Contact Parameters -SOLs 101 and 400

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
ERROR	Distance below which a node is considered touching a body. (Real; Default = blank). Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest value resulting from: Either dividing the smallest nonzero element dimension (plates or solids) in the contact body by 20. Or dividing the thinnest shell thickness in the contact body by 4. This value is then used for all contact pairs.
BIAS	Contact tolerance bias factor. ( $0.0 \leq \text{Real} \leq 1.0.$ ); Default = 0.9 for IGLUE=0, if field left blank. Default = 0.0 for IGLUE <>0. Note that when IGLUE<>0, BIAS can only be given by the BCTABLE or BCONPRG. Default = 0.0 for BEHAVE=SYMM on BCBODY, if field left blank or 0.0.
FGCFLG	Flag to activate Cohesive (Flexible) Glued Contact: (See Remark 12.)
0	No Cohesive (Flexible) Glued Contact
1	Activate Cohesive (Flexible) Glued Contact
FGCNST	Equivalent normal contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCTST	Equivalent tangential contact stiffness of connector for Cohesive (Flexible) Glued Contact
FGCNSTR	ID of table for define contact normal stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCTSTR	ID of table for define contact tangential stress versus relative displacement for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCNSTI	ID of table for define contact normal stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGCTSTI	ID of table for define contact tangential stiffness versus relative displacement or temperature for Cohesive (Flexible) Glued Contact. Only TABL3D allowable.
FGRCEN	Reference Grid at which resultant contact force of secondary body to primary body is evaluated. See Remark 12.
FGCRCN1	Reference Grid at which resultant contact force of primary body to secondary body is evaluated.  Default of FGCRCN1 is FGRCEN
FNTOL	Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank). See Remark 10.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
MAXSEP	Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)	
ICSEP	Flag to control separation. Not used for segment-to-segment contact. (Integer ≥ 0; Default = 0)	
	0	The node separates and an iteration occurs if the force on the node is greater than the separation force.
	1	If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment.
	2	If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.
	3	Both 1 and 2 are in effect.
IBSEP	Flag for separation based on stresses or forces. (Integer ≥ 0; Default = 0)	
	0	Separation based on forces.
	1	Separation based on absolute stresses (force/area)
	2	Separation based on absolute stress (extrapolating integration point stresses)
	3	Relative nodal stress (force/area)
	4	Separation based on relative stress (extrapolating integration point stresses)
	Segment to segment contact ignores IBSEP and will use 2 internally. See Remarks 6. and 8.	
RVCNST	For the bilinear friction model, RVCNST is the slip threshold. (Real ≥ 0.0; Default = 0.0).	
	Default=0.0 means that the actual value is automatically calculated.	
FTYPE	Friction type. See Remark 5. (Integer)	
	0	No friction. (Default)
	6	Bilinear Coulomb friction.
	7	Bilinear Shear friction.
BEAMB	Beam-Beam contact flag. See Remark 7. (Integer 0 or 1)	
	0	No beam-beam contact. (Default for SOLs 101/400)
	1	Activate beam-beam contact options. (Default for SOL 700)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
NLGLUE (SOLs 101 and 400 only)	If all secondary's for the BCTABLE or BCONPRG corresponding to the first loadcase (first subcase and first step) contain IGLUE >0, permanent glued contact with small rotation condition will be used for all SECNDRY entries in all subcases and all steps unless BCPARA,0,NLGLUE,1 is specified. If IGLUE < 0 exists, permanent glued contact for large deformation and large rotation is activated. (Integer; Default = 0)
NODSEP	Skip separation check in contact for any grid already touched and separated.  NODSEP  times within the current increment. (Integer; Default = 5)
	> 0      Keep grid contact status
	< 0      Separate grid from contact
METHOD	Flag to select Contact methods. (Character)
	NODESURF      Node to segment contact. (Default)
	SEGTOSEG      Segment to segment contact.
AUGMENT	Augmentation method used in a segment-to-segment contact analysis. (Integer)
	0      No augmentation (Default)
	1      Augmentation based on a constant Lagrange multiplier field for linear elements and on a (bi)linear Lagrange multiplier field for quadratic elements
	2      Augmentation based on a constant Lagrange multiplier field
	3      Augmentation based on a (bi)linear Lagrange multiplier field
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real > 0.0; see BCTABLE or BCONPRG entries for default)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real > 0; see BCTABLE entry for default)
SLDLMT	Maximum allowed sliding distance, beyond it the contact segments are redefined, for segment to segment contact analysis with large deformation. (Real ≥ 0.0; Default = 0.0) See remark 9.
SEGSYM	Specify symmetric or non-symmetric friction matrix in segment to segment contact analysis. (Integer 0 = symmetric matrix or 1 = non-symmetric matrix; Default = 0)
ERRBAS	Error computation option. (Integer 0 = compute error globally or 1 = calculate error based on each pair of secondary-primary; Default = 0)
DDULMT	Maximum value of DDU in a Newton-Raphson cycle. (Real ≥ 0.0; Default = 0.0, no limitation)
TAUGMNT	Augmentation for the sticking part of friction in a segment-to-segment contact analysis. (Integer 0 = no augmentation or 1 = use augmentation; Default = 0)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
TPENALT	Augmented Lagrange penalty factor for sticking part of friction, used by the segment-to-segment contact algorithm only. (Real > 0.0) The default is PENALT/1000, where PENALT parameter is the Augmented Lagrange penalty factor for normal contact.
STKSLP	Maximum allowable slip distance for sticking, beyond it there is no sticking, only sliding exists, used by the segment-to-segment contact algorithm only. (Real ≥ 0.0; Default = 0.0) See Remark 11.
THKOFF	Ignore thickness from the tolerance used by ISEARCH=2 in node-to-surface contact or from the characteristic length (for PENALT and AUGDIST) in segment-to-segment contact. (Integer 0 = do not ignore thickness or 1 = remove thickness; Default = 0)
LINCNT	Flag for Linear Contact (under infinitesimal assumption, small sliding with small deformation and rotation). Supports both node-to-segment and segment-to-segment method. See Remark 14.. 0 - (Default) general contact 1 - activate Linear Contact, the contact force is distributed based upon undeformed geometry. -1 - force Linear Contact with large displacement (not recommended).
SFNPNT	Scale factor of augmented Lagrange penalty factor along contact normal direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFNPNT*PENALT is used in analysis. A nonblank value of SFNPNT entered in BCONPRG will override it.
SFTPNT	Scale factor of augmented Lagrange penalty factor along contact tangential direction; used by segment-to-segment contact method only. (Real>0.0;Default=1.0) SFTPNT*TPENALT is used in analysis. A nonblank value of SFTPNT entered in BCONPRG will override it.
SEGANGL	Minimum angle between segment normal vectors allowing the segments to come into contact; used by segment-to-segment contact only. (Real>90.0 <180.0; Default=120.0).
TCNTCTL	Flag to define the touching contact status in linear perturbation and ignoring the touching Contact status determined by the contact analysis in NLSTATIC analysis at the load point assigned by NLIC. (Integer; Default = 0) Value of TCNTCTL defined in BCONPRG will override one defined in BCPARA. 0 - Skip this option, keep the original contact status obtained in NLSTATIC. 1 - Assuming sliding status of touching contact for contact and frictional stiffness matrix calculation. 2 - Assuming sticking status of touching contact for contact and frictional stiffness matrix calculation. 3 - Assuming glued contact status for touching contact for contact stiffness matrix calculation.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
VERSION	Defaults version control in Segment-to-Segment method, see remark 15.
	1           Version 1
	2 (default)   Version 2 (lower default penalty, recommended)
BACKCTL	Backward compatible bit-wise control in contact analysis, see remark 16. 1 <sup>st</sup> to 7 <sup>th</sup> bit segment-to-segment only
	-1           Activate all of following items (back to 2020 SP1)
	0           None of following items will be activated
	1           Body order independent
	2           Evaluation of contact matrix/force on segment coordinates (old logic)
	4           Segment sequence renumbering (old method)
	8           No reset iteration count after separation (old logic)
	16          Creating new polygon under sliding condition (old logic)
	32          Scaled incremental displacement update in small rotation (old method)
	64          Ramping down penalty when angle of segment normals between 180 degree and SEGANGL
	128         Old algorithm of nodal projection in contact  2021.3 release onwards, the default uses the enhanced contact projection algorithm, which helps in consistent contact detection at corners and for warped surface.
GLUEOUT	Contact force and stress output preference in glued contact.
	0           Normal and tangential components are separated (segment-to-segment only)
	1 (default)   Normal and tangential components are combined
SEPACC	Flag to control the accelerated separation check in node-to-segment contact for SOL 400. See remark 17. (Integer ≥ 0; Default = 0).
	0           Accelerated separation check turned off
	1           Accelerated separation check turned on
DYNPRFA	Dynamic contact projection factor (0.0 ≤ Real ≤ 1.0; Default= 0.0)
	A value of 0.0 may show some penetration.
	Values greater than 0 reduce penetration and can introduce transient vibration. It is strongly recommended to use the default values in NLTRAN to avoid the chattering in contact analysis.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
OSPLNFRQ	Output frequency of smooth spline representation of deformable contact body when BOUTPUT output requested. (Integer $\geq -1$ ; Default=-1).	
	-1	Initial output only
	0	No output
	n	Every n <sup>th</sup> output increment along with initial output.  Note that OSPLNFRQ is based on the number of output load increments, but not calculated load increments. It is determined by "NO" in fixed time stepping or "NOUT" in adaptive time stepping on NLSTEP.

#### Remarks:

1. The ID of BCPARA should be left blank or a value of zero. Blank is treated as zero.
2. Only one BCPARA entry should be made. Multiple BCPARA entries with ID=0 cause a Fatal Error.
3. All SOLs only support ID=0, this entry is applied to all the subcases, except the corresponding ones are assigned in BCTABLE or BCONPRG.
4. Refer to the Case Control command [BCONTACT \(Case\)](#), to see the lists of parameters in BCPARA that are not supported by SOLs 101 and 400.
5. In SOLs 101/400, FTYPE only supports options 0, 6, and 7. Friction with segment-to-segment contact is a pre-release capability in the 2012 release. In the 2012.2 version and later, it is fully supported.
6. Shell elements (CQUAD4 CQUAD8 CTRIA3 CTRIA6) in 3D Contact only supports IBSEP = 0, 1 and 3. If IBSEP = 2 or 4 in this case, it will automatically be set to 1 or 3, respectively. In SOLs 101/400, LINQUAD is not supported and it is set to -1 in order that the mid-side nodes are considered in Contact. Quadratic shell elements consider mid-side nodes only in 2D Contact.
  - a. If IBSEP is set to 1, then for bodies consisting of mid-side node elements, IBSEP will automatically be set to 2. This allows for using IBSEP=1 for elements without mid-side nodes and IBSEP=2 for elements with mid-side nodes in one analysis.
  - b. If IBSEP is set to 3, then for bodies consisting of mid-side nodes elements, IBSEP will automatically be set to 4. This allows for using IBSEP=3 for elements without mid-side nodes and IBSEP=4 for elements with mid-side nodes in one analysis.

This rule is always applied to the solid elements with mid-side nodes. However, the mid-side nodes of CQUAD8 and CTRIA6 shell elements are ignored in node-to-segment 3D contact with separation because there is no normal stress in shell elements, and the contact normal forces for these elements do not give a good estimation of the nodal contact stress. Hence it is not recommended to use CQUAD8 and CTRIA6 shell elements in a node-to-segment contact analysis with separation.

7. For SOLs 101/400 if BEAMB=1, Bulk Data entry BCBMRAD must be entered. Those entries are not required in the beam contact of segment-to-segment contact.

8. Parameters MAXSEP, ICSEP, IBSEP, RVCNST and BEAMB are not supported in segment to segment contact analysis.
9. The default of SLDLMT parameter is 5 times the default error tolerance. See the description of ERROR parameter for the definition of default error tolerance.
10. IBSEP and FNTOL are related to each other. The Default value of FNTOL is dependent on IBSEP value.
  - a. IBSEP=0 : Separation is based on forces. A contact node will separate if the contact pulling force exceeds FNTOL. If FNTOL is zero it will be taken as the absolute value of the largest component in the residual force vector.
  - b. IBSEP=1 : Separation is based on nominal contact stresses (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL. If FNTOL is zero, it will be 0.1 times of the maximum nominal contact pushing stress of any node of any contact body.
  - c. IBSEP=2 : Separation is based on nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL. If FNTOL is zero it will be 0.1 times of the maximum nodal pushing stress of any node of any contact body
  - d. IBSEP=3 : Separation is based on relative nominal contact stress (force/area). A contact node will separate if the nominal contact pulling stress exceeds FNTOL times of maximum nominal contact pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1
  - e. IBSEP=4 : Separation is based on relative nodal stress (extrapolating integration point stresses). A contact node will separate if the nodal pulling stress exceeds FNTOL times of maximum nodal pushing stress of any node of any contact body. If FNTOL is zero, it will be taken as 0.1
11. If STKSLP is set to 0.0 (Default), the sticking stiffness  $K_1$  is equal to the maximum friction force ( $\mu \cdot F_{NORMAL}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise,  $K_1$  is equal to the maximum friction force divided by the value of STKSLP.
12. The parameters of cohesive contact defined in BCPARA will be applied to all the contact pairs unless:
  - The contact pair has its own definition for cohesive contact in BCONPRG; or
  - Its IGLUE=0, i.e., general touching contact.
  - FGRCEN/FGRCN1 applies to all contact pairs, not limited to cohesive contact (FGCFLG=1) pairs.

Cohesive contact is only applied to glued contact, but not touching contact.

13. If Modules are present then this entry may only be specified in the main Bulk Data section.
14. LINCNT does not support RIGID contact body with velocity control.
15. Version 1 is equivalent to system cell 701=0; Version 2 is equivalent to system cell 701=1  
Version 2 is improved default values for contact problems involving geometrically and physically nonlinear behavior:
  - a. Augmented Lagrange penalty factor. For bodies with the same body stiffness, this factor is chosen to be lower than the default value. For a body combination with different values of the body stiffness, the smallest of the two is used.



- b. Penetration distance beyond which an augmentation is applied. This distance is chosen to be larger than the default value, thus avoiding too many iterations due to augmentation. It is recommended to use an augmentation procedure for contact.
  - c. Augmented Lagrange penalty factor for sticking contact. This factor is chosen to be dependent on the normal pressure.
16. Following features are defaults of segment-to-segment method since 2021.0
  - Body order dependent
  - Evaluate contact matrix/force based upon true updated segment coordinates
  - New segment sequence renumbering
  - Reset iteration count after separation (Version 2 only)
  - New logic in creating new polygons under sliding condition
  - Scaled incremental displacement update in large rotation
  - Ramping down penalty when angle of segment normals below SEGANGL (minimum segment angle, default 120 degree)
17. SEPACC=0: Nodes are checked for separation only after satisfying all specified convergence criteria.  
SEPACC=1: Nodes are checked for separation as soon as the maximum residual force in the model drops to less than 10% of the maximum reaction force in the model.
  - a. SEPACC is only applicable to node-to-segment method, not supported in segment-to-segment method.
  - b. Accelerated separation check could reduce iteration when EPSP less than 0.1
  - c. Accelerated separation check may increase total separation count, hence default MAXSEP is preferred when SEPACC=1.
  - d. Accelerated separation check algorithm is only available in NLSTEP scheme, not supported in NLPARM.

## BCPARA

## Contact Parameters in SOL 600

Defines contact parameters used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCPARA	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

BCPARA		NBODIES	4	BIAS	0.5				
--------	--	---------	---	------	-----	--	--	--	--

Describer	Meaning
ID	ID is not used and should be set to zero. Only one BCPARA should be entered and it applies to all subcases. (Integer)
Param(i)	Name of a parameter. Allowable names are given in <a href="#">Table 9-7</a> . (Character)
Value(i)	Value of the parameter. See the <a href="#">Table 9-7</a> . (Real or Integer)

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)
NBODIES (2,1)	Number of contact bodies defined in the analysis. (Integer $\geq 0$ or blank)
MAXENT (2,2)	Maximum number of entities created for any contact body. (Integer $> 0$ or blank; default is max element number or 1.5 times the number of nodes whichever is smaller)
MAXNOD (2,3)	Maximum number of nodes that lie on the periphery of any deformable contact body. (Integer $> 0$ or blank; default is the number of nodes)
ERROR (3,2)	Distance below which a node is considered touching a body. Automatically calculated if left blank. If left blank, the code calculates ERROR as the smallest nonzero element dimension divided by 20 or the shell thickness divided by 4. (Real; Default = blank)
BIAS (3,6)	Contact tolerance bias factor. (Real, $0 \leq \text{BIAS} \leq 1$ , Default = 0.9 if field left blank or not entered in the file. To obtain a near zero value, enter 1.0E-16)
ISPLIT (2,7)	Flag for increment splitting procedure. (Integer $\geq 0$ ; Default = 3 for statics and 0 for dynamics)
	0 Uses increment splitting procedures for the fixed time step procedures.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	1	Suppresses splitting for the fixed time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods
	2	Suppresses splitting for adaptive time step procedures. Any penetration that occurred in the previous increment is adjusted for equilibrium at the start of the next increment. This method may require smaller time steps than the other methods.
	3	Uses contact procedure which does not require increment splitting (3 is not available for dynamics). If a run does not converge due to an excessive number of “iterative penetration checking” messages, ISPLIT=2 may help, however the time steps may need to be set to smaller values.
FNTOL (3,5)	Separation force (or stress if separation is controlled by stress as determined by IBSEP) above which a node separates from a body. Automatically calculated if left blank. (Real; Default = blank)	
MAXSEP (2,6)	Maximum number of separations allowed in each increment. After MAXSEP separations have occurred, if the standard convergence tolerance conditions are achieved, the step will converge. (Integer > 0; Default = 9999)	
ICHECK (2,8)	Flag for interference kinematic check and bounding box check. (Integer > 0)	
	1	Activates interference kinematic check.
	2	Suppress bounding box checking.
	3	No reset of NCYCLE to zero.
	4	Check for separation only when solution has converged, for analytical surfaces only.
ICSEP (2,9)	Flag to control separation. (Integer $\geq$ 0; Default = 0)	
	0	The node separates and an iteration occurs if the force on the node is greater than the separation force.
	1	If a node which was in contact at the end of the previous increment has a force greater than the separation force, the node does NOT separate in this increment, but separates at the beginning of the next increment.
	2	If a new node comes into contact during this increment, it is not allowed to separate during this increment, prevents chattering.
	3	Both 1 and 2 are in effect.
IBSEP (2,12)	Flag for separation based on stresses or forces. (Integer $\geq$ 0; Default = 0)	
	0	Separation based on forces.

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	1	Separation based on absolute stresses (force/area).
	2	Separation based on absolute stress (extrapolating integration point stresses).
	3	Relative nodal stress (force/area).
	4	Separation based on relative stress (extrapolating integration point stresses).
	Only option 2 and 4 can be used with mid-side node elements where the mid-side nodes contact (LINQUAD=-1).	
ISHELL (2,10)	Parameter governing normal direction and thickness contribution of shells. (Integer $\geq 0$ ; Default = 0)	
	0	Check node contact with top and bottom surface.
	1	Nodes only come into contact with bottom layer.
	2	Nodes only come into contact with bottom layer and shell thickness is ignored.
	-1	Nodes only come into contact with top layer.
	-2	Nodes only come into contact with top layer and shell thickness is ignored.
IPRINT (2,11)	Flag to reduce print out of surface definition. (Integer $\geq 0$ ; Default = 0)	
	0	Full print out.
	1	Reduced print out.
RVCNST (3,1)	Relative sliding velocity between bodies below which sticking is simulated. If FTYPE=5, then the value of RVCNST is the stick-slip transition region.	
FTYPE (2,4)	Friction type. (Integer)	
	0	No friction. (Default)
	1	Shear friction.
	2	Coulomb Friction.
	3	Shear friction for rolling.
	4	Coulomb friction for rolling.
	5	Stick-slip Coulomb friction.
	6	Bilinear Coulomb friction. (Default if friction is present and FTYPE is not entered.)
	7	Bilinear Shear friction.
FKIND (2,5)	Friction kind. (Integer 0 or 1)	

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
	0	Friction based on nodal stress.
	1	Default if friction is present and FKIND is not entered. Friction based on nodal force.
BEAMB (2,13)	Beam-Beam contact flag. (Integer 0 or 1)	
	0	No beam-beam contact. (Default)
	1	Activate beam-beam contact options.
FSSMULT (3,7)	Stick-slip friction coefficient multiplier. Applicable only to stick-slip friction. (The friction coefficient is multiplied by this value for the sticking condition.) (Real $\geq 0$ ; Required; Default = 1.05)	
FSSTOL (3,8)	Stick-slip friction force tolerance. Applicable only to stick-slip friction. (Real; Default = 0.05)	
LINQUAD (2,14)	Higher order element contact flag (Integer; Default = 1).	
	1	The outer boundary of a contact body is described by the corner nodes only and mid-side nodes can't come into contact.
	-1	The outer boundary is described by a quadratic field and both corner and mid-side nodes are considered in contact. If this flag is set to -1 and IBSEP is blank, IBSEP will be re-set to 2. This option is only available with Marc 2003 and subsequent releases.
INITCON (2,16)	If INITCON is set, tying relations (MPC's) for surfaces initially in contact will be saved. This option maybe used to model dissimilar meshes. See CONTINUE=101+ on the SOL 600 entry to use these items in the same MSC Nastran execution. The following options are available:	
	1	MPC's in Marc format are saved for initial contact (if any) in file jid.marc.t01.
	3	MPC's for each increment are saved in Marc format in file jid.marc.conmpc_xxxx where xxxx is the increment number.
	4	MPC's for each increment are saved in Nastran MPC format in file jidd.marc.conmpc_xxxx where xxxx is the increment number.
	When initcon=1, the job will stop at the end of increment zero. When initcon=3 initcon=4, the job will run to completion and the information in jid.marc.conmpc_0000 for increment zero will usually not be useful, but the information for increments one and above will contain the proper contact tying relations or mpc's.	
NVSURF	Rigid contact surface ID (BCBODY ID) for which the next 4 approach velocity values apply. Leave this entry and the next 4 out if not applicable. This entry and the next 4 may be repeated as many times as necessary to define all rigid contact surfaces with approach velocity values. (Integer; Default = 0)	

Name	Description, Type and Value (Default is 0 for Integer, 0.0 for Real Unless Otherwise Indicated)	
VELAPP1 (6,1)	Approach velocity in direction 1 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPP2 (6,2)	Approach velocity in direction 2 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPP3 (6,3)	Approach velocity in direction 3 of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,11 (Real; Default = 0.0)	
VELAPAN (6,3)	Approach angular velocity about local axis through center of rotation of rigid contact surface - used only in increment zero. Requires PARAM,MRCONVER,1 (Real; Default = 0.0)	
NODSEP	If NODSEP is positive, ignore separation of any node that has touched and separated NODSEP times (corresponds to feature, 31xx where xx=nodsep). If NODSEP is negative, NODSEP is the number of separations of a particular node allowed. If more separations of that node actually occur, the node is assumed to be separated (corresponds to feature, -31xx where xx=nodsep). (Integer between 0 and 99; Default if not entered is 2, corresponds to feature,3102)	
METHOD	Flag to select Contact Methods (Character)	
	NODES URF	Regular 3D contact (Default: node to surface contact)
	SEGSMA LL	Segment to segment contact with small sliding
AUGMENT	Augmentation method used in a segment to segment contact analysis. (Integer)	
	0	No augmentation (Default)
	1	Augmentation based on a constant Lagrange multiplier field for linear elements and on a bilinear Lagrange multiplier field for quadratic elements.
	2	Augmentation based on a constant Lagrange multiplier field.
	3	Augmentation based on a bilinear Lagrange multiplier field.
PENALT	Augmented Lagrange penalty factor used by the segment to segment contact algorithm only. (Real > 0; set BCBODY for Default)	
AUGDIST	Penetration distance beyond which an augmentation will be applied. Used by the segment to segment contact algorithm only. (Real > 0.0; set BCBODY for Default)	

Remarks:

1. (i,j) refers to data block i and field j of the CONTACT model definition option in Marc.
2. Field 2 of the primary line should be left blank or a value of zero should be entered.
3. Only one BCPARA entry should be made. If multiple entries are made, the last will be used.

4. For FTYPE, SOLs 400 and 600 differ as follows. For SOL 400, if friction is entered but FTYPE is blank, friction is ignored. For SOL 600, if friction is entered but FTYPE is blank COULOMB friction is used. (FTYPE is reset internally to 2). In addition SOL 400 can only use friction types 0, 6 and 7.

**BCPATCH** Defines a Rigid Contact Body Made up of Quadrilateral Patches in SOLs 101 and 400

Defines a rigid contact body made up of quadrilateral patches used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCPATCH	RBID								
	IDP	G1	G2	G3	G4				
	IDP	G1	G2	G3	G4				
	-etc.-								

Example:

BCPATCH	12								
	1	101	102	103	104				

Describer	Meaning
RBID	Unique identification number referenced by a BCRGSRF or BCBODY1 entry (Integer > 0). See Remark 1.
IDP	ID of the patch (Integer number 1 through highest value).
G1, G2, G3, G4	Grid numbers for each of the 4 nodes of the patch.

Remarks:

1. If BCRGSRF entry does not exist, BCPATCH entry will be referenced by the BCBODY1 entry directly.



## BCPFLG Defines a beam section segment flag

Defines a beam section branch and segment for segment to segment beam contact used in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued Contact.

Format:

1	2	3	4	5	6	7	8	9	10
BCPFLG	PID	IBRNCH1	IOUTIN 1	IBRNCH2	IOUTIN2	IBRNCHi	IOUTINi		

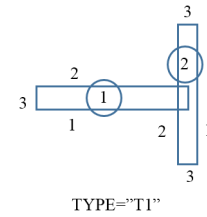
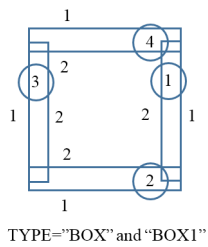
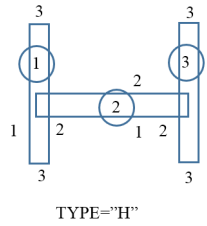
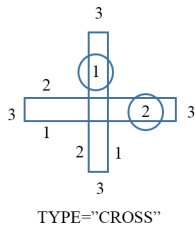
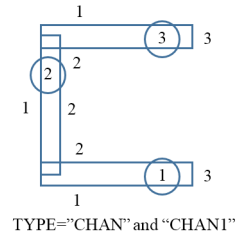
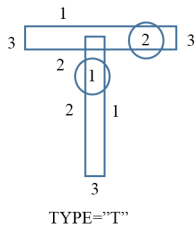
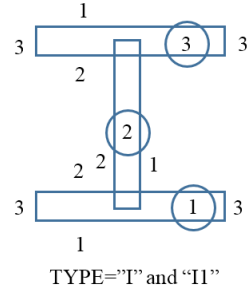
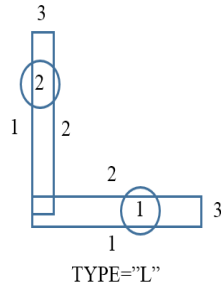
Example:

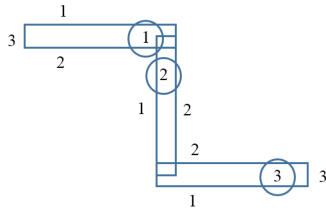
BCPFLG	2	1	2	3	13				
--------	---	---	---	---	----	--	--	--	--

Describer	Meaning
PID	Beam property/section identification number of PBEAML/PBARL. (Integer > 0; Required)
IBRNCHi	Branch selection flag for segment generation. Only required for thin wall sections. See Remark 1, 2, 3 and 4. (Integer; default=0).  0 select all branches for thin wall sections
IOUTINi	Beam segment location. See Remark 4, 5, 6 and 7. (Integer; default=0; up to three unique integers may be placed in the field with no embedded blanks.)  0 - all segments 1 - location 1 of thin wall section or only outer of solid or tube sections 2 - location 2 of thin wall section or only inner of tube section 3 - location 3 of thin wall section (thickness direction of branch) 4 - segment not including thickness

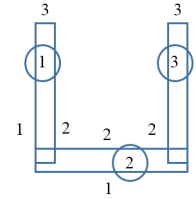
Remarks:

- If the card is not defined but the beam property is included in segment to segment contact, all segments will be generated and used. This card is only useful when a user needs to define the specific beam segments for segment to segment contact. It will help the performance of calculation because only selected segments will be considered in the contact calculation.
- IBRNCHi fields are only required for thin wall beam sections. Solid beam sections such as rectangular, rod, etc or tube sections require only IOUTIN1.
- Each branch of thin wall beam section has four segments.
- Segment locations of thin wall beam sections are listed below. TYPE is defined by PBEAML/PBARL. The number circled is the branch number of each shape and the number without circle is the segment number which is used in IOUTINi. Segment 1 and 2 of each branch are the segments in the branch direction and segment 3's are the segment in the thickness direction of the branch.

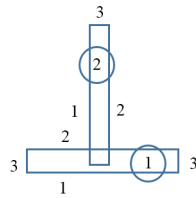




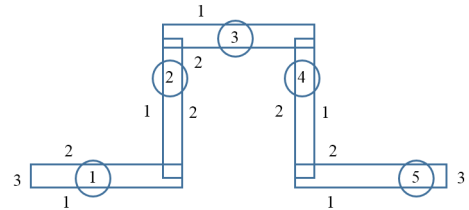
TYPE="Z"



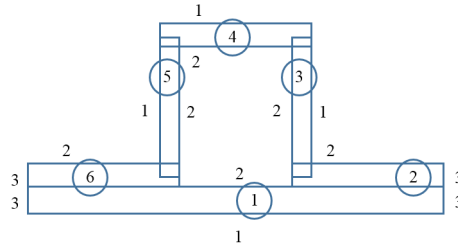
TYPE="CHAN2"



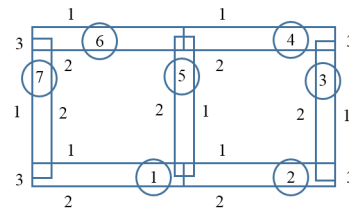
TYPE="T2"



TYPE="HAT"



TYPE="HAT1"



TYPE="DBOX"

5. If there is not thickness, one segment will be generated in the branch.
6. "3" cannot be used alone and "4" must be used only with "1" or "2". IOUTIN<sub>i</sub> must be one of "0", "1", "2", "12", "13", "14", "23", "24" and "123".
7. "14" and "24" generate the identical segment since the thickness is ignored but segment normal direction will be different.

## BCPROP

## 3D Contact Region by Element Properties

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					
--------	---	-----	-----	-----	--	--	--	--	--

Alternate Format:

BCPROP	ID	IP1	THRU	IP2	IP3	IP4	IP5	IP6	
	IP7	IP8	IP9	etc.					

Alternate Example:

BCPROP	1	101	THRU	102	105	THRU	109	110	
--------	---	-----	------	-----	-----	------	-----	-----	--

Describer	Meaning
ID	Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0)
IPi	Property ID. A minimum of one entry is required. (Integer; no Default)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU. The THRU keyword may appear in fields 4 thru 8.
7. BCBOX and BCMATL are not available for SOL 101 or SOL 400

## BCPROP

## 3D Contact Region by Element Properties in SOL 600

Defines a 3D contact region by element properties. All elements with the specified properties define a contact body used in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
BCPROP	ID	IP1	IP2	IP3	IP4	IP5	IP6	IP7	
	IP8	IP9	etc.						

Example:

BCPROP	1	101	201	301					
--------	---	-----	-----	-----	--	--	--	--	--

Alternate Format:

BCPROP	ID	IP1	THRU	IP2					
--------	----	-----	------	-----	--	--	--	--	--

Example for Alternate Format:

BCPROP	25	101	THRU	102					
--------	----	-----	------	-----	--	--	--	--	--

Describer	Meaning
ID	Identification number of a deformable surface corresponding to a BSID value on the BCBODY entry or if the Case Control BCONTACT = BCPROP is specified. All elements corresponding to the property IDs specified that may potentially come into contact. Do not specify mixed property types (use all shell, all solid or all beam properties only). See Remark 1. (Integer > 0)
IPi	Property ID. A minimum of one entry is required. (Integer; no Default)

Remarks:

1. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
2. The deformable surface may alternately be defined using BSURF, BCBOX, or BCMATL entries.
3. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.
4. All elements corresponding to the IDs entered will be used to define the deformable surface.
5. As many continuation lines as necessary may be used to define all property IDs associated with a particular deformable body.
6. The alternate format is triggered if field 4 contains THRU.

**BCRIGID**

Defines a Rigid Contact Body in SOLs 101 and 400

Defines a rigid contact body used in SOLs 101 and 400 only.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCRIGID	BCRIGID	CGID	CONTROL						
	NLOAD	ANGVEL	DCOS1	DCOS2	DCOS3	VELRB1	VELRB2	VELRB3	
	“APPROV”	A	N1	N2	N3	V1	V2	V3	
	“GROW”	GF1	GF2	GF3	TAB-GF1	TAB-GF2	TAB-GF3		

Example:

BCRIGID	1001	1	1						
	0	0.	0.	0.	1.	1.	1.	1.	

**Describer**      **Meaning**

**BCRIGID**      Unique identification number referenced by a BCBODY1 entry. (Integer > 0)

**CGID**      Grid point identification number defining the initial position of the reference point of the rigid body or the point where a concentrated force or moment is applied. (Integer > 0) See Remark 1.

**CONTROL**      Indicates the type of control for the body.  
Integer:  
= -1 for position control. The coordinates of the final position of GRID Point defined in CGID is given in VELRB<sub>i</sub> in the following line.  
  
= 0 for velocity control. (default)  
= positive number for “load control”. The positive number is the grid number defined in CGID at which translational forces or SPCD are applied. (Note: The rotation in this case is defined by NLOAD in the following line.)

**NLOAD**      Enter a positive number if “load controlled” and rotations are allowed. (Integer) The positive number is the grid number where the moments or rotations are applied. The rotations are specified using SPCD at grid ID NLOAD and can be specified using dof's 1-3 (for translation along x, y, z respectively), or by dof's 4-6 (for rotation about x, y, z respectively). Note: This rotation takes the position of the grid point defined in CGID field as the center of rotation.

**ANGVEL**      Angular velocity or angular position about local axis through center of rotation. If the value is an integer it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default = 0.0)

DCOSi	Components of direction cosine of local axis if ANGVEL is nonzero. If the value is an integer, it represents the ID of a TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. (Real or Integer; Default=0.0) In 2D contact only DCOS3 is used and the Default is 1.0.
VELRBi	Tranlational velocity or final position (depending on the value of CONTROL) of rigid body at the grid point defined in CGID filed. For velocity control only, if the value is an integer, it represents the ID of TABLED1, TABLED2 or TABL3D, i.e., a time-dependent or multi-dimensional table; however, no log scales, only linear scales. Only VELRB1 and VELRB2 are used in 2D contact. (Real or Integer; Default = 0.0)
“APPROV”	The entries of this continuation line are for approaching velocity to establish initial contact.
A	Angular velocity about local axis through center of rotation. (Real, Default = 0.0)
Ni	Components of direction cosines of local axis of the angular velocity. The N1, N2, N3 define the axis through the point defined in the CGID field. Only N1 and N2 are used in 2D contact. (Real, Default = 0.0)
Vi	V1, V2 and V3 define the three components of the approaching velocity. Only V1 and V2 are used in 2D contact. (Real; Default = 0.0)
“GROW”	The entries of this continuation line are for rigid body growth. If tables are used for growth, they should either be TABLED1, TABLED2 (growth vs time) or TABL3D (growth vs one or more variables).
GFi	Components of growth factor of rigid body in the coordinate system of the CGID field. (Real, Default = 1.0)
TAB-GFi	Table IDs for growth factor of rigid body in the coordinate system of the CGID field. (Integer > 0 or blank, Default is blank)

#### Remarks:

1. The grid CGID is the reference grid for the rigid body motion. Loads and enforced motion must be defined in the global coordinate system of CGID. If CGID is not specified, basic coordinate will be used.
2. All continuation lines may be omitted if not required.

**BCRGSRF** Rigid Contact Surface List in SOLs 101 and 400

Defines a list of rigid contact surfaces used in SOLs 101 and 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCRGSRF	BSID	RBID1	RBID2	RBID3	RBID4	RBID5	RBID6	RBID7	
	RBID8	-etc.-							

Example:

BCRGSRF	1008	35	2						
---------	------	----	---	--	--	--	--	--	--

**Describer**    **Meaning**

BSID            Unique identification number referenced by a BCBODY1 entry. (Integer > 0) See Remark 1.

RBIDi          Identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry. (Integer > 0)

Remarks:

1. If BCRGSRF entry does not exist, BCBODY1 entry can refer to one of the identification number of BCPATCH, BCBZIER, BCNURB2 or BCNURBS entry directly.



## BCSCAP

Defines a cap and number of segments

Defines whether a cap is added to a beam section and/or defines the number of segments used to model rod and tube sections for segment to segment beam contact in SOL 101 and 400 for general contact or in SOLs 101, 103, 105, 107 - 112, 200 and 400 for the permanently glued contact

Format:

1	2	3	4	5	6	7	8	9	10
BCSCAP	EID	IESCAP	NSEG						

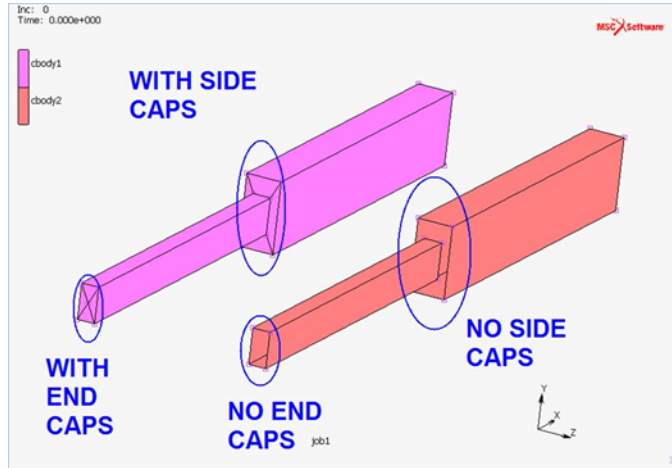
Example:

1	2	3	4	5	6	7	8	9	10
BCSCAP	2	1	20						

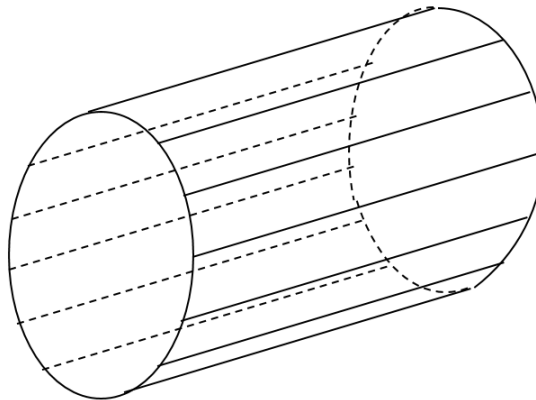
Describer	Meaning
EID	Beam element identification number. (Integer > 0; Required)
IESCAP	End and side cap flag (Integer ≥ 0; default=0).  0 no end or side cap 1 end cap only 2 side cap only 3 averaged side cap only 4 end cap and side cap 5 end cap and averaged side cap
NSEG	Number of segments for circular sections TYPE=ROD, TUBE or TUBE2 beam sections defined on PBARL or PBEAML. See remarks 2. (Integer ≥ 3; default=32)

Remarks:

1. If the card is not defined but the beam is included in segment to segment contact, all caps will be ignored. This card is only useful when a user needs to define the cap of beams for segment to segment contact.



2. The segments for circular cross sections will be generated as the figure below.



## BCSEG

## Contact Segment Defined Using Grids

Grids which are part of an element to be used in contact analyses in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
BCSEG	ID	IBODY	G1	G2	G3	G4			

Example:

1	2	3	4	5	6	7	8	9	10
BCSEG	100	1005	11	12	13	14			

Describer	Meaning
ID	Unique identification number for this BCSEG entry. (Integer > 0; Required)
IBODY	Identification number of a surface that is called out on the 5 <sup>th</sup> field of a BCBODY entry. (Integer > 0; Required)
G1,G2,G3,G4	GRID point identification numbers of an element on this surface. For quad plates and quad surfaces of solids, enter four grid id's. For triangular plates or triangular faces of solids, leave G4 blank. (Integer > 0; Required)

Remarks:

1. This entry is used as shown in the example below:  
 BCBODY, 201,,,1005  
 BCSEG,1,1005,11,12,13,14  
 BCSEG,2,1005,21,22,23,24  
 BCSEG,3,1005,31,32,33,34  
 (In the above 11-14, 21-24 and 31-34 are GRID ID's)

## BCSURF

## Contact entry by element faces

Define a contact region by element faces for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400. This has three forms based the FORM (the 6th) field. "FACE" (default) is to define contact face by element face IDs, "GRID" is to define the same by grid point ids, and "RIGID" is to define a rigid face.

### Format 1: FORM="FACE" (default)

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		FORM	INCTHK	EDGCNT		
	ESID1	FACEID1	IDTYPE1		ESID2	FACEID2	IDTYPE2		
	ESID3	FACEID3	IDTYPE3		...				
	...								

### Format 2: FORM="GRID"

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		"GRID"	INCTHK	EDGCNT		
	EID1	G11	G12	G13	EID2	G21	G22	G23	
	EID3	G31	G32	G33	...				
	...								

### Format 3: FORM="RIGID"

1	2	3	4	5	6	7	8	9	10
BCSURF	BID	BPID	DIM		"RIGID"				
	BSID	BCRGID							

Example:

#### FACE form (default)

BCSURF	2	3	3D			NO			
	12	S2			4	S4	SET		

#### GRID form

BCSURF	3	3	3D		GRID	YES			
	11	1	3	6	16	31	14	15	

#### RIGID form

BCSURF	4		3D		RIGID				
	101	201							

Describer	Meaning
BID	Unique contact face identification number referenced by BCONECT (Integer > 0; Required). See remark 1.
BPID	Parameter identification number of a BCBDPRP entry. (Integer > 0 or blank).
DIM	Dimension of body. (Character; Default= "3D"), "3D" or "2D".
FORM	Select a type of the entry. (Character; Default = "FACE"). FACE define the contact surface by element face IDs GRID define the contact surface by grid points on an element face RIGID define the contact surface by rigid surface
INCTHK	Option to include shell thickness offset. (Character; Default = "YES"), "YES" or "NO".
EDGCNT	Option for edge contact. (Integer, default=1), below three values can be specified. 1: only the beam/bar edges are included in the contact description (Default). 10: only the free and hard shell edges are included in the contact description. 11: both the beam/bar edges and the free and hard shell edges are included in the contact description.
ESIDi	Element id or set id. (Integer, no default). If IDTYPEi is "SET", it is a set id defined by a SET3, and the SET3 must be element type. If IDTYPEi is "ELEM" (default), it is an element id.
FACEIDi	Element face ID. (Character, blank can be used to express BOTH for shell elements and the whole bodies of beam or bar elements). See remarks 6., 7., 8. and 9.
IDTYPEi	Type of ESIDi. (Character, "ELEM"). ELEM ESIDi is an element id. SET ESIDi is a SET3 id.
EIDi	Element id. (Integer > 0)
GIDij	Three corner grid point ids of the element face. (Integer, no default). All the three fields of for an EIDi are blanks or zeros mean BOTH for shell elements and the whole bodies of bar or beam elements. See remark 2.
BSID	Identification number of a BCRGSRF, BCPATCH,BCBZIER, BCNURB2, or BCNURBS. (Integer > 0).
BCRGID	Identification number of a BCRIGID entry. (Integer > 0).

#### Remarks:

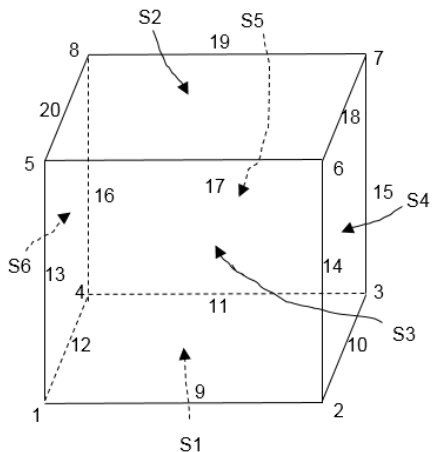
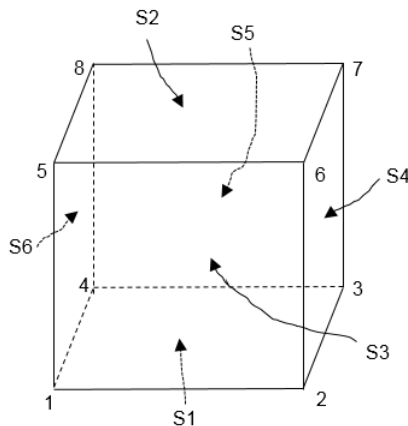
1. BID must be unique among all BCSURF, BCBODY1 and BCGRID entries.
2. BCSURF entries are able to coexist with BCBODY1 entries. A BCSURF can be used with a BCGRID, another BCSURF or a BCBODY1 entry to construct a contact pair.

3. BCSURF cannot be specified in the continuations of a BCONECT.
4. If a BCSURF is referenced by a BCONECT, COPTS and COPTM in the BCONPRG referenced by the BCONECT will be used, and FACEIDs and INCTHK fields of BCSURF are ignored.
5. For GRID form, three corner grid point IDs are required to specify a 3D element face, two corner grid point ids are required to specify a 2D element edge. For shell faces, the sequence of G1,G2,G3 of the element definition means TOP(top) and the reverse order means BTM(bottom), blanks or zeros means BOTH(both).
6. FACEIDi varies depending on element type and DIM. Refer to remark 9 for the list of FACEIDs.
7. For shell elements, TOP, BTM and BOTH are to define the contact face on top, bottom and both sides of the element. A blank is the same as BOTH. It is possible to have TOP and BTM in the same BCSURF entry but not allowed to combine BTM with BOTH or TOP with BOTH.
8. For beam elements, ENDA and ENDB express beam or bar enda and endb points respectively. A blank means the whole beam or bar body.
9. List of face ID.

Refer to following pages

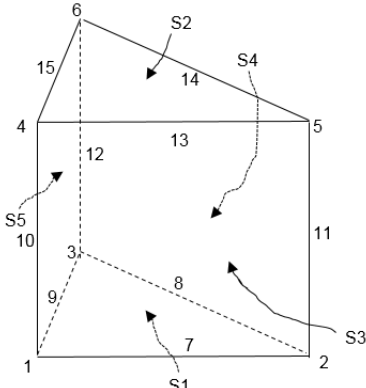
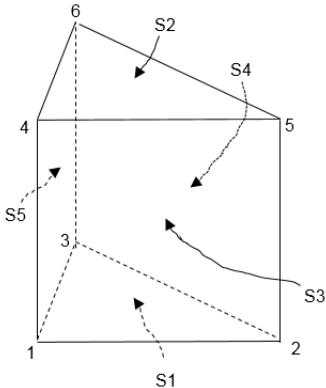
CHEXA

	Linear Element	Quadratic Element
S1	4-3-2-1	4-3-2-1-11-10-9-12
S2	5-6-7-8	5-6-7-8-17-18-19-20
S3	1-2-6-5	1-2-6-5-9-14-17-13
S4	2-3-7-6	2-3-7-6-10-15-18-14
S5	3-4-8-7	3-4-8-7-11-16-19-15
S6	4-1-5-8	4-1-5-8-12-13-20-16



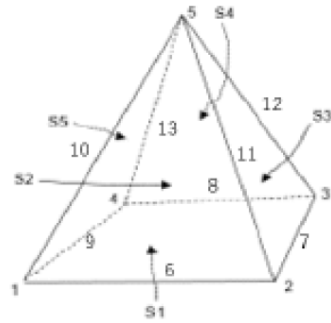
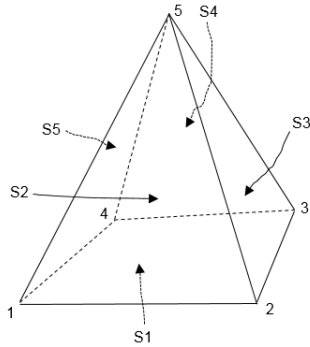
CPENTA

	Linear Element	Quadratic Element
S1	3-2-1	3-2-1-8-7-9
S2	4-5-6	4-5-6-13-14-15
S3	1-2-5-4	1-2-5-4-7-11-13-10
S4	2-3-6-5	2-3-6-5-8-12-14-11
S5	3-1-4-6	3-1-4-6-9-10-15-12



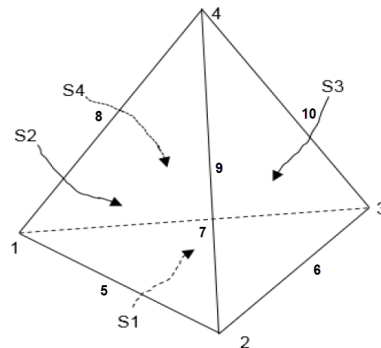
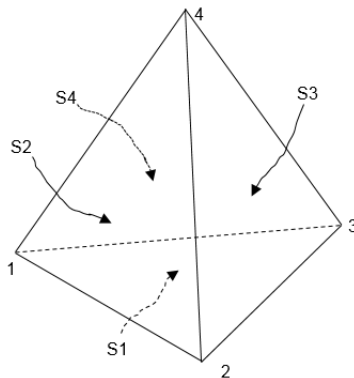
CPYRAM

	Linear Element	Quadratic Element
S1	4-3-2-1	4-3-2-1-8-7-6-9
S2	1-2-5	1-2-5-6-11-10
S3	2-3-5	2-3-5-7-12-11
S4	3-4-5	3-4-5-8-13-12
S5	4-1-5	4-1-5-9-10-13



CTETRA

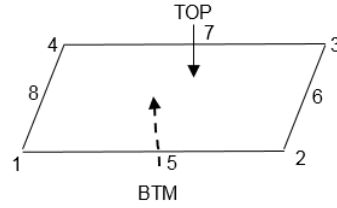
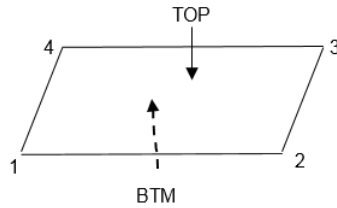
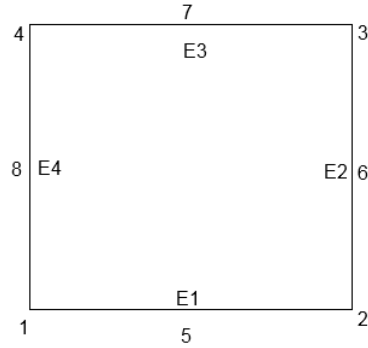
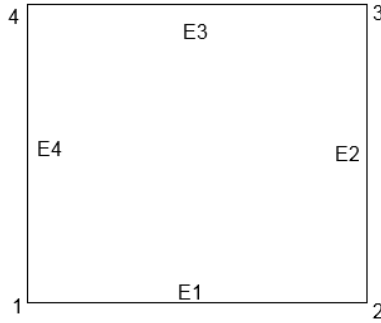
	Linear Element	Quadratic Element
S1	3-2-1	3-2-1-6-5-7
S2	1-2-4	1-2-4-5-9-8
S3	2-3-4	2-3-4-6-10-9
S4	3-1-4	3-1-4-7-8-10





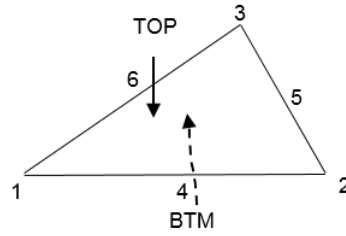
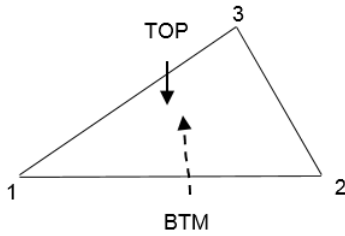
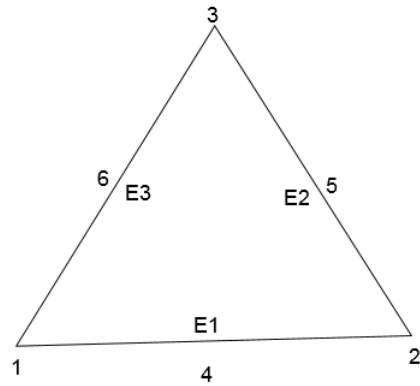
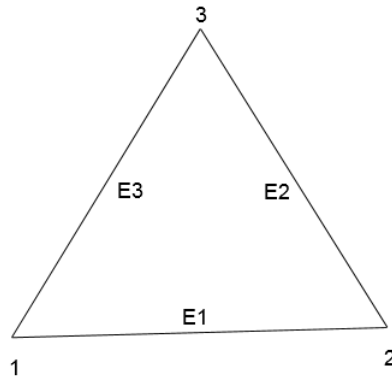
CQUAD4/CQUAD/CQUAD8/CQUADR when DIM is 3D

	Linear Element	Quadratic Element
E1	1- 2	1-2-5
E2	2-3	2-3-6
E3	3-4	3-4-7
E4	4-1	4-1-8
TOP	1-2-3-4	1-2-3-4-5-6-7-8
BTM	4-3-2-1	4-3-2-1-7-6-5-8



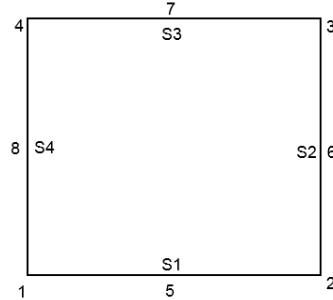
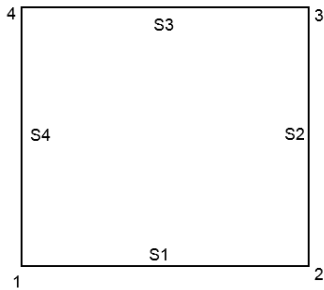
CTRIA3/ CTRIA6/CTRIAR when DIM is 3D

	Linear Element	Quadratic Element
E1	1- 2	1-2-4
E2	2-3	2-3-5
E3	3-1	3-1-6
TOP	-2-3	1-2-3-4-5-6
BTM	3-2-1	3-2-1-5-4-6



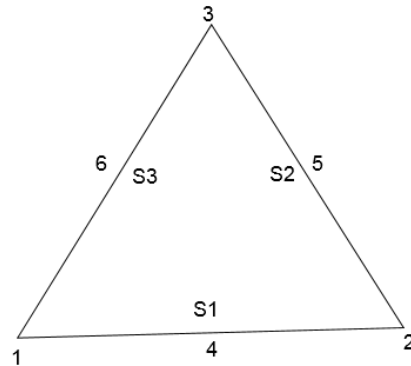
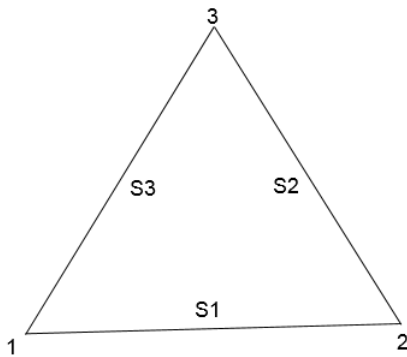
CQUAD4, CQUAD8 and CQUADX when DIM is 2D

	Linear Element	Quadratic Element
S1	1- 2	1-2-5
S2	2-3	2-3-6
S3	3-4	3-4-7
S4	4-1	4-1-8



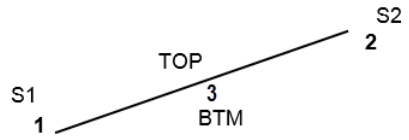
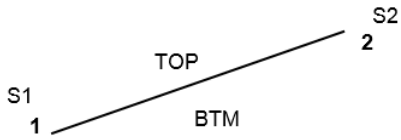
CTRIA3, CTRIA6, CTRIA8, CTRIA9 when DIM is 2D

	Linear Element	Quadratic Element
S1	1- 2	1-2-4
S2	2-3	2-3-5
S3	3-1	3-1-6



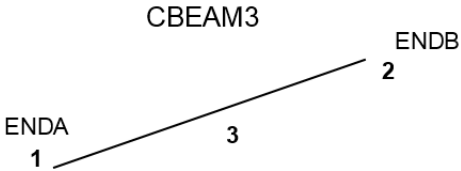
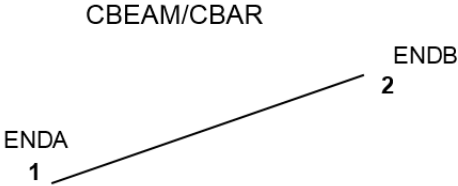
CAXISYM

	Linear Element	Quadratic Element
S1	1	1
S2	2	2
TOP	1-2	1-2-3
BTM	2-1	2-1-3



CBAR, CBEAM and CBEAM3

	Linear Element	Quadratic Element
ENDA	1	1
ENDB	2	2
(BLANK)	1-2	1-2-3



**BCTABLE**

Defines a Contact Table for General Contact

Defines a contact table used in SOL 101, 400 and 700. SOL 200 (without calling SOL 400) can only support permanent glue. When SOL 200 calls SOL 400 (or say SOL 400 optimization), it can support all contact types. The BCTABL1 option is the preferred method to define a contact table.

Format: (SOLs 101 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSCND	IDPRIM	NGROUP	COPTS	COPTM			
	"SECNDRY"	IDSEC1	ERROR	FNTOL	FRIC	CINTERF	IGLUE		
		ISEARCH	ICOORD	JGLUE		DQNEAR			
		"FBSH"	FRLIM	BIAS	SLIDE	HARDS	COPTS1	COPTM1	
		"BKGL"	BGSN	BGST	BGM	BGN			
		"SEGS"	PENALT	AUGDIST	TPENALT	STKSLP			
		"HHHB"	HCT	HCV	HNC	BNC	EMISS	HBL	
			HNL	BNL	HGLUE				
	"PRIMARY"	IDPRIM1	IDPRIM2	IDPRIM3	IDPRIM4	IDPRIM5	IDPRIM6	IDPRIM7	
		IDPRIM8	IDPRIM9	...					

Format: (SOL 700 only)

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID	IDSCND	IDPRIM	NGROUP					
	"SECNDRY"	IDSEC1			FRIC		IGLUE		
				JGLUE					
		FK	EXP	METHOD	ADAPT	THICK	THICKOF	PENV	
		FACT	TSTART	TEND	MAXPAR				
			IADJ						
				IGNORE					
								AUTO	



1	2	3	4	5	6	7	8	9	10
		SIDE	WEIGHT	MONDIS	MONDIS V	INITMON	DAMPIN G		
	"PRIMARY"	IDPRIM1	IDPRIM2	IDPRIM3	IDPRIM4	IDPRIM5	IDPRIM6	IDPRIM7	
		IDPRIM8	IDPRIM9	...					

**Examples:**

BCTABLE	2			3					
	SECNDRY	10			0.2				
	PRIMARY	20	30						
	SECNDRY	20			0.3				
	PRIMARY	10							
	SECNDRY	30			0.2				
	PRIMARY	10							

BCTABLE	0	1	2	0					

Describer	Meaning
ID	Identification number referenced by a BCONTACT Case Control command. See Remark 5. (Integer; Required)
IDSCND	Identification number of a BCBODY entry defining the touching body. (Integer > 0 or blank)
IDPRIM	Identification number of a BCBODY Bulk Data entry defining the touched body. (Integer > 0 or blank)
NGROUP	Flag to indicate that the continuation entries "SECNDRY" and "PRIMARY" are entered or not. Zero means no continuation entries are entered. Any positive integer means one or more sets of secondary/primary entries are entered. (Integer; Default = 1).
COPTS, COPTM, COPTS1, COPTM1	Flag to indicate how secondary and primary surfaces may contact. See Remark 9. (Integer; Default = 0) COPTS and COPTM apply to all secondary and primary surfaces on this BCTABLE. To set individual secondary/primary combination differently, use COPTS1 and/or COPTM1. Ignored by SOL 700.
"SECNDRY"	Indicates that this line defines the touching body and its parameters.
IDSEC1	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0) For SOL 700, leaving IDSEC1 blank will result in contact for all elements in the model. In this case, you are allowed to use ADAPT=YES.

Describer	Meaning
ERROR	Distance below which a node is considered touching a body. Default = blank automatic calculation. See the Bulk Data entry <a href="#">BCPARA</a> for more details. Ignored by SOL 700. (Real)
FNTOL	Separation force, stress, or fraction above which a node separates from a body. FNTOL is closely related to IBSEP. It's default value is dependent on the IBSEP value. See remark <a href="#">10</a> . Ignored by SOL 700. (Real)
FRIC	Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multidimensional table. SOL700 does not support the table definition. (Real > 0.0 or Integer > 0; Default = 0.0 for SOL400, Real >0.0; Default=0.0 for SOL700)
CINTERF	Interference closure amount, normal to the contact surface. Default = 0. For CINTERF > 0, overlap between bodies. For CINTERF < 0., gap between bodies. Ignored by SOL 700. (Real)
IGLUE	Flag to activate glue option (Integer ≠ 0). Default is 0, no glue option. For SOL 700, IGLUE=1 is only acceptable. JGLUE option controls a glue contact type in details. Without assigning IGLUE and JGLUE options, general contact methods (METHOD=TIEDxxx) can assign glue contact types as well. (See Remarks <a href="#">8</a> through <a href="#">16</a> .)
	±1 Activates the glue option. In the glue option, all degrees-of-freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. The node will be projected onto the contact body.
	±2 Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.
	±3 Insures full moment carrying glue when shells contact. The node will be projected onto the contacted body.
	±4 Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.
ISEARCH	Option for contact searching order, from Secondary to Primary or from Primary to Secondary, for deformable contact bodies. Ignored by SOL 700. ISEARCH is not necessary in segment-to-segment contact. (Integer; Default = 0)



Describer	Meaning
	<p>0 (Double orders search) ) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19.</p> <p>1 (Single order search) the searching order is from Secondary to Primary</p> <p>2 (Single order search) let the program decide which search order. See Remark 19.</p> <p>Note that ISTYP in BCBDRP or BCBODY is supported with ISEARCH=0 only.</p>
ICOORD	<p>Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. Ignored by SOL 700. (Integer; Default = 0)</p>
JGLUE	<p>This option is only relevant if the general glue option is invoked (JGLUE &gt; 0 and =1). Enter 0 if a node should not separate (default). Enter 1 to invoke the standard separation behavior based on the maximum residual force. For SOLs 101 and 400, enter 2 to activate breaking glue. (Integer; Default = 0)</p> <p>For SOL 700 the following options are available:</p> <p>0 Secondary nodes in contact and which come into contact will permanently stick. Tangential motion is inhibited. This option is only available with METHOD=SS1WAY or SS2WAY and AUTO=YES.</p>
DQNEAR	<p>Distance below which near thermal contact behavior occurs. Used in heat transfer analysis only. Ignored by SOL 700. (Real; Default = 0; which means near contact does not occur)</p>
"FBSH"	<p>Enter character string FBSH if the line with items FRLIM, BIAS, SLIDE, HARDS is required to change the defaults of any of these values. (SOLs 101 and 400)</p>
FRLIM	<p>Friction stress limit. This entry is only used for friction type 6 (Coulomb friction using the bilinear model). If the shear stress due to friction reaches this limit value, then the applied friction force will be reduced so that the maximum friction stress is given by <math>\min(\mu\sigma_n, \sigma_{limit})</math>, with <math>\mu</math> the friction coefficient and <math>\sigma_n</math> the contact normal stress. (Real; Default = 1.0E20)</p>
BIAS	<p>Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. (0.0&lt;=Real&lt;=1.0)</p>

Describer	Meaning
SLIDE	Delayed slide off distance ratio. This entry should not be made unless ICOORD $\geq$ 2 (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.2)
HARDS	Hard-soft ratio. This entry is only used if double-sided contact with automatic constraint optimization is used, (ISTYP=2 on the BCBODY entry). The hard-soft ratio can be used by the program if there is a significant difference in the (average) stiffness of the contact bodies (expressed by the trace of the initial stress-strain law). If the ratio of the stiffnesses is larger then the hard-soft ratio, the nodes of the softest body are the preferred secondary nodes. (Real; Default = 2.0)
“BKGL”	Enter the character string BKGL if the line with items BGST, BGSN, BGM, BGN is required. (SOLs 101 and 400)
BGSN	Maximum normal stress for breaking glue. See Remark 10. (Real; Default = 0.0)
BGST	Maximum tangential stress for breaking glue. See Remark 10. (Real; Default = 0.0)
BGM	Exponent for the tangential stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)
BGN	Exponent for the normal stress term in deciding if a glue-contact will break. See Remark 10. (Real; Default = 2.0)
“SEGS”	Enter the character string SEGS if the line with items PENALT and AUGDIST, TPENALT and STKSPL is required for the segment-to-segment contact.
PENALT	Augmented Lagrange penalty factor; used by the segment-to-segment contact algorithm only. (Real $\geq$ 0.0; see Remark 16. for default)
AUGDIST	Penetration distance beyond which an augmentation will be applied; used by the segment-to-segment contact algorithm only. (Real $\geq$ 0, see Remark 17. for default)
“HHHB”	Enter the character string HHHB if the line with items HCT, HCV, HNC, ..., up to HGLUE is required in a pure thermal analysis or in a coupled thermal/structural analysis. In a pure structural analysis they are ignored.
HCT	Contact heat transfer coefficient. If Real, the value entered is the contact heat transfer coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the contact heat transfer coefficient vs temperature or a TABL3D entry specifying the contact heat transfer coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0; In a thermal/mechanical coupled analysis a default value of 1.0E6 is used when the mechanical glue option is activated, i.w., when IGLUE > 0.)

Describer	Meaning
HCV	Convection coefficient for near field behavior. If Real, the value entered is the near field convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field convection coefficient vs temperature or a TABL3D entry specifying the near field convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
HNC	Natural convection coefficient for near field behavior. If Real, the value entered is the near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field natural convection coefficient vs temperature or a TABL3D entry specifying the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
BNC	Exponent associated with the natural convection coefficient for near field behavior. If Real, the value entered is the exponent associated with near field natural convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the exponent associated with the near field natural convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field natural convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0.)
EMISS	Emissivity for radiation to the environment or near thermal radiation. If Real, the value entered is the emissivity. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the emissivity vs temperature or a TABL3D entry specifying the emissivity vs temperature and possibly other variables. (Real or Integer; Default = 0.0)
HBL	Separation distance dependent thermal convection coefficient. If Real, the value entered is the convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the convection coefficient vs temperature or a TABL3D entry specifying the convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0).
HNL	Heat transfer coefficient for nonlinear convective heat flow for near field behavior.. If Real, the value entered is the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 or TABLEM2 entry specifying the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 0.0)
BNL	Exponent associated with the nonlinear convective heat flow for near field behavior. If Real, the value entered is the exponent associated with the near field nonlinear convection coefficient. If Integer, the value entered is the ID of a TABLEM1 OR TABLEM2 entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature or a TABL3D entry specifying the exponent associated with the near field nonlinear convection coefficient vs temperature and possibly other variables. (Real or Integer; Default = 1.0)

Describer	Meaning
HGLUE	Flag to activate the thermal glue option. When left blank or set to zero, thermal contact conditions will be treated by convective heat transfer between the bodies. When set to 1, the temperature fields of the bodies are tied as soon as they come in contact and there will be no convective heat transfer over the body interfaces. ( $0 \leq \text{Integer} \leq 1$ ; Default = 0)
FK (SOL 700 only)	Kinetic coefficient of friction. (Real $\geq 0.0$ ; Default = 0.0)
EXP (SOL 700 only)	Exponential decay coefficient. (Real $\geq 0.0$ ; Default = 0.0)
METHOD (SOL 700 only)	Character, Influences the contact type used. See Remark 19. Options are: FULL: Regular Contact (Default) AIRBAG: Single Surface Contact SS1WAY: Surface To Surface One Way SS2WAY: Surface To Surface Two Way RB1WAY: Rigid Body One Way To Rigid Body RB2WAY: Rigid Body Two Way To Rigid Body RNRB: Rigid Nodes To Rigid Body TIEDNS: Tied Nodes to Surface TIEDNSO: Tied Nodes to Surface with Offset RELIPS: Tied contact between grid points or rigid materials to ATB dummies. See Remark 23. BELT: Suited for modeling contact between a belt element and a rigid structure. Primary secondary contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the secondary nodes exactly on top of the primary face. The secondary node does not slide relative to the primary face when the friction coefficient (FS) is set to 1E20. BELT1: Identical to BELT algorithm, except that the secondary nodes are initially repositioned on top of the closest primary face. All secondary nodes initially penetrated or within a distance of INITMON from a primary face, are repositioned. DRAWBEAD: Suited for modeling a drawbead. STYPE Must be GRID. The list of secondary nodes must be ordered along the drawbead line. MTYPE Must be SURE. The restraining force per unit drawbead length is specified in the field DRWBEADF. It is V2 version contact. DRAWBDV4: Suited for modeling a drawbead. STYPE Must be GRID. The list of secondary nodes must be ordered along the drawbead line. MTYPE Must be SURE. The restraining force per unit drawbead length is specified in the field DRWBEADF. It is V4 version contact.

Describer	Meaning
ADAPT (SOL 700 only)	Character, influences the contact type used. Options are NO or YES. Default = NO When ADAPT=YES, the BCBODY entries IDPRIMi must be defined as: behav=DEFORM bsid references a BCPROP
THICK (SOL 700 only)	Shell thickness scale factor. (Real $\geq$ 0.0; Default = same as the value in DYPARAM,CONTACT,THICK)
THICKOF (SOL 700 only)	Artificial contact thickness offset. (Real $\geq$ 0.0; Default = 0.0)
PENV (SOL 700 only)	Overwrites the default maximum penetration distance factor. (Real $\geq$ 0.0; Default = 1.E20)
FACT (SOL 700 only)	Scale factor for the contact forces. (Real $\geq$ 0.0; Default = 0.1)
TSTART (SOL 700 only)	Time at which the contact is activated. (Real $\geq$ 0.0; Default = 0.0)
TEND (SOL 700 only)	Time at which the contact is deactivated. (Real $\geq$ 0.0; Default = 1.e20)
MAXPAR (SOL 700 only)	Maximum parametric coordinate in segment. (Real $\geq$ 0.0; Default = 1.025), search (values 1.025 and 1.20 recommended). Larger values can increase cost. If zero, the default is set to 1.025. This factor allows an increase in the size of the segments. May be useful at sharp corners.
IADJ (SOL 700 only)	Adjacent material treatment option for solid elements. (Integer $\geq$ 0; Default = 1)  0            Solid element faces are included only for free boundaries. 1            Solid element faces are included if they are on the boundary of the material subset. This option also allows the erosion within a body and the subsequent treatment of contact.
IGNORE (SOL 700 only)	Ignore initial penetrations. (Integer $\geq$ 1)  1            Allow initial penetrations to exist by tracking it 2            Move nodes to eliminate initial penetrations
AUTO (SOL 700 only)	(Character) Options are: YES        Automatic Contacts Activated (Default) NO         Non-Automatic Contact Activated. This option is not recommended when Distributed Memory Parallel is activated.

Describer	Meaning
SIDE (SOL 700 only)	<p>Defines which side will be the monitoring side of a primary face. The opposite side of the primary face will be the penetration side. See Remark 24. (Character; default=BOTH)</p> <p>BOTH: The side from which a secondary node approaches the primary face will become the monitoring side.</p> <p>TOP: The monitoring side will always be on the side of the primary face that its normal is pointing at.</p> <p>BOTTOM: The monitoring side is always on the opposite side of the primary face that its normal is pointing at.</p>
WEIGHT (SOL 700 only)	<p>The contact force is multiplied by a mass-weighting factor. The following options are available. See Remark 25. (Character; default=BOTH)</p> <p>BOTH: <math display="block">M_{scale} = \frac{M_{secndry}M_{primary}}{M_{secndry} + M_{primary}}</math></p> <p>SECNDRY: <math>M_{scale} = M_{secndry}</math></p> <p>PRIMARY: <math>M_{scale} = M_{primary}</math></p> <p>NONE: <math>M_{scale} = 1.0</math></p>
MONDIS (SOL 700 only)	<p>Defines the fixed part of the monitoring distance. When the normal distance of a secondary node to a primary face becomes smaller than the monitoring distance the secondary node will tag itself to the primary face. The side from which the secondary node is moving towards the primary face becomes the monitoring region. (Character; default=FACTOR)</p> <p>FACTOR: The monitoring distance is equal to a factor times a characteristic length of the primary faces. The factor is specified in MONDISV.</p> <p>DISTANCE: The monitoring distance is specified in MONDISV.</p>
MONDISV (SOL 700 only)	<p>Value of the monitoring distance or value of the FACTOR to calculate the monitoring distance. (Real; default=2.0)</p>
INITMON (SOL 700 only)	<p>Fixed part of the monitoring distance used during the initialization. If not specified, the value of MONDIS is used. (Real &gt; 0.0; default=MONDIS)</p>
DAMPING (SOL 700 only)	<p>Specifies if a high frequency damping is active or not. The damping force is based on the relative velocity of a secondary node with respect to a primary face. The damping is preferably turned on in all cases, except for RIGID-RIGID contact. In RIGID-RIGID contact it can result in a substantial loss of energy. VERSION V4 only. (Character; Default=YES)</p> <p>YES damping is activated</p> <p>N damping is not activated</p>

Describer	Meaning
“PRIMARY”	Indicates the start of the list of bodies touched by touching body IDSEC1.
IDPRIMi	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0)

#### Remarks:

1. BCTABLE defines surface contact.
2. If BCTABLE is not given, the default for contact analysis is assumed if BCONTACT=ALLBODY in Case Control Section, and, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABLE is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABLE is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. A short input to define two contact bodies exits if the user provides IDSCND and IDPRIM. Then it is assumed that there are only two contact bodies, NGROUP is ignored and continuation entries are not allowed. Default values are set for the parameters on the continuation entry.
4. If the user leaves IDSCND and IDPRIM blank, then NGROUP is normally required and continuation entries are usually expected for NGROUP SECNDRY/PRIMARY combinations. Exceptions are for SOL 700 self-contact may be designated using a secondary IDSEC1 of zero and no PRIMARY entry.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY's. BCTABLE, 0 is not used to define the contact relationship for any loadcases. ONLY BCTABLE, n(n>0), will dominate the contact analysis for each loadcase. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.
6. The line starting with “HHHB” is used for heat transfer or thermal contact analyses only. Also, see Remark 17.
7. It is not necessary to enter all continuation lines between SECNDRY and PRIMARY. For example, if an entry is required on the 4<sup>th</sup> SECNDRY line, the first 3 must be entered with some being blank and those after the 4<sup>th</sup> may be omitted. All continuation lines prior to the last needed must be entered.

8. Many secondary/primaries may be entered up to the number specified in Remark 5. A new secondary entry may not begin until the primaries from the previous entry are finished (as shown in the example). The number of primary surfaces for any given secondary surface is limited by the number specified in Remark 5., however most GUI's produce one secondary surface and one primary surface per secondary/primaries pairs.
9. COPTS, COPTM, COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$COP_{xxx}=A+10*B+1000*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA).

Note that C has no effect if segment-to-segment contact is used.

10. Breaking Glue provides glued contact to all GRID's at their very 1st contact. This kind of glued-contact will break if

$$(\sigma_n/BGSN)**bgn + (\sigma_t/BGST)**bgm > 1.0$$

When a contact node breaks due to the above criterion, standard contact is activated if it comes into contact again. If BGSN = 0.0 the first term is ignored. If BGST is zero, the second term is ignored.



11. For hard contact, with HGLUE=1:
  - a. The temperature of the contacting grid is tied to the temperatures of the contacted element face or the temperature of the rigid geometry when it has a scalar point associated with it.
  - b. The temperature of the contacting grid is set to the rigid geometry temperature when it has no scalar point associated with it.

**Note:** “Glued” thermal contact can result in overshoot of the temperatures at the interface, in particular, if two bodies that have non-uniform initial temperatures are placed in contact. The overshoot effect may be damped somewhat if one uses a near contact distance with some convective heat transfer.

12. For hard contact, with HGLUE=0:

The convective heat flow per unit area over the two interfaces is given by:

$$q = HCT \cdot (T_A - T_B)$$

where  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

13. For near contact:

$$\begin{aligned} q = & HCV \cdot (T_A - T_B) + \\ & HNC \cdot (T_A - T_B)^{BNC} + \\ & HNL \cdot (T_A^{BNL} - T_B^{BNL}) + \\ & \sigma \cdot EMISS \cdot (T_A^A - T_B^A) + \\ & \left[ HCT \cdot \left( 1 - \frac{dist}{DQNEAR} \right) + HBL \cdot \frac{dist}{DQNEAR} \right] \cdot (T_A \cdot T_B) \end{aligned}$$

where the last term is only activated when  $HBL \neq 0$ ,  $T_A$  is the contacting grid temperature and  $T_B$  is the face temperature in the contact point in case of a meshed body or the  $T_{BODY}$  temperature in case of a rigid geometry.

14. The heat transfer coefficients and associated exponents can all be temperature dependent, when they are entered as an integer value. This integer value is the table ID of a TABLEM1, TABLEM2 or TABL3D entry (formulas are not supported on TABL3D).
15. Table IDs of tables used on the BCTABLE and the BCBODY entry must be unique.
16. The penalty factor used in the augmented Lagrange method is by default derived from the contact characteristic distance and the stiffness of the deformable contact bodies involved (note that the dimension of the penalty factor is force per cubic length).

$$\text{PENALT} = \frac{0.5(S_i + S_j)}{\bar{L}}$$

The body stiffness ( $S_i$  and  $S_j$ ), are either defined by the average trace of the initial stress-strain law of the elements defining the two contact bodies or by the average bulk modulus for (nearly) incompressible rubber materials, whichever of the two is the largest.

For continuum elements, the characteristic length ( $\bar{L}$ ) is given by one half of the average length of all the edges being part of the contact boundary. For shell elements, the characteristic length is given by half of the average thickness of all the shell elements being part of a contact body. When there is contact between a solid and a shell element, then the characteristic length is defined by the shell element.

In case of contact with a rigid body, since there is no body stiffness associated with a rigid body, the default value is related to the deformation body only and given by:

$$\text{PENALT} = \frac{1000S_i}{\bar{L}}$$

17. By default, the threshold value of this penetration distance is 0.05 times the default contact characteristic distance.

$$\text{AUGDIST} = 0.05\bar{L}$$

18. The multipoint constraint equations (MPC equations) internally created from body contact can be printed out in standard Nastran punch file by using Case Control command, NLOPRM MPCPCH. This capability is good for all solutions except SOL 700.
19. When ISEARCH=0 (and ISTYP=0 in default on BCBODY), the search order is from lower BCBODOY ID to higher one to create the first set of contact constraints and then add the constraints in the search order from higher BCBODY ID to lower one as long as they are not in conflict with the first set.

When ISEARCH=2, the program looks into the smallest element edge at the outer boundary (and the smallest thickness in case of shell elements) of each BCBODY. Then, the search order of the two deformable contact bodies is determined by the following rule when ID1 < ID2

CL1= Min(1/20 of the smallest edge, 1/4 of the smallest thickness) of BCBODY ID1

CL2= Min(1/20 of the smallest edge, 1/4 of the smallest thickness) of BCBODY ID2

CL1 and CL2 refers to characteristic length of BCBODY ID1 and ID2.

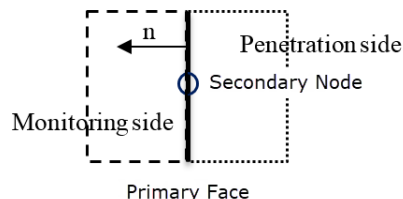
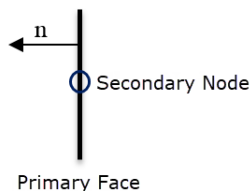
The search order is from lower BCBODY ID1 to higher BCBODY ID2 if  $CL1 \leq 1.05 * CL2$ .

Otherwise, if  $CL1 > 1.05 * CL2$  the search order is from BCBODY ID2 to BCBODY ID1.

20. Fields ISEARCH, DQNEAR and "BKGL" are not supported in segment to segment contact analysis.

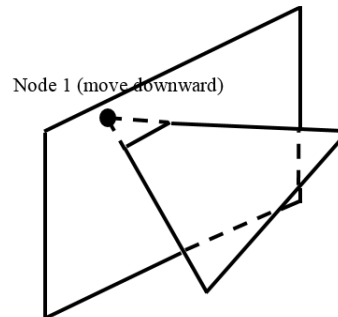
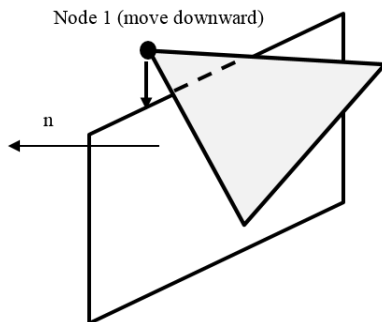
21. If STKSLP is set to 0.0 (Default), the sticking stiffness  $K1$  is equal to the maximum friction force ( $\mu \cdot F_{NORMAL}$ , where  $\mu$  is the friction coefficient) divided by the maximum sticking displacement. Otherwise,  $K1$  is equal to the maximum friction force divided by the value of STKSLP.
22. When the initial stress-free (ICoord=1), Node-to-Seg updates model geometry to close the gap, nodes are projected onto the contact body; but Seg-to-Seg treats initial stress-free as pre-stress in the equation without geometry update, no node projection is considered.
23. When METHOD=RELLIPS is used, BCGRID or BCMATL are only available for SECNDRY body and BCELIPS is only available for PRIMARY body. When BCMATL is used, the MATRIG id or RBE2 id are only acceptable.
24. The options TOP/BOTTOM are useful in the following cases:
  - a. When a secondary node initially is located on the primary face (see the picture below), the contact situation is uniquely defined, only if the TOP or BOTTOM side of the primary surface is defined:
 

SIDE=TOP case:



- b. When hooking of secondary nodes on the wrong side of a primary face might occur. This often is the case when the primary face is at the edge of a shell element structure:
 

SIDE=BOTTOM case: penetration of node 1 (SIDE=BOTH: no penetration of node 1)



25. Recommended usage of WEIGHT:

SECNDRY	PRIMARY	WEIGHT
Deformable	Deformable	BOTH
Deformable	Rigid	SECNDRY
Rigid	Deformable	PRIMARY
Rigid	Rigid	NONE

Default setting when BCELIPS is used in SECNDRY or PRIMARY:

SECNDRY	PRIMARY	WEIGHT
Non-BCELIPS	BCELIPS	SECNDRY
BCELIPS	Non-BCELIPS	NONE
	BCELIPS	

## BCTABLE - Glued Option

Defines a Contact Table for Permanently Glued Contact

Defines a glued contact used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and SOL 400 for the permanently-glued or general contact.

Format: (SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200 and 400 only)

1	2	3	4	5	6	7	8	9	10
BCTABLE	ID			NGROUP	COPTS	COPTM			
	“SECNDRY”	IDSEC1	ERROR			CINTERF	IGLUE		
		ISEARCH	ICOORD						
		“FBSH”		BIAS	SLIDE		COPTS1	COPTM1	
	“PRIMARY”	IDPRIM1	IDPRIM2	IDPRIM3	IDPRIM4	IDPRIM5	IDPRIM6	IDPRIM7	
		IDPRIM8	IDPRIM9	...					

Examples:

BCTABLE	101			2	61	61			
	SECNDRY	20	0.002		0.2		1		
		1							
		FBSH		0.01			60	60	
	PRIMARY	31	32	33					
	SECNDRY	40					1		
	PRIMARY	51	52	53	54				

Describer	Meaning
ID	Identification number of a BCTABLE entry to be selected by BCONTACT Case Control command. See Remark 5. for the general contact. Case Control, BCONTACT=ALLBODY, cannot be used for the permanently glued contact. See Remark 1. (Integer ≥ 0; Required)
NGROUP	Flag to indicate the number of pairs of “SECNDRY” and “PRIMARY” entries in the continuation lines. (Integer > 0; Required)
COPTS1, COPTM	Flag to indicate how secondary and primary surfaces may contact. See Remark 6. for the general contact.
“SECNDRY”	Indicates that this line defines the secondary or touching body and its parameters.
IDSEC1	Identification number of a BCBODY Bulk Data entry defining the touching body. (Integer > 0; Required)
ERROR	Distance below which a node is considered touching a body. When this field is left blank or 0.0, the value is taken from BCPARA = 0, ERROR, if it is specified. Otherwise, Nastran automatically calculates the value. See the Bulk Data entry <a href="#">BCPARA, 1333</a> for more details. (Real)

Describer	Meaning						
CINTERF	Interference closure amount, normal to the contact surface. For CINTERF > 0.0, overlap between bodies, CINTERF < 0.0, gap between bodies. (Real; Default = 0.0)						
IGLUE	<p>Flag to activate the glue options. See the descriptions of various options in BCTABLE for the general contact. This field must be specified for the glued contact. See Remark 4. (Integer &gt; 0; Required).</p> <ol style="list-style-type: none"> <li>1. Activates the glue option. In the glue option, all degrees-of- freedom of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified.</li> <li>2. Activates a special glue option to insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.</li> <li>3. Insures full moment carrying glue when shells contact. This option is recommended when moments are important and there is no gap or overlap between contact surfaces or initial stress free contact is specified.</li> <li>4. Insures full moment carrying glue when shells contact. The node will not be projected onto the contact body and an existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body.</li> </ol>						
ISEARCH	<p>Option for contact searching order, from a secondary (touching) to a primary (touched) body or visca versa, for deformable contact. See the options in BCTABLE for the general contact. (Integer; Default = 0)</p> <table border="0"> <tr> <td data-bbox="482 1211 571 1237">0</td> <td data-bbox="571 1211 1330 1307">(Double orders search) 0(Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE.</td> </tr> <tr> <td data-bbox="482 1315 571 1341">1</td> <td data-bbox="571 1315 1330 1341">(Single order search) the searching order is from Secondary to Primary</td> </tr> <tr> <td data-bbox="482 1350 571 1376">2</td> <td data-bbox="571 1350 1330 1428">(Single order search) let the program decide which search order. See Remark 19. in BCTABLE.</td> </tr> </table>	0	(Double orders search) 0(Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE.	1	(Single order search) the searching order is from Secondary to Primary	2	(Single order search) let the program decide which search order. See Remark 19. in BCTABLE.
0	(Double orders search) 0(Double orders search) the search order is from lower BCBODOY ID's to higher ones first. Then it searches the opposite order. See Remark 19. in BCTABLE.						
1	(Single order search) the searching order is from Secondary to Primary						
2	(Single order search) let the program decide which search order. See Remark 19. in BCTABLE.						
ICOORD	Enter 1 to modify the coordinates of a node in contact with a deformable body so that stress-free initial contact can be obtained. Enter 2 to extend the tangential error tolerance at sharp corners of deformable bodies to delay sliding off a contacted segment. Enter 3 to have both 1 and 2 active. (Integer; Default = 0)						
"FBSH"	Character string used to introduce BIAS, COPTS1, and COPTM1.						

Describer	Meaning
BIAS	Contact tolerance bias factor. If this field is left blank, the default is the BIAS of the BCPARA entry. A nonblank entry will override the BIAS entered on the BCPARA entry. Note 0.0 is not default, and will override the BIAS on BCPARA. SYSTEM(758)=1 will set 0.0 same as blank. ( $0.0 \leq \text{Real} \leq 1.0$ )
SLIDE	Delayed slide off distance. This entry should not be made unless ICOORD $\geq 2$ (see above). When using the delayed slide off option, a node sliding on a segment will slide off this segment only if it passes the node (2-D) or edge (3-D) at a sharp corner over a distance larger than the delayed slide off distance. By default, the delayed slide off distance is related to the dimensions of the contacted segment by a 20 percent increase of its isoparametric domain. (Real; Default = 0.2)
COPTS1, COPTM1	Flag to indicate how individual pair of secondary and primary contact surfaces may contact. These two fields override COPTS and COPTM. See Remark 6.
“PRIMARY”	Indicates the start of the list of bodies touched by touching body IDSEC1.
IDPRIMi	Identification numbers of BCBODY Bulk Data entries defining touched bodies. (Integer > 0; Required)

#### Remarks:

1. This BCTABLE entry is tailored specifically for what we called the permanently-glued contact (or tied contact), from the entry of BCTABLE for the general contact, where the glued is an option in the approaching-and-touched contact. Most of the descriptions and remarks on the fields in this entry can be found in the BCTABLE for the general contact.
2. Permanent glued contact for small rotation condition is defined when all IGLUE fields of BCTABLE, reference by the very first loadcase, (the 1st STEP of the 1st SUBCASE in SOL 400 or the 1st SUBCASE in others) is set to larger than 0, IGLUE(>0). In BCPARA with ID=0, the value of NLGLUE may not be 1.

For large deformation and large rotation condition, permanent glued contact is activated by negative IGLUE (<0) and works in corresponding subcase or step in the corresponding subcase or step where it is defined, no matter what is set (1 or 0). This type of permanent glued contact with large deformation and large rotation may be applied together with combination of general glued contact and touching contact. IGLUE will be changed to negative automatically if SYSTEM(758)=2 when Permanently Glued setting is found with large rotation/deformation effect turned on.

3. The MPC equations internally created for the glued contact can be printed out in Nastran standard punch file by using Case Control command, NLOPRM MPCPCH=BEGN.
4. IGLUE= $\pm 2$  or  $\pm 4$  is favorable to passing the GROUNDCHECK. The initial stress-free contact is also available to preserve the six rigid-body modes with IGLUE = 1 or 3.
5. For SOLs 101 and 400, the BCTABLE with ID=0 will be used in loadcase automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which

are the Grid Points on the surface of all deformable BCBODY's. To place an entry in any physical loadcase (SUBCASE or STEP), the ID must be selected by the Case Control command BCONTACT=ID. When BCONTACT=ALLBODY, there is no ID of BCTABLE specified; therefore, the default values of all entries of BCTABLE are assumed.

- COPTS, COPTM COPTS1 and COPTM1 are packed numbers designating how the surfaces may contact using the formula

$$COP_{xxx}=A+10*B+1000*C$$

where the following codes apply:

A: the outside of the solid elements in the body

- = 1: the outside will be in the contact description (Default)

B (flexible bodies): the outside of the shell elements in the body

- = 1: both top and bottom faces will be in the contact description, thickness offset will be included (Default)
- = 2: only bottom faces will be in the contact description, thickness offset will be included
- = 3: only bottom faces will be in the contact description, shell thickness will be ignored
- = 4: only top faces will be in the contact description, thickness offset will be included
- = 5: only top faces will be in the contact description, shell thickness will be ignored
- = 6: both top and bottom faces will be in the contact description, shell thickness will be ignored

Note: The choice B = 6 for both bodies in a contact combination is only meaningful for glued contact. If in such cases separation is allowed, separated nodes will not come into contact anymore, unless a new CONTACT TABLE is defined to reset the value of B.

B (rigid bodies): the rigid surface

- = 1: the rigid surface should be in the contact description (Default)

C (flexible bodies): the edges of the body

- = 1: only the beam/bar edges are included in the contact description (Default)
- = 10: only the free and hard shell edges are included in the contact description
- = 11: both the beam/bar edges and the free and hard shell edges are included in the contact description

Note that C has no effect if beam-to-beam contact is **not** switched on (BEAMB ≠ 1 on BCPARA)

- It is not supported that a permanently-glued contact is used to glue a deformable body to a rigid one. If it is a SOL 101 or SOL 400 job, the general contact with glued option must be performed.



## BCTABL1 Defines a Contact Table

BCTABL1 specifies a list of contact pairs through the BCONECT option for SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112, 200, 400 and 700. Only SOL 101, 400 and 700 can support all contact types: touching contact, glued, step glue and permanent glue (see [Chapter 7: Contact Types](#) of *Nastran SOL 400 Getting Started Guide*). Note that SOL 700 calls it “tie” contact to glue (also step glue or permanent glue). SOL 103, 105, 107 - 112 and standard 200 can only support permanent glue. When SOL 400 optimization, it can support all contact types. This contact table is activated in the BCONTACT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
BCTABL1	BCID	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Examples:

BCTABL1	2	198	62	75	8	159	31	82	
	17								

BCTABL1	0	23	56						
---------	---	----	----	--	--	--	--	--	--

BCTABL1	5	31	35	THRU	40	163	THRU	169	
	1077	1088							

Describer	Meaning
BCID	Unique identification number referenced by a BCONTACT Case Control command.  See Remark 3. (Integer ≥ 0; default=0)
IDi	Identification number of BCONECT entry (Integer > 0 or “THRU”; ID1 is required). See Remark 4. and 5.

Remarks:

1. BCTABL1 defines surface contact. If BCTABL1 does not exist, the Case Control command BCONTACT=BCID may refer to the BCONECT Bulk Data entry directly.
2. If neither BCTABL1 nor BCONECT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If BCTABL1 or BCONECT is given, the default for every body is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. BCTABL1 or BCONECT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.

3. For SOLs 101 and 400, the BCTABL1 or BCONECT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable BCBODY1's. To place an entry in any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or BCONECT=BCID. When BCONTACT=ALLBODY, there is no BCID of BCTABL1 or BCONECT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.
4. When the "THRU" option is used, all BCONECT entries associated with intermediate BCONECT IDs must exist. The word "THRU" may not appear in field 3 or 9 (2 or 9 for continuations).
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**BCTRIM****Defines the Geometry of a Trimming Curve**

Defines the geometry of a trimming curve used to specify a NURBS for a rigid contact body.

Format:

1	2	3	4	5	6	7	8	9	10
BCTRIM	IDtrim	NPTUtrim	NORUtrim	NSUBtrim					
	“COORD”	Xisoparam1	Yisoparam1	Xisoparam2	Yisoparam2	-etc.-			
	“HOMO”	Homot1	Homot2	Homot3	-etc.-				
	“KNOT”	Knott1	Knott2	Knott3	-etc.-				

Examples:

BCTRIM	202	2	2	50					
	COORD	0.0	0.0	1.0	0.0				
	HOMO	1.	1.						
	KNOT	0.0	0.0	1.	1.				

Describer	Meaning
IDtrim	Unique trimming vector identification number referenced by a BCNURBS entry. (Integer > 0)
NPTUtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
“COORD”	Indicate the start of the list of isoparametric coordinates of points.
Xisoparam1, Yisoparam1, Xisoparam2, Yisoparam2, etc.	First and second coordinates of point in isoparametric space. There must be NPTUtrim set of (Xisoparam, Yisoparam) entries. (Real)
“HOMO”	Indicate the start of the list of homogeneous coordinates of this trimming vector.
Homot1, Homot2, Homot3, etc.	Homogeneous coordinates of this trimming vector. There must be NPTUtrim entrie. (Real; 0.0 to 1.0)
“KNOT”	Indicate the start of the list of knot vectors of this trimming vector.
Knot1, Knot2, Knot3, etc.	Knot vectors of this trimming vector. There must be (NPTUtrim+NORUtrim) entries. (Real; 0.0 to 1.0)

Remarks:

1. BCTRIM is referenced by a BCNURBS entry to define the trimming curve of a NURBS for a rigid contact body.

## BDYLIST Fluid Boundary List

Defines the boundary between a fluid and a structure.

Format:

1	2	3	4	5	6	7	8	9	10
BDYLIST	RHO	IDF1	IDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	-etc.-							

Example:

BDYLIST	.037	AXIS	432	325	416	203	256	175	
---------	------	------	-----	-----	-----	-----	-----	-----	--

Describer	Meaning
RHO	Fluid mass density at boundary. (Real $\geq 0.0$ or blank; Default is DRHO on the AXIF entry)
IDF1	Identification number of a RINGFL entry. (Integer $> 0$ or Character = "AXIS" or "LAXIS") See Remark 7.
IDF2-IDFi	Identification number of additional RINGFL entries. (Unique Integers $> 0$ )

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Each entry defines a boundary if  $RHO \neq 0.0$ . The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word "AXIS" defines an intersection with the polar axis of the fluid coordinate system.
4. There is no limit to the number of BDYLIST entries specified. If the fluid density varies along the boundary, there must be one BDYLIST entry for each interval between fluid points.
5. The BDYLIST entry is not required and should not be used to specify a rigid boundary where structural points are not defined. Such a boundary is automatically implied by the omission of a BDYLIST.
6. If  $RHO=0.0$ , no boundary matrix terms will be generated to connect the GRIDB points to the fluid. This option is a convenience for structural plotting purposes. GRIDB points may be located on a fluid ring (RINGFL entry) only if the rings are included in a BDYLIST.
7. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS.

## BDYOR

### CHBDYi Entry Default Values

Defines default values for the CHBDYP, CHBDYG, and CHBDYE entries.

Format:

1	2	3	4	5	6	7	8	9	10
BDYOR	TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		PID	GO	
	CE	E1	E2	E3					

Example:

BDYOR	AREA4	2	2	3	3		10		
-------	-------	---	---	---	---	--	----	--	--

Describer	Meaning
TYPE	Default surface type. See Remark 2. (Character)
IVIEWF	Default identification number of front VIEW entry. (Integer > 0 or blank)
IVIEWB	Default identification number of back VIEW entry. (Integer > 0 or blank)
RADMIDF	Default identification number of a RADM entry for front face. (Integer ≥ 0 or blank)
RADMIDB	Default identification number of a RADM entry for back face. (Integer ≥ 0 or blank)
PID	Default PHBDY property entry identification number. (Integer > 0 or blank)
GO	Default orientation grid point. (Integer ≥ 0; Default = 0)
CE	Default coordinate system for defining the orientation vector. (Integer ≥ 0 or blank)
E1, E2, E3	Default components of the orientation vector in coordinate system CE. The origin of this vector is grid point G1 on a CHBDYP entry. (Real or blank)

Remarks:

1. Only one BDYOR entry may be specified in the Bulk Data Section.
2. TYPE specifies the type of CHBDYi element surface; allowable values are: POINT, LINE, REV, AREA3, AREA4, ELCYL, FTUBE, AREA6, AREA8, and TUBE.
3. IVIEWF and IVIEWB are specified for view factor calculations only (see VIEW entry).
4. GO is only used from BDYOR if neither GO nor the orientation vector is defined on the CHBDYP entry and GO is > 0.
5. E1, E2, E3 is not used if GO is defined on either the BDYOR entry or the CHBDYP entry.

## BEADVAR Topography Design Variable

Defines design region for topography (bead or stamp) optimization.

Format:

1	2	3	4	5	6	7	8	9	10
BEADVAR	ID	PTYPE	PID	MW	MH	ANG	BF	SKIP	
	“DESVAR”	NORM/XD	YD	ZD	CID	XLB	XUB	DELXV	
	“GRID”	NGSET	DGSET						

Example using NORM:

BEADVAR	10	PSHELL	101	10.0	20.0	70.0			
---------	----	--------	-----	------	------	------	--	--	--

Example using “DESVAR” and “GRID”:

BEADVAR	10	PSHELL	101	10.0	20.0	70.0		NONE	
	DESVAR	2.0	3.0	4.0		-1.0	1.0		
	GRID	100							

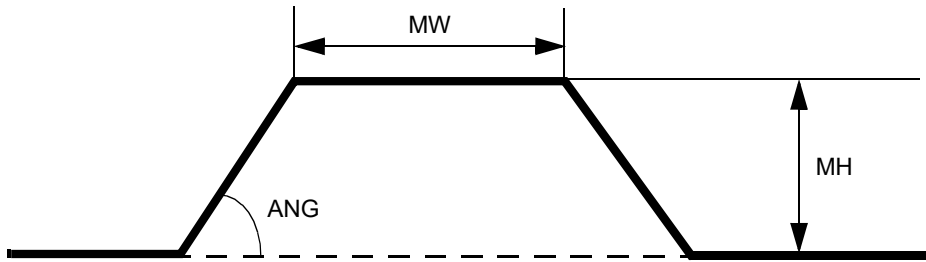
Descriptor	Meaning
ID	Unique topography design region identification number. (Integer > 0)
PTYPE	Property entry type. Used with PID to identify the element nodes to be designed. (Character: “PSHELL”, “PSHEAR”, “PCOMP”, or “PCOMPG”.)
PID	Property entry identifier. See Remark 1. (Integer > 0)
MW	Minimum bead width. This parameter controls the width of the beads. The recommended value is between 1.5 and 2.5 times the average element width. See Remark 2. (Real > 0.0)
MH	Maximum bead height (Real > 0.0). This parameter sets the maximum height of the beads when XUB=1.0 (as Default). See Remark 2.
ANG	Draw angle in degrees (0.0 < Real < 90.0). This parameter controls the angle of the sides of the beads. The recommended value is between 60 and 75 degrees.
BF	Buffer zone ('yes' or 'no'; Default='yes'). This parameter creates a buffer zone between elements in the topography design region and elements outside the design region when BF='yes'. See Remark 3.

Describer	Meaning
SKIP	Boundary skip (“bc”, “load”, “both”, or “none”; Default = “both”). This parameter indicates which element nodes are excluded from the design region. “bc” indicates all nodes referenced by “SPC” and “SPC1” are omitted from the design region. “load” indicates all nodes referenced by “FORCE”, “FORCE1”, “FORCE2”, “MOMENT”, “MOMENT1”, “MOMENT2”, and “SPCD” are omitted from the design region. “both” indicates nodes with either “bc” or “load” are omitted from the design region. “none” indicates all nodes associated with elements referencing PID specified in field 4 are in the design region.
“DESVAR”	Indicates that this line defines bead design variables that are automatically generated.
NORM/XD, YD, ZD	Bead vector (draw direction). Norm indicates the shape variables are created in the normal directions to the elements. If XD, YD, and ZD are provided, the shape variables are created in the direction specified by the xyz vector defined by XD/YD/ZD that is given in the basic coordinate system or CID. See Remark 4. (Character or Real; Default = blank = norm).
CID	Coordinate system ID used for specifying draw direction (Blank or Integer > 0; Default = blank = basic coordinate system)
XLB	Lower bound. (Real < XUB or blank; Default = blank=0.0). This ensures the lower bound on grid movement equal to XLB*MH. See Remark 5.
XUB	Upper bound. (Real > XLB or blank; Default = 1.0). This sets the upper bound of the beads equal to XUB*MH. See Remark 5.
DELXV	Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)
“GRID”	Indicates this line defines what element nodes can be added and/or removed from topography design regions.
NGSET	All grids listed on a Bulk Data entry SET1 = NGSET are removed from topography design regions.
DGSET	All grids listed on a Bulk Data entry SET1 = DGSET are added to topography design regions.

#### Remarks:

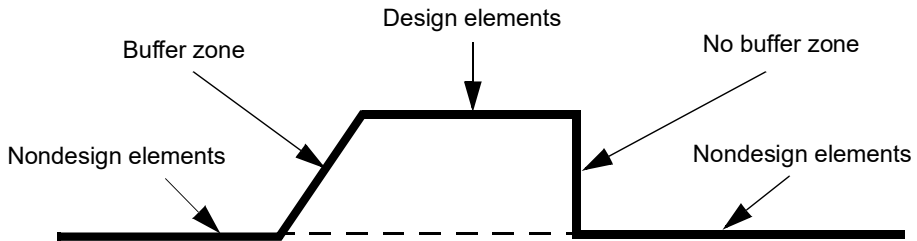
1. Multiple BEADVAR’s are allowed in a single file. Combined topometry, topology, topography, sizing, and shape optimization is supported in a single file.

2. The user can provide allowable bead dimensions.

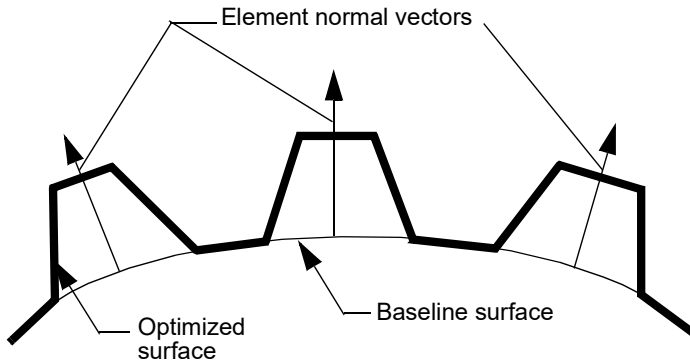


Bead Dimensions

3. It is recommended to set buffer zone = yes to maintain a good quality of mesh during topography optimization.

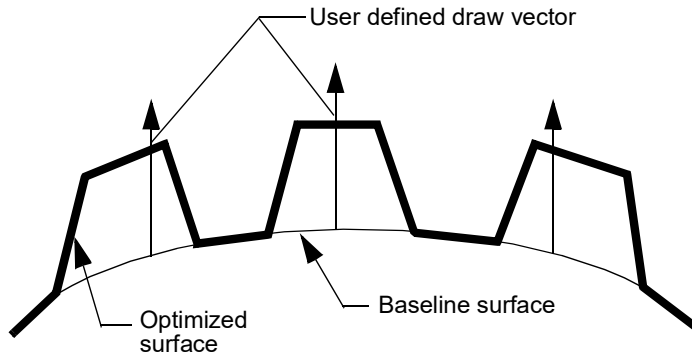


4. The grids moves in the normal direction. All element grids referenced by one BEADVAR entry must follow the right hand rule.



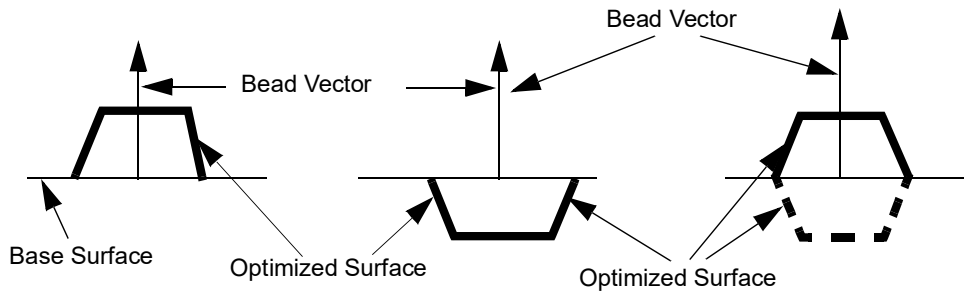
Element Normal





User's Provided Draw Direction

- To force the grids to move only in the positive bead vector direction (one side of the surface), use  $XLB = 0.0$ . To force the grids to move only in the negative bead vector direction (another side of the surface), use  $XUB = 0.0$ . To allow grids to move in both positive and negative bead vector directions, use  $XLB < 0.0$  and  $XUB > 0.0$ . For example,



(a)  $XLB = 0.0$  and  $XUB = 1.0$     (b)  $XLB = -1.0$  and  $XUB = 0.0$     (c)  $XLB = -1.0$  and  $XUB = 1.0$

- The jobname.h5 file (created by specifying MDLPRM,HDF5,0 in the bulk data input) has topography optimization results (shape changes) that can be viewed in Patran. The text file jobname.pch also has updated grid coordinates that can be copied to the original file, replace the original grids, and imported to Patran and other post-processors to view topography optimization results.
- The BEADVAR entry cannot be used with thermal loads.

## BEAMOR CBEAM Entry Default Values

Defines default values for field 3 and fields 6 through 9 of the CBEAM entry.

Format:

1	2	3	4	5	6	7	8	9	10
BEAMOR		PID			X1	X2	X3	OFFT	

Example:

BEAMOR		39			0.6	2.9	-5.87	GOG	
--------	--	----	--	--	-----	-----	-------	-----	--

Alternate Format and Example:

BEAMOR		PID			G0			OFFT	
--------	--	-----	--	--	----	--	--	------	--

BEAMOR		39			86			GOG	
--------	--	----	--	--	----	--	--	-----	--

Describer	Meaning
PID	Property identification number of the PBEAM or PBCOMP entry. (Integer > 0 or blank)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (default), or in the basic coordinate system. See Remark 5. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ , using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB on CBEAM entry)
OFFT	Offset vector interpretation flag. See Remark 5. (Character or blank)

Remarks:

1. The contents of fields on this entry will be assumed for any CBEAM entry with corresponding fields that are blank.
2. Only one BEAMOR entry is allowed.
3. For an explanation of beam element geometry, see the CBEAM entry description.
4. If X1 or G0 is integer, G0 is used. If X1 or G0 is blank or real, then X1, X2, X3 is used.
5. OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\hat{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. (Note: The character “O” in the table replaces the obsolete character “E”.)

## BFRIC

## Contact Friction

Defines frictional properties between two bodies in contact using the slideline contact in SOL 106 and SOL 129. SOL 400 is the recommended approach for contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BFRIC	FID		FSTIF	MU1					

Example:

BFRIC	33			0.3					
-------	----	--	--	-----	--	--	--	--	--

Descriptor	Meaning
FID	Friction identification number. See Remark 1. (Integer > 0)
FSTIF	Frictional stiffness in stick. See Remarks 2. and 3. (Real > 0.0; Default = automatically selected by the program)
MU1	Coefficient of static friction. (Real > 0.0)

Remarks:

1. This identification number must be unique with respect to all other friction identification numbers. This is used in the FRICID field of BCONP Bulk Data entry.
2. The value of frictional stiffness requires care. A method of choosing its value is to divide the expected frictional strength ( $MU1 \cdot \text{expected normal force}$ ) by a reasonable value of the relative displacement that may be allowed before slip occurs. The relative value of displacement before slip occurs must be small compared to expected relative displacements during slip. A large stiffness value may cause poor convergence, while too small a value may cause poor accuracy.  
Frictional stiffness specified by the user is selected as the initial value. If convergence difficulties are encountered during the analysis, the frictional stiffness may be reduced automatically to improve convergence.
3. The stiffness matrix for frictional slip is unsymmetric. However, the program does not use the true unsymmetric matrix. Instead the program uses only the symmetric terms. This is to avoid using the unsymmetric solver to reduce CPU time.

BIAS

MESH Bias Definition

Specifies a variation of the mesh-size in one direction for use in the MESH entry in SOL 700. The MESH entry can create a biased or non-uniform mesh. A uniform mesh consists of a number of planes separated by a fixed distance, but for a biased mesh the distance between subsequent planes can differ. The BIAS definition allows specifying the locations of planes in one direction. For a number of intervals the density of planes can be specified.

Format:

1	2	3	4	5	6	7	8	9	10
BIAS	BID								
	X0	GROWTH0	N0	DXS0	DXE0				
	X1	GROWTH1	N1	DXS1	DXE1				
	Xi	GROWTHi	Ni	DXSi	DXEi				

Example:

BIAS	100								
	-4.5	0.2	15						
	-1	1	20						
	1			0.1	0.46				

Describer	Meaning
BID	Unique bias number. Must be referenced from MESH entry. (Integer; Required)
Xi	Begin coordinate of an interval. The interval ends at the $X_{i+1}$ entry. (Real; Default = 0.0)
GROWTHi	GROWTHi is the ratio between the step size at the beginning of the interval and at the end of the interval. If it is smaller than 1.0 then the mesh refines when going from the beginning of the interval to the end of the interval. (Real>0) See Remarks 1.- 3.
Ni	Ni is the number of elements inside the interval. (Integer>0) See Remarks 1.- 3.
DXSi	DXSi is the start element size of the interval. (Real>0) See Remarks 1.- 3.
DXEi	DXEi is the end element size of the interval. (Real>0) See Remarks 1.- 3.

Remarks:

1. The begin point of the first interval has to be equal to the X0 field of the MESH entry and may be left unspecified. The end point of the last interval is given by X0 + DX as specified by the MESH entry. In the example above, the first interval is given by [-4.5,-1], the second one by [-1,1] and the last one by [1,4.5], assuming that X0 + DX = 4.5 on the MESH entry that references the bias definition from IBIDX.

2. To define the bias in an interval four fields are available. These are GROWTHi, Ni, DXSi and DXEi. To specify the bias inside an interval two of these four variables have to be specified. The other two variables have to be left blank. In addition the mesh size can be chosen constant by defining Ni and leaving GROWTHi, DXSi and DXEi blank.

This gives seven methods of specifying a bias;

1. Define GROWTHi and Ni. DXSi and DXEi have to be left blank.
2. Define DXSi and DXEi. GROWTHi and Ni have to be left blank.
3. Define DXSi and Ni. GROWTHi and DXEi have to be left blank.
4. Define DXEi and Ni. GROWTHi and DXSi have to be left blank.
5. Define GROWTHi and DXSi. Ni and DXEi have to be left blank.
6. Define GROWTHi and DXEi. Ni and DXSi have to be left blank.
7. Only define Ni and leave GROWTHi, DXSi and DXEi blank.

For method 2 it can happen that the biased elements do not exactly fit into the interval.

To get a good fit a small change to defined start and end step sizes is made. These changes in general amount to a few percents.

For method 5 and 6 a small change to the specified growth factor can be made so that the biased elements fit into the interval.

3. The algorithm for each method first determines an appropriate growth factor so that the biased elements fit into the interval. In addition the number of planes is determined. For method 2, 3 and 4 this is done by using bisection. The location of the planes is given by:

$$X(i) = X(i - 1) + \Delta X(i - 1)$$

$$\Delta X(i) = GROWFAC \times \Delta X(i - 1)$$

$$GROWTHi = GROWFAC^{N-1}$$

Here X(i) denotes the location of the Euler plane,  $\Delta X_i$  denotes the step size of the interval and N denotes the number of Euler planes. The index i runs across the Euler planes. The variable GROWFAC denotes the grow factor between planes within the interval. The locations of the planes X(i) are written to the OUT file as part of MESH output. In addition, the growth of the element sizes is written out in the next column. This is given as the ratio in element size between the layer of elements to the right of the plane and to the left of the plane. Let x0, x1 and x2 denote three subsequent planes, then the element size to the left of the x1-plane is given x1-x0 and to the right it is given by x2-x1. The ratio by which the element size grows if one goes across the x1-plane is:

$$\frac{X2 - X1}{X1 - X0}$$

To get physically meaningful results, this value should not exceed 1.3 or be smaller than 0.7 .

Within each interval the ratio in element size equals GROWFAC. But the element size between two adjacent elements that are in two different intervals can differ from GROWFAC. Here each interval has a distinct GROWFAC variable.

For both method 2, 5 and 6 The growth factor GROWFAC and start and end step sizes that are actually used can be obtained from this plane summary in the OUT file. In this summary for each plane a growth factor and step size is specified.

Also the total number of elements is written out.

## BJOIN

Defines (multiple) pairs of grid points of one-dimensional and/or shell elements to be joined during the analysis in SOL 700. When the failure criterion for a grid-point pair is satisfied, the grid-point pair is removed from the join and the grid-point motion is computed for the separate grid points. The join ceases to exist when all pairs of the join have failed, after which all of the grid points of the join are treated as separate grid points.

Format:

1	2	3	4	5	6	7	8	9	10
BJOIN	BID	SID	TOL	TYPE		SN		SS	
					MULTI				
		TF							

Example:

BJOIN	1	2		SPOTWELD		1.E3		1.E3	
-------	---	---	--	----------	--	------	--	------	--

Describer	Meaning	Type	Default
BID	BJOIN number.	Integer > 0	Required
SID	BCGRID ID of set of grid points.	Integer > 0	Required
TOL	Tolerance used in matching grid point pairs.	Real > 0.0	1.E-4
TYPE	Type of failure criterion. SPOTWELD Spotweld-type failure.	C	SPOTWELD
SN	Failure force in tension.	Real > 0.0	No failure
SS	Failure force in shear.	Real > 0.0	No failure
MULTI	Multiple breakable joins, where the grid points must be entered as a sequence of BJOIN pairs. YES The grid points are entered on the BCGRID entry as a sequence of BJOIN pairs. NO The code creates BJOIN pairs for every two grid points entered on the BCGRID entry when the grid point positions fall within the tolerance (TOL).	C	YES
TF	Failure time for nodal constraint set.	Real	1.0E20



**Remarks:**

1. Nodes connected by a spot weld cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.
2. When the failure time,  $TF$ , is reached the spot weld becomes inactive and the constrained nodes may move freely.
3. Note that the shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted.
4. For MULTI=YES, a spotweld or rivet will always be created, regardless if their distance is larger than the value of TOL. TOL will only be used for MULTI=NO.

## BLSEG                      Boundary Line Segments

Defines a curve that consists of a number of line segments via grid numbers that may come in contact with another body.

Format:

1	2	3	4	5	6	7	8	9	10
BLSEG	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BLSEG	ID	G1	"THRU"	G2	"BY"	INC			
-------	----	----	--------	----	------	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	G8	"THRU"	G9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BLSEG	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Describer	Meaning
ID	Line segments identification number. See Remark 2. (Integer > 0)
Gi	Grid point identification numbers on a curve in a continuous topological order so that the normal to the segment points toward the other curve. See Remark 3. (Integer > 0)
INC	Grid point identification number increment. See Remark 3. (Integer or blank)

Remarks:

1. A line segment is defined between every two consecutive grid points. Thus, the number of line segments defined is equal to the number of grid points specified minus 1. A corresponding BWIDTH Bulk Data entry may be required to define the width/thickness of each line segment. If the corresponding BWIDTH is not present, the width/thickness for each line segment is assumed to be unity.

2. ID must be unique with respect to all other BLSEG entries. Each line segment has a width in 3-D sideline and a thickness in a 2-D slideline contact to calculate contact stresses. The width/thickness of each line segment is defined via BWIDTH Bulk Data entry. The ID in BLSEG must be same as the ID specified in the BWIDTH. That is, there must be a one to one correspondence between BLSEG and BWIDTH. BWIDTH Bulk Data entry may be omitted only if the width/thickness of each segment is unity.
3. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The normal to the segment is determined by the cross product of the slideline plane vector (i.e., the Z direction of the coordinate system defined in the 'CID' field of BCONP Bulk Data entry) and the tangential direction of the segment. The tangential direction is the direction from node 1 to node 2 of the line segment.
5. A curve may be closed or open. A closed curve is specified by having the last grid point identification number the same as the first grid number.
6. See BCBODY for use of BLSEG in 3D contact.

**BNDFIX****Fixed Boundary Degrees-of-Freedom**

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFIX	2	135	14	6					
--------	---	-----	----	---	--	--	--	--	--

Describer	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BNDFIX and BSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREEi/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREEi/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREEi/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREEi/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

## BNDFIX1 Fixed Boundary Degrees-of-Freedom, Alternate Form of BNDFIX Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFIX1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BNDFIX1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BNDFIX1	C	ID1	“THRU”	ID2					
BNDFIX1	3	6	THRU	32					

Describer	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1 < ID2)

Remarks:

1. BNDFIX1 and BSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

## BNDFREE

## Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFREE	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BNDFREE	124	1	5	23	6	16			
---------	-----	---	---	----	---	----	--	--	--

Describer	Meaning
ID <sub>i</sub>	Grid or scalar point identification number. (Integer > 0)
C <sub>i</sub>	Component numbers. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

Remarks:

1. BNDFREE and CSET entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSET<sub>i</sub>/BNDFIX<sub>i</sub> or CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSET<sub>i</sub>/BNDFIX<sub>i</sub> entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present or both BSET<sub>i</sub>/BNDFIX<sub>i</sub> and CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDFRE1** Free Boundary Degrees-of-Freedom, Alternate Form of BNDFREE Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BNDFRE1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

Example:

BNDFRE1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

BNDFRE1	C	ID1	“THRU”	ID2					
BNDFRE1	3	6	THRU	32					

BNDFRE1		“ALL”							
BNDFRE1		ALL							

Descriptor	Meaning
C	Component number. (Integer zero or blank for scalar points, or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks)
IDI	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2)
ALL	All a-set degrees-of-freedom will be set free (included in c-set).

Remarks:

1. BNDFRE1 and CSET1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.



4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**BNDGRID****Boundary Grid Points**

Specifies a list of grid point identification numbers on design boundaries or surfaces for shape optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
BNDGRID	C	GP1	GP2	GP3	GP4	GP5	GP6	GP7	
	GP8	-etc.-							

Example:

BNDGRID	123	41	42	43	44	45	46	47	
	49								

Alternate Format and Example:

BNDGRID	C	GP1	"THRU"	GP2					
BNDGRID	123	41	THRU	49					

Describer	Meaning
C	Component number (any unique combination of integers 1 through 6 with no embedded blanks). See Remark 1.
GPi	Shape boundary grid point identification number. (0 < Integer < 1000000; For THRU option, GP1 < GP2)

Remarks:

1. C specifies the components for the listed grid points for which boundary motion is prescribed.
2. Multiple BNDGRID entries may be used to specify the shape boundary grid point identification numbers.
3. Both fixed and free shape boundary grid point identification numbers are listed on this entry.
4. The degrees-of-freedom specified on BNDGRID entries must be sufficient to statically constrain the model.
5. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
6. Fluid grids are not supported for BNDGRID.

## BOLT Defines the Multi-Point Constraints for a Bolt

Defines a rigid bolt by a set of MPC constraints.

Format:

1	2	3	4	5	6	7	8	9	10
BOLT	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	etc.					
	BOTTOM	GB1	GB2	GB3	GB4	GB5	GB6	GB7	
		GB8	GB9	etc.					

Example:

BOLT	100	1025							
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Describer	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)

Remarks:

1. The GRIDS entries of the TOP and BOTTOM keywords are open-ended.
2. GRIDC is the control grid point and usually not connected to any element.
3. (GT<sub>i</sub>, GB<sub>i</sub>) are pairs of grid on top and bottom.
4. To each pair of (GT<sub>i</sub>, GB<sub>i</sub>) and GRIDC, MPCs are created internally to all 6 DOFs. Since the GBs always belong to dependent-DOFs, they cannot be applied to any SPC, SPC1, SPCD and SPCR.
5. Same number of grid points in TOP and BOTTOM. They should be coincident but it is not required. Users who do otherwise do so at their own risk since the current design does not consider the initial offset between them.

6. Bolt loads, including enforced motion, are usually prescribed on GRIDC to represent the pre-tension, overlap or loading of the bolt. BOLT relative displacements are given in the global coordinate system of the control node.
7. Global Coordinate System may have to be defined at the Control Node if the bolt direction is not a Basic Coordinate direction and the user wants to apply the loads along the shaft of the bolt.
8. Loads in directions other than the shaft of the bolt direction are possible.
9. The internally written MPC relationship is of the form:

$$u_{GB} = u_{GT} + u_{GC}$$

10. In 3D Contact analysis, it replaces GBi (Bottom bolt segment) by GTi (Top bolt segment) on the internally generated contact surface, which makes contact surface continuous across the mesh split between them.
11. Force output is obtained through Case Control MPCFORCE.
12. The internal MPCs generated by the BOLT entry are valid for small rotations only.
13. The BOLT entry must be defined in the residual structure only.
14. Sufficient boundary conditions must be placed on the BOLT to prevent mechanisms.
15. In contact analysis, care should be taken that the preload displacement does not exceed the local element width.
16. Pretension of a BOLT is a three step process in SOL 400:

SUBCASE 1

STEP 1

- LOAD=n Applies a pretension load to the bolt on the control grid.
- SPC=m If any required to prevent motion of structure do not include the control grid.

STEP 2

- LOAD=p1 Load entry pointing to a SPCR (relative value of 0.0) and any other desired type of loading.
- SPC=q1 SPC or SPC1 includes control grid and any other desired SPC requirements.

STEP 3

- LOAD=p2 Load entry pointing to a SPCR and any other additional desired type of loading.
- SPC=q2 SPC or SPC1 includes control grid and any other additional desired SPC requirements.

17. The BOLT may also be used in SOL101 and SOL103. In this case PARAM, AUTOMSET, YES is recommended.

## BOLT1 Bolt Model

Defines an easy-to-use pre-tension bolt model that supports nonlinear analysis and large rotation. BOLT1 allows bolt forces or displacements to be applied at a layer of elements along a cross-section in the bolt, thus modeling the effect of bolt pre-tension without the need for duplicating grid points or splitting the bolt finite element mesh into top and bottom sections around the bolt cross-section. The bolt's local coordinate system and axial direction are continuously updated to track bolt rotation. Used in SOL 400 with advanced nonlinear elements only.

Format:

1	2	3	4	5	6	7	8	9	10
BOLT1	ID	GRIDC	FORM	N1	N2	N3	OFFSET	IDTYPE	
	"ELEM"	E1	E2	E3	E4	E5	E6	E7	
		E8	E9	etc.					
	"GRID"	G1	G2	G3	G4	G5	G6	G7	
		G8	G9	etc.					

Example: Simple Method using FORM = 0

BOLT1	1	10							
	ELEM	1	2	3	4	5	6		

Example: Cross-section method using FORM = 1

BOLT1	2	20	1	0.0	0.0	1.0		SET	
	ELEM	11							
	GRID	12							

Describer	Meaning
ID	Unique bolt ID. (Integer; Required; no Default)
GRIDC	Control grid ID where pre-tension loads or tightening displacements are applied. (Integer; Required; no Default)
FORM	Specify input format for bolt definition. ((Integer; Default = 0) 0: Simple method, enter the list of elements making the bolt, no need to enter bolt axial direction or grid points forming the bolt cross-section. 1: Cross-section method, enter vector defining bolt axis, enter the list of grid points forming the bolt cross-section and list of elements sharing cross-section grid points and lying on the opposite side of the normal vector.
Ni	Components of a vector defining bolt axis (direction normal to cross-section). (Real; Required only if FORM = 1)

Descriptor	Meaning
OFFSET	Cross-section offset in axial direction (fraction of bolt length). (Real; Used for FORM=0 only; Default = 0.0)
IDTYPE	Method for defining a list of element and grid IDs. (Character; Default = "LIST" or blank)  "LIST": Enter the character string LIST, or leave blank, to allow entering lists of element IDs $E_i$ and grid point IDs $G_i$ .  "SET": Enter the character string SET to allow entering lists of element and grid points as sets. E1 and G1 refer to SET3 entries with DES = "ELEM" and "GRID", respectively.
ELEM	Enter the character string ELEM to define a list of elements. (Character; Required)
$E_i$	Element IDs (Integer; no Default)  If FORM = 0, list of element IDs of the bolt (must enter at least 2 elements along the axis of the bolt).  If FORM = 1, list of element IDs sharing grid points in the bolt cross-section and lying on the opposite side of the cross-section normal vector.  IF IDTYPE = "SET", E1 is the ID of a SET3 entry with DES = "ELEM" used to define a list of element IDs other $E_i$ entries will be ignored.
GRID	Enter the character string GRID to define a list of grid points. (Character; Required only if FORM = 1)
$G_i$	Grid IDs for the bolt cross-section. (Integer; Required only if FORM = 1; no Default)  IF IDTYPE = "SET", G1 is the ID of a SET3 entry with DES = "GRID", other $G_i$ entries will be ignored.

#### Remarks:

1. The bolt control grid point GRIDC is unique to each bolt and should be used only to apply bolt pre-tension forces or enforced displacements. GRIDC is a special grid point that has a single degree-of-freedom only aligned along the bolt axial direction and should not be connected to any element or grid in the model. GRIDC should not have an associated displacement coordinate system CD defined on the GRID entry. Applied loads and displacements should always be along the first degree-of-freedom (T1) of the control grid.
2. For the cross-section input format of the bolt definition, FORM=1, user must enter the initial direction of the bolt axis, or bolt cross-section normal vector,  $N_i$ . User must give the list of grid points defining the bolt cross-section, where pre-tension is applied, using GRID. User must also specify the list of elements sharing grid points in the bolt cross-section and lying on the opposite side of the cross-section normal vector using ELEM.

3. Using the simple input format of bolt definition, FORM=0, the complete list of bolt elements is specified using ELEM. The bolt axial direction is automatically computed by the program to be along the principal axis of the bolt elements. The bolt cross-section is also internally selected by the program to pass by the bolt center of mass by default. User can move the position of the bolt cross-section along the bolt axis away from the center of mass using the OFFSET parameter if needed.
4. For FORM=1, the cross-section normal vector cannot be of zero length.
5. For FORM=0, the number of elements entered must be at least 2 in order to be able to compute the bolt axis. In addition, bolt elements are not allowed to have a negative Jacobian in this case. The order of element connectivity must follow element definition rules in order to correctly compute the bolt axis.
6. The method for entering bolt element and grid IDs under ELEM and GRID can be controlled using IDTYPE either as list of IDs or defining a list on SET3 entries.
7. BOLT1 is supported for advanced nonlinear elements in SOL 400 only. As such, nonlinear property extensions should be defined for bolt elements. The program will attempt to internally map the element properties used by BOLT1 elements to advanced nonlinear elements if possible. Alternatively, MDLPPRM,BOLTCNEL,1 can be used to search for all elements connected to BOLT1 elements and map their element properties. Automatic mapping is turned off if the user has set NLMOPTS,SPROPMAP,-1 in the Bulk Data section. In this case nonlinear property extensions must be defined for bolt elements.
8. Current limitations:
  - BOLT1 is currently supported for 3D solid elements only.
  - BOLT1 does not support Herrmann elements.
  - GPFORCE output for the control grid does not show contributions of cross-section elements.

## BOUTPUT

### Output for Slideline Contact

Defines secondary nodes at which output is requested.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
BOUTPUT	ID	G1	G2	G3	G4	G5	G6	G7	

Alternate Format:

BOUTPUT	ID	G1	“THRU”	G2	“BY”	INC			
---------	----	----	--------	----	------	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	G8	G9	G10	G11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	G8	“THRU”	G9	“BY”	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BOUTPUT	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Format and Example Using “ALL” (No continuation entry is allowed):

BOUTPUT	ID	ALL							
BOUTPUT	15	ALL							

Describer	Meaning
ID	Contact region identification number of a BCONP entry for which output is desired, or the contact Grid ID, in 3D contact. (Integer > 0)
Gi	Secondary node numbers for which output is desired. (Integer > 0)
INC	Grid point identification number increment. See Remark 1. (Integer or blank)



**Remark:**

1. For automatic generation of grid numbers, the default increment value is 1 if grid numbers are increasing or -1 if grid numbers are decreasing (i.e., the user need not specify BY and the increment value).

## BRKPROP

## Specifies Property of Brake System for Brake Squeal Analysis Using SOL 400

Defines property of brake system for brake squeal calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BRKPROP	ID		Param_i	Value	Param_i	Value	Param_i	Value	
	Param_i	Value	Param_i	Value					

Example:

1	2	3	4	5	6	7	8	9	10
BRKPROP	28		AVSTIF	1.0E7	FRICT	0.25			

Describer	Meaning
ID	Identification ID of the Brake Property referred by the entry card <a href="#">BRKSYS</a>
AVSTIF	Approximate average stiffness per unit area between the two surfaces. AVSTIF is used as penalty contact stiffness, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; Required if TAVSTIF=0).
TAVSTIF	Table ID of a TABL3Di providing temperature or spatial variation of the average stiffness (Integer; Default = 0 which means AVSTIF should be defined).
FRICT	Enter the friction coefficient. If 0, the friction coefficient is obtained from the CONTACT TABLE option.

Remarks:

1. When TAVSTIF is 0, AVSTIF should be defined. Otherwise, Nastran will issue a fatal message and stop.

**BRKSQL****Specifies Data for Brake Squeal Calculations In SOL 600**

Specifies data for brake squeal calculations in SOL 600 only. Obsolete, please see [BSQUEAL](#) for an enhanced version.

Format:

1	2	3	4	5	6	7	8	9	10
BRKSQL	METH	AVSTIF				GLUE	ICORD		
	R1	R2	R3	X	Y	Z			
	NASCMD								
	RCFILE								

Example:

BRKSQL	1	5.34E6							
	0.0	0.0	1.0	2.0	3.0	4.0			
	tran								
	nastb								

Describer	Meaning
METH	Method flag corresponding to the type of brake squeal calculations to be performed. (Integer; Default = 1) 0 = Performs brake squeal calculations before any nonlinear analysis has taken place (corresponds to Marc feature, 4302). 1 = Performs brake squeal calculations after all nonlinear load cases (corresponds to Marc feature, 4304). -1 = Same as ID=0 except it corresponds to Marc feature, 4301 (not recommended).
AVSTIF	Approximate average stiffness per unit area between the pads and disk. Corresponds to Marc's PARAMETERS fifth datablock, field 1. This value is also known as the initial friction stiffness in Marc Volume C documentation. AVSTIF is used as a penalty contact stiffness for brake squeal, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default. Required field)
GLUE	Flag specifying whether MPC for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC; a value of 1 means include the MPCs. See Remark 6. (Integer; Default = 1)

Describer	Meaning
ICORD	Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula $C_{new} = C_{orig} + D_{efl}$ where $C_{new}$ are updated coordinates, $C_{orig}$ are original coordinates, and $D_{efl}$ are the final displacements from last Marc increment. (Integer; Default = 0)
R1	X direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)
R2	Y direction cosine (basic coord system) of axis of rotation; corresponds to Marc ROTATION A second datablock
R3	Z direction cosine (basic coord system); corresponds to Marc ROTATION A second datablock. (Real; no Default. Required field)
X	X coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
Y	Y coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
Z	Z coordinate in basic coord system of a point on the axis of rotation; corresponds to Marc ROTATION A third datablock. (Real; no Default. Required field)
NASCMD	Name of a command to run Nastran (limited to 64 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. The string will be converted to lower case. See Remark 2. (Character; Default=nastran)
RCFILE	Name of an RC file to be used with a secondary Nastran job (limited to 8 characters) -- used in conjunction with the CONTINUE options on the SOL 600 entry. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. See Remark 2. (Character; Default=nastb.rc)

#### Remarks:

1. This entry is used to calculate complex eigenvalues for brake squeal using unsymmetric stiffness friction matrices calculated by Marc. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
2. SOL 600 performs brake squeal calculations, using the following approach. The main (original) Nastran job with input file `jid.dat` or `jid.bdf` spawns Marc just as it does for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices that are saved on a file (`jid.marc.bde` with associated file `jid.marc.ccc`). The primary Nastran job then creates input data for a second Nastran job (`jid.nast.dat`) to use the unsymmetric stiffness matrices in a complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in `jid.nast.f06`, `jid.nast.op2`, etc.

NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case.

RCFILE is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and `batch=no` should also be specified for Linux systems. RCFILE is limited to 8 characters and an extension of “.rc” will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.

3. MPCs are produced for contact surfaces with glued contact. DMIGs are produced for contact surfaces without glued contact. The brakes and drums should not use glued contact; other regions of the structure can use glued contact.
4. The continuation lines may be omitted if defaults are appropriate.
5. When a BRKSQL entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
6. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPCs are output. The GLUE flag signals SOL 600 to look for these MPCs and combine them with other MPCs that might be in the model using MPCADD, or if no MPCs were originally used to add the MPCs due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPCs for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPCs with only one degree-of-freedom defined. Such MPCs will be ignored; otherwise Nastran will generate a fatal error.
7. If METH=1, a Marc t19 file will be produced.
8. The names NASCMD and RCFILE must be entered in small fixed field and start in column 9 (i.e., left justified in the field).
9. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
10. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMARC when using the BRKSQL entry.

**BRKSYS**

Specifies Data for Brake System for Brake Squeal Analysis Using SOL 400

Defines data for brake system for brake squeal calculations involving one wheel or multiple wheels with one axis or multi-axes.

Format:

1	2	3	4	5	6	7	8	9	10
BRKSYS	ID	OMETH	IVEC	BSONLY	ISLIDEBS				
	Disk1	BD1_ID	BD2_ID	MT1_ID	MT2_ID	BProp_ID			
	Disk2	BD1_ID	BD2_ID	MT1_ID	MT2_ID	BProp_ID			
	Diski	BD1_ID	BD2_ID	MT1_ID	MT2_ID	BProp_ID			

Example:

1	2	3	4	5	6	7	8	9	10
BRKSYS	900	0.5	0	YES					
	INNER	8	9	18	19	28			
	OUTER	8	9	18	19	28			

Describer	Meaning
-----------	---------

- ID Identification ID of number of a corresponding BSQUEAL Case Control command
- OMETH Specifies the corresponding load factor (or time step) where the brake squeal analysis is to be performed. (Real; Default = 0.0)
- BSONLY Brake-Squeal-Only flag to control whether or not to continue nonlinear iterations after brake squeal analysis is performed. (Character; Default = YES)
  - YES: Means to perform brake squeal analysis only and exit nonlinear iteration immediately
  - NO: means to continue nonlinear analysis.
- IVEC Flag specifying whether friction vector at center of primary contact body is in the same direction as that of the secondary nodes; See [Figure 9-3](#). (Integer; Default = 0)
  - 0: In same direction
  - 1: In tangential direction
- ISLIDEBS Flag indicating whether contact status is treated as sliding or general static contact for brake disk/pad pairs. (Integer; Default = 0)
  - 0: Sliding contact for brake disk/pad pairs
  - 1: General static contact
- Disk<sub>i</sub> Name of the i<sup>th</sup> brake system
- BD1\_ID ID of **BCBODY1** of the first body which is defined in this brake system
- BD2\_ID ID of **BCBODY1** of the second body which is defined in this brake system



Descriptor	Meaning
MT1_ID	ID of <b>MOTION</b> which is defined for the first body with BD1_ID
MT2_ID	ID of <b>MOTION</b> which is defined for the second body with BD2_ID. If blank, MT1_ID will be used
BProp_ID	ID of <b>BRKPROP</b> which defines the property of this brake system

**Remark:**

- When modules are present, multiple BRKSYS entries with the same ID can be put in different modules and selected by BSQUEAL Case Control command. In addition with this, MDBKSYS in module 0 with the same ID can also be selected.  
MSC Nastran only uses one set of fields in the first line of those entries (fields OMETH, IVEC, BSONLY and ISLIDEBS).

- If not all the sets are identical:
  - When MDBKSYS is selected, its fields will be used.
  - If MDBKSYS doesn't exist or isn't selected and all the sets of those fields of BRKSYS entries are not identical, use the default values.

If all the sets are identical, use the specified values.

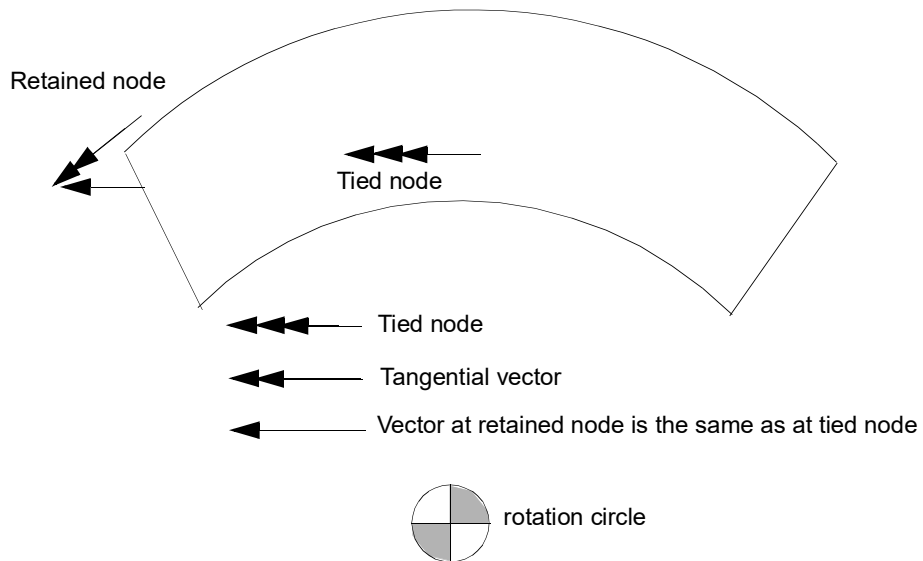


Figure 9-3 Vectors of frictional stiffness at tied and retained node.

## BSET Fixed Analysis Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

BSET	2	135	14	6					
------	---	-----	----	---	--	--	--	--	--

Describer	Meaning
ID <sub>i</sub>	Grid or scalar point identification number. (Integer > 0)
C <sub>i</sub>	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. BSET and BNDFIX entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSET<sub>i</sub>/BNDFIX<sub>i</sub> or CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSET<sub>i</sub>/BNDFIX<sub>i</sub> entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present or both BSET<sub>i</sub>/BNDFIX<sub>i</sub> and CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSET<sub>i</sub>/BNDFIX<sub>i</sub> entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.
5. SOL 400 does not support ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, SUPORT<sub>i</sub>, and QSET<sub>i</sub> except in the following situations:



- a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
- b. Linear perturbation with:
  - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
  - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
- c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## BSET1 Fixed Analysis Degrees-of-Freedom, Alternate Form of BSET Entry

Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
BSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

BSET1	2	135	14	6	23	24	25	26	
	122	127							

Alternate Format and Example:

BSET1	C	ID1	“THRU”	ID2					
BSET1	3	6	THRU	32					

Describer	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
IDI	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID1 < ID2)

Remarks:

1. BSET1 and BNDFIX1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on BSETi/BNDFIXi entries form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.
5. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:
    - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

**BSQUEAL** Specifies Data for Brake Squeal Analysis Using SOLs 400 and 600

Defines data for brake squeal calculations involving one wheel (Primary Format) or multiple wheels (Alternate Format) as well as other rotating or stationary sliding objects used in SOLs 400 and 600.

Primary Format (One Wheel, SOL 400 and SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	ID	OMETH	AVSTIF			BSONLY	IGLUE	ICORD	
	RX	RY	RZ	X	Y	Z			
	NASCMD								
	RCFILE								

Alternate Format (Multiple Wheels, SOL 600 Only):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	ID	OMETH	NGROUP	ICORD	ITYPE				
	"BODY"	ID1	ID2			AVSTIF	T(AVSTIF)		
		IVEC	IDRF1	IDRF2	IGLUE				
		RX1	RY1	RZ1	X1	Y1	Z1		
		RX2	RY2	RZ2	X2	Y2	Z2		
	"BODY"	ID1	ID2			AVSTIF	T(AVSTIF)		
		IVEC	IDRF1	IDRF2	IGLUE				
		RX1	RY1	RZ1	X1	Y1	Z1		
		RX2	RY2	RZ2	X2	Y2	Z2		
	NASCMD								
	RCFILE								

Example, Primary Format (SOL 400):

1	2	3	4	5	6	7	8	9	10
BSQUEAL	100	0.2	5.34E6			NO			
	0.0	0.0	1.0	2.0	3.0	4.0			

Example, Primary Format (SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL		1.0	5.34E6						
	0.0	0.0	1.0	2.0	3.0	4.0			
	nastran								
	nastb.rc								



## Example, Alternate Format (SOL 600):

1	2	3	4	5	6	7	8	9	10
BSQUEAL		1.0	2	1	1				
	BODY	2	3			60000.	0		
		0			1				
		0.0	0.0	1.0	10.0	10.0	25.0		
		0.0	0.0	-1.0	10.0	10.0	30.0		
	BODY	6	8			1.0	5		
		0	110	120	1				
	nastran								
	nastb.rc								

Describer	Meaning
ID	Identification number of a corresponding BSQUEAL Case Control command (Integer > 0). Ignored by SOL 600.
OMETH	Specifies the corresponding load factor (or time step) where the brake squeal analysis is to be performed. (Real; Default = 0.0 in SOL 400, see Remark 5., for SOL 600 the value must be 0.0 or 1.0, Remark 11.)
NGROUP	Number of pairs of contact bodies (Integer > 0; Default = 1)
ICORD (SOL 600 Only)	Flag indicating whether coordinates are updated or not. A value of 0 means coordinates are not updated. A value of 1 means coordinates are updated using the formula $C_{new} = C_{orig} + Defl$ where $C_{new}$ are updated coordinates, $C_{orig}$ are original coordinates, $Defl$ are the final displacements from last Marc increment. (Integer; Default = 0)
ITYPE (SOL 600 Only)	Flag determining type of analysis. See Figure 9-4 and Figure 9-5 (Integer; Default = 1) 1 Brake squeal 2 Complex engenvalue analysis following general sliding contact analysis.
BSONLY (SOL 400 Only)	Brake-Squeal-Only flag to control whether or not to continue nonlinear iterations after brake squeal analysis is performed. BSONLY=YES means to perform brake squeal analysis only and exit nonlinear iteration immediately; BSONLY=NO means to continue nonlinear analysis. (Character; Default = YES)
“BODY”	Character string that signals the start of a pair of contact bodies for which instability calculations are to be made (Character; Required)
ID1	BCBODY ID of the first contact body. (Integer > 0; no Default; bcbody id1 must exist)
ID2	BCBODY ID of the second contact body. (Integer > 0; no Default; bcbody id2 must exist)

Descriptor	Meaning
AVSTIF	Approximate average stiffness per unit area between the two surfaces. AVSTIF is used a penalty contact stiffness, it needs to be a large value but not so large that numerical instabilities result. If AVSTIF is large enough, increasing it by a few orders of magnitude will not appreciably affect the squeal modes. (Real; no Default; Required field)
T(AVSTIF)	Table ID of a TABL3Di providing temperature or spatial variation of the average stiffness (Integer; Default = 0 which means FC is constant)
IVEC	Flag specifying whether friction vector at center of primary contact body is in the same direction as that of the secondary nodes; see <a href="#">Figure 9-4</a> . (Integer; Default = 0)
	0            In same direction
	1            In tangential direction
IDRFi (SOL 600 Only)	ID of a matching RFORCE entry (if any) for body 1 or 2. (Integer; Default = 0, which means this body is not rotating)
IGLUE (SOL 600 Only)	Flag specifying whether MPC's for non-pad/disk surfaces with glued contact are used or ignored. A value of 0 means ignore the MPC's, a value of 1 means include the MPC's. See <a href="#">Remarks 12</a> and <a href="#">14</a> . (Integer; Default = 1)
RXi	X direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
RYi	Y direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
RZi	Z direction cosine (basic coord system) of axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
Xi	X coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
Yi	Y coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
Zi	Z coordinate in basic coord system of a point on the axis of rotation for body i if IDRFi is zero or blank (Real; or blank)
NASCMD (SOL 600 Only)	Name of a command to run Nastran (limited to 64 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. The full path of the command to execute Nastran should be entered. Enter the string entirely in lower case. (Character; Default = nastran) If the string exceeds 8 characters, do not enter this string using free-field also see <a href="#">Remark 9</a> .
RCFILE (SOL 600 Only)	Name of a RC file to be used with a secondary Nastran job (limited to 8 characters) - used in conjunction with the CONTINUE options on the SOL 600 statement. An extension of ".rc" will automatically be added. Enter the string entirely in lower case. (Character; Default=nastb.rc) If the string exceeds 8 characters, do not enter this string using free-field also see <a href="#">Remark 9</a> .

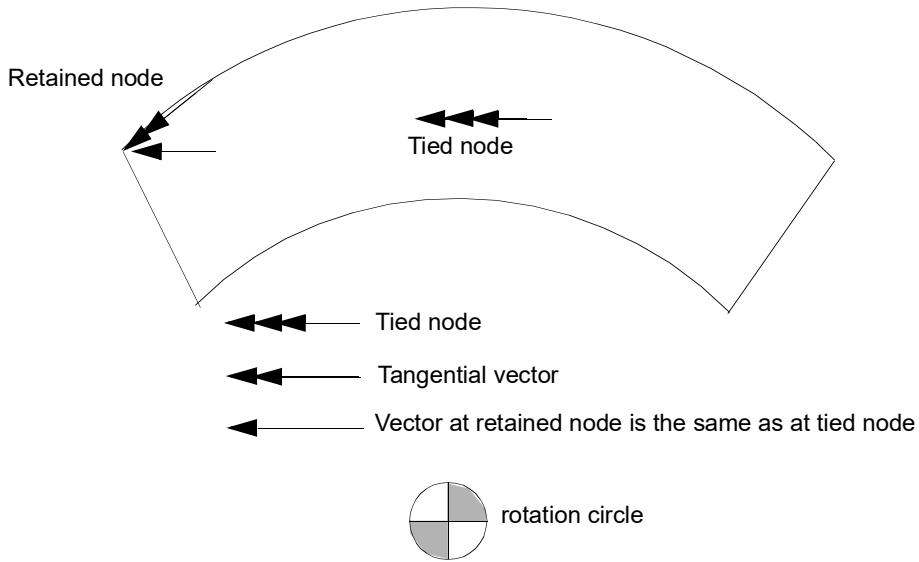


Figure 9-4 Vectors of frictional stiffness at tied and retained node.

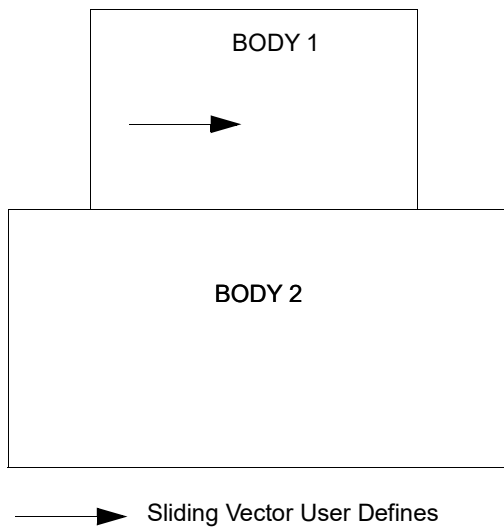


Figure 9-5 Sliding model

Remarks:

1. This entry is used to perform multiple-body brake squeal analysis or other contact with friction sets of bodies using unsymmetric stiffness friction matrices.

2. One or more pairs of bodies may be specified. Each pair of bodies is described by the four lines beginning with the header "BODY". For example, there may be one rotating body, such as brake disk or rotor, and one contact body or bodies, such as brake pad under each "BODY" keyword. There may be other bodies or parts in the assembly.
3. The disks should not be glued with pads. When bodies are intended to be glued, turn on BCTABLE / IGLUE or BCONPRG/IGLUE for those contact bodies.
4. Entries ICORD, ITYPE, IDBFi, IGLUE, NASCMD and RCFILE are used by SOL 600 only.
5. Entry BSONLY is used by SOL 400 only.
6. Brake squeal is not available with segment-to-segment contact.

### SOL 400 Only

7. For nonlinear static analysis,  $0.0 < \text{OMETH} < 1.0$ , OMETH is overridden by the load factor from Case Control command, NLIC or its default, when a brake squeal analysis is performed in a separate SUBCASE-STEP other than ANALYSIS = NLSTATIC.
8. User subroutine UBSQUEAL is available using the BCONUDS entry.

### SOL 600 Only

9. SOL 600 performs brake squeal calculations as follows. The main (original) Nastran job with input deck jid.dat or jid.bdf spawns Marc just as done for any other SOL 600 job. Marc calculates unsymmetric friction stiffness matrices which are saved on a file (jid.marc.bde with associated file jid.marc.ccc). The primary Nastran job then creates input data for a second Nastran job (jid.nast.dat) to use the unsymmetric stiffness matrices in an complex eigenvalue extraction. The primary Nastran job spawns a second Nastran job to calculate the complex eigenvalues. The complex eigenvalues and eigenvectors are found in jid.nast.f06, jid.nast.op2, etc.  
 NASCMD is the name of the command to execute the secondary Nastran job. NASCMD can be up to 64 characters long and must be left justified in field 2. The sting as entered will be used as is - except that it will be converted to lower case regardless of whether it is entered in upper or lower case. RCFILE is the name of an RC file to be used for the secondary Nastran job. Normally it should be similar to the RC file used for the primary run except that additional memory will normally be necessary to calculate the complex eigenvalues and batch=no should also be specified. RCFILE is limited to 8 characters and an extension of ".rc" will be added automatically. This entry will be converted to upper case in Nastran but will be converted to lower case before spawning the complex eigenvalue run. This RC file must be located in the same directory as the Nastran input file. This entry is the same as specifying PARAM,MRRCFILE. Only one or the other should be used.
10. OMETH can only be one of the following two values in SOL 600.  
 0.0=Perform brake squeal calculations before any nonlinear analysis has taken place  
 1.0=Perform brake squeal calculations after all nonlinear load cases
11. Options exist to obtain the unsymmetric stiffness matrices using the undeformed geometry (initial contact) or after all specified nonlinear subcases.
12. MPC's are produced for contact surfaces with glued contact. DMIG's are produced for contact surfaces without glued contact. The brakes and drums may not use glued contact, other regions of the structure can used glued contact.



13. When the BSQUEAL entry is used, PARAM,MRMTXNAM and PARAM,MARCFIL1 should not be entered.
14. When brake squeal matrices are output by Marc, unsymmetric friction stiffness matrices are output for non-glued contact surfaces. For surfaces with glued contact, MPC's are output. The GLUE flag signals SOL 600 to look for these MPC's and combine them with other MPC's that might be in the model using MPCADD, or if no MPC's were originally used to add the MCP's. due to glued contact. Glued contact surfaces may not be used for the disk-rotor interface. If IGLUE is zero or blank, the MPC's for glued contact in the Marc brake squeal bde file (if any) will be ignored. Sometimes, Marc puts out MPC's with only one degree-of-freedom defined. Such MPC's will be ignored otherwise Nastran will generate a fatal error.
15. The names NASCMD and RCFILE must be entered in small fixed field and start in column 9 (i.e. left justified in the field).
16. The Nastran input file name used for a brake squeal analysis may only contain lower case letters and the underscore and/or dash characters.
17. Brake squeal is not available with DDM (parallel processing). Do not enter a PARAMARC when using the BSQUEAL entry.
18. The Bulk Data entry BRKSQL is the alternate format in SOL 600 when there is only one body.

## BSURF

## Contact Body or Surface

Defines a contact body or surface by Element IDs.

Format:

1	2	3	4	5	6	7	8	9	10
BSURF	ID	ELID1	ELID2	ELID3	ELID4	ELID5	ELID6	ELID7	
	ELID8	ELID9	etc.						

Alternate Format:

BSURF	ID	ELID1	THRU	ELID2	BY	INC			
	ELID3	THRU	ELID4	BY	INC2				

Example:

BSURF	15	5	THRU	21	BY	4			
	27	30	32	33					
	35	THRU	44						
	67	68	72	75	84	93			

Describer	Meaning
ID	Identification of a deformable surface corresponding to a BSID value on the BCBODY entry. See Remark 2. (Integer > 0)
ELIDi	Element identification numbers. If the curve or surface is defined with element ids only, the direction of the normal depends on the grid point numbering. Keywords THRU and BY can be used to assist the listing. For SOL 600 and SOL 700 the Alternate Format must be used with Remarks 6. and 7.
INC	Identification number increment. See Remark 3. (Integer or blank)

Remarks:

1. BSURF can be used in SOL101, 103, 105, 107, 108, 109, 110, 111, 112, 400, 600, and 700.
2. ID must be unique with respect to all other BSURF, BCBOX, BCPROP, and BCMATL entries.
3. For automatic generation of element IDs, the default increment value is 1 if element numbers are increasing or -1 if element numbers are decreasing (i.e., the user need not specify BY and the increment value).
4. The deformable surface may alternately be defined using BCBOX, BCPROP, or BCMATL entries.
5. Only one kind of entry (BSURF, BCBOX, BCPROP, or BCMATL) may be used to define a particular deformable surface.

6. For SOLs 600 and 700, Format 1 and the Alternate Format cannot be mixed for a particular BSURF entry.
7. For SOLs 600 and 700 if the Alternate Format is used, THRU must be in column 4 for entries with the BSURF header and in column 3 for continuation entries. If BY is not used, columns 6-9 of the BSURF entry and columns 5-9 of continuation entries must be blank.

**BWIDTH**                      Boundary Line Segment Width or Thickness

Defines widths or thicknesses for line segments in 3-D or 2-D sideline contact defined in the corresponding BLSEG Bulk Data entry for SOL 106 or SOL 129. SOL 400 general contact capability, defined with BCONTACT, BCTABL1, etc. is the recommended approach for contact analysis.

Format:

1	2	3	4	5	6	7	8	9	10
BWIDTH	ID	W1	W2	W3	W4	W5	W6	W7	

Alternate Format:

BWIDTH	ID	W1	"THRU"	W2	"BY"	INC			
--------	----	----	--------	----	------	-----	--	--	--

The continuation entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

	W8	W9	W10	W11	-etc.-				
--	----	----	-----	-----	--------	--	--	--	--

Continuation Entry Format 2:

	W8	"THRU"	W9	"BY"	INC				
--	----	--------	----	------	-----	--	--	--	--

Example:

BWIDTH	15	2.0	THRU	5.0	BY	1.0			
	2.0	2.0	2.0	2.0					
	35.	THRU	44.						
	1.5	3.4	7.6	0.4	0.7				

Descriptor	Meaning
------------	---------

ID	BLSEG entry identification number. (Integer > 0)
Wi	Width values for the corresponding line segments defined in the BLSEG entry. See Remark 1. (Real > 0.0)
INC	Width value increment. See Remark 2. (Real or blank)

Remarks:

1. BWIDTH may be omitted if the width of each segment defined in the BLSEG entry is unity. The number of widths to be specified is equal to the number of segments defined in the corresponding BLSEG entry.

2. The default value for INC is 1.0 if the width is increasing or -1.0 if the width is decreasing. That is, the user need not specify BY and the increment value. If the number of widths specified is less than the number of segments defined in the corresponding BLSEG entry, the width for the remaining segments is assumed to be equal to the last width specified.
3. If there is only one grid point in the corresponding BLSEG entry, there is no contributory area associated with the grid point. To compute correct contact stresses an area may be associated with the single grid point by specifying the area in field W1.

## Entries CA - CM

### CAABSF

### Frequency-Dependent Acoustic Absorber Element

Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CAABSF	EID	PID	G1	G2	G3	G4			

Example:

CAABSF	44	38	1	10	20				
--------	----	----	---	----	----	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number that matches a PAABSF entry. (Integer > 0; Default = EID)
Gi	Grid point identification number of fluid connection points. (Integer ≥ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If only G1 is specified then a point impedance is assumed. If G1 and G2 are specified then a line impedance is assumed. If G1, G2, and G3 are specified, then an impedance is associated with the area of the triangular face. If G1 through G4 are specified, then an impedance is associated with the quadrilateral face. See [Figure 9-6](#).
3. The CAABSF element must connect entirely to fluid points on the fluid-structure boundary.
4. This element is used only in frequency response and is ignored in all other solutions.

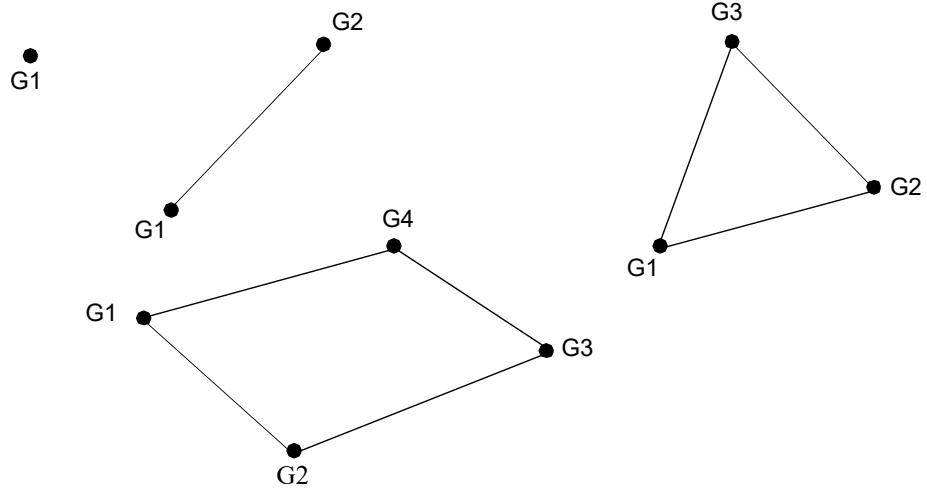


Figure 9-6 Four Types of CAABSF Elements

## CACINF3

## Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with triangular base.

Format:

1	2	3	4	5	6	7	8	9	10
CACINF3	EID	PID	G1	G2	G3				

Example:

CACINF3	111	10	1004	1008	1011				
---------	-----	----	------	------	------	--	--	--	--

Describer	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.



## CACINF4 Acoustic Conjugate Infinite Element Base Connection

Defines an acoustic conjugate infinite element with quadrilateral base.

Format:

1	2	3	4	5	6	7	8	9	10
CACINF4	EID	PID	G1	G2	G3	G4			

Example:

CACINF4	275	10	1027	1032	1056	1021			
---------	-----	----	------	------	------	------	--	--	--

Describer	Meaning
EID	Element Identification Number. (0 < Integer < 100,000,000)
PID	Property Identification Number of a PACINF entry. (Integer > 0)
Gi	Grid Point Identification Numbers of Element Base Connection Points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The element must be connected to acoustic grid points.
3. The element normal is defined by the right-hand rule. If the normal does not point into the exterior domain, the element orientation will be changed automatically, and an information message will be written to the .f06 file.

## CAERO1

## Aerodynamic Panel Element Connection

Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory for subsonic aerodynamics and the ZONA51 theory for supersonic aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO1	EID	PID	CP	NSPAN	NCHORD	LSPAN	LCHORD	IGID	
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO1	1000	1		3			2	1	
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO1 entry; used to specify associated bodies. Required even if there are no associated bodies. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
NSPAN	Number of spanwise boxes; if a positive value is given NSPAN, equal divisions are assumed; if zero or blank, a list of division points is given at LSPAN, field 7. (Integer ≥ 0)
NCHORD	Number of chordwise boxes; if a positive value is given NCHORD, equal divisions are assumed; if zero or blank, a list of division points is given at LCHORD, field 8. (Integer ≥ 0)
LSPAN	ID of an AEFAC data entry containing a list of division points for spanwise boxes. Used only if NSPAN, field 5 is zero or blank. (Integer > 0)
LCHORD	ID of an AEFAC data entry containing a list of division points for chordwise boxes. Used only if NCHORD, field 6 is zero or blank. (Integer > 0)
IGID	Interference group identification; aerodynamic elements with different IGIDs are uncoupled. (Integer > 0)
X1, Y1, Z1 X4, Y4, Z4	} Location of points 1 and 4, in coordinate system CP. (Real)
X12, X43	

Remarks:

1. The boxes and corner point nodes are numbered sequentially, beginning with EID. The user should be careful to ensure that all box and corner point node numbers are unique. There can be overlapping IDs between the structural and aerodynamic model, but MSC Patran will not then be able to display any results. Also, non-unique corner IDs are allowed, but results cannot be visualized in MSC Patran.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0. and 1. (In which case the corners of the panel would not be at the reference points.)
3. A triangular element is formed if X12 or X43=0.0
4. The element coordinate system is right-handed as shown in [Figure 9-7](#).
5. The continuation is required.
6. It is recommended that NCHORD or LCHORD be chosen so that the typical box chord length  $\Delta x$  satisfies the condition  $\Delta x < 0.08 V/f$  (recent studies indicate that  $.02 V/f$  is needed to get converged stability derivatives) where  $V$  is the minimum velocity and  $f$ , in hertz, is the maximum frequency to be analyzed (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

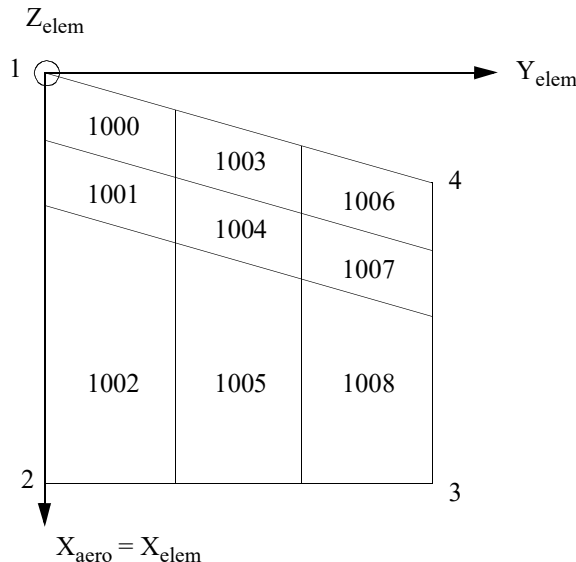


Figure 9-7 Element Coordinate System for Aerodynamic Panel

7. This entry can be used for two different aerodynamic theories: Doublet-Lattice for subsonic and ZONA51 for supersonic. The proper theory is selected based on the specification of Mach number on the MKAEROi or TRIM entry.

## CAERO2

## Aerodynamic Body Connection

Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO2	EID	PID	CP	NSB	NINT	LSB	LINT	IGID	
	X1	Y1	Z1	X12					

Example:

CAERO2	1500	2	100		4	99		1	
	-1.0	100.	-30.	175.					

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PAERO2 entry. ( $\text{Integer} > 0$ )
CP	Coordinate system for locating point 1. ( $\text{Integer} \geq 0$ ; Default = 0)
NSB	Number of slender body elements. If $\text{NSB} > 0$ , then NSB equal divisions are assumed; if zero or blank, specify a list of divisions in LSB. ( $\text{Integer} \geq 0$ )
NINT	Number of interference elements. If $\text{NINT} > 0$ , then NINT equal divisions are assumed; if zero or blank, specify a list of divisions in LINT. ( $\text{Integer} \geq 0$ )
LSB	ID of an AEFACCT Bulk Data entry for slender body division points; used only if NSB is zero or blank. ( $\text{Integer} \geq 0$ )
LINT	ID of an AEFACCT data entry containing a list of division points for interference elements; used only if NINT is zero or blank. ( $\text{Integer} \geq 0$ )
IGID	Interference group identification. Aerodynamic elements with different IGIDs are uncoupled. ( $\text{Integer} \geq 0$ )
X1, Y1, Z1	Location of point 1 in coordinate system CP. (Real)
X12	Length of body in the x-direction of the aerodynamic coordinate system. ( $\text{Real} > 0.0$ )

Remarks:

1. Point 1 is the leading point of the body.
2. All CAERO1 (panels) and CAERO2 (bodies) in the same group (IGID) will have aerodynamic interaction.
3. At least one interference element is required for the aerodynamic body specified by this entry.

4. The beams and connection points are numbered sequentially beginning with EID. The user should be careful to ensure that all aero elements and connection point IDs are unique. Overlapping IDs between structure and aerodynamic models are allowed, but will prevent results visualization in Patran.

Old rules regarding numbering among Z, ZY, Y bodies and CAERO1 no longer apply: arbitrary ordering is allowed.

5. At least two slender body elements are required for each aerodynamic body.
6. Interference elements are only intended for use with panels.
7. Determining the size of the j-set (i.e., the number of aerodynamic elements) is essential to input D1JE and D2JE matrices. Use the following expressions for locating the proper row in the two matrices:

$$\begin{aligned} J = & \text{Number of boxes} + \text{Number of I-elements, z} \\ & + 2 * (\text{Number of I-elements, zy}) \\ & + \text{Number of I-elements, y} \\ & + \text{Number of S-elements, z} \\ & + 2 * (\text{Number of S-elements, zy}) \\ & + \text{Number of S-elements, y} \end{aligned}$$

where I-elements denote interference and S-elements denote slender body.

## CAERO3

## Aerodynamic Panel Element Configuration

Defines the aerodynamic edges of a Mach Box lifting surface. If no cranks are present, this entry defines the aerodynamic Mach Box lifting surface.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO3	EID	PID	CP	LISTW	LISTC1	LISTC2			
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO3	2000	2001	0	22	33				
	1.0	0.0	0.0	100.	17.	130.	0.	100.	

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO3 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
LISTW	Identification number of an AEFACCT entry that lists (x,y) pairs for structural interpolation of the wing. (Integer > 0)
LISTC1, LISTC2	Identification number of AEFACCT entries that list (x,y) pairs for control surfaces, if they exist. (Integer ≥ 0)
X1, Y1, Z1 X4, Y4, Z4	} Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	

Remarks:

1. EID must be unique with respect to all other element identification numbers.
2. The (x,y) pairs on LISTW, LISTC1 and LISTC2 AEFACCT entries are in the aero element coordinate system (see [Figure 9-8](#)). The (x,y) pairs define a set of aerodynamic grid points that are independent of Mach number and are selected by the user to be representative of the planform and motions of interest. The (x,y) pairs must be sufficient in number and distribution such that: the surface spline provides an accurate interpolation between them and the Mach Box centers that are variously located on the planform as a function of Mach number (a complete description of the Mach Box Method is given in the [MSC Nastran Aeroelastic Analysis User's Guide](#)).

3. The (x,y) pairs are numbered sequentially, beginning with EID for LISTW, then LISTC1, and finally for LISTC2. On SPLINEi entries, the box numbers (BOX1 and BOX2 on SPLINE1, ID1 and ID2 on SPLINE2, and UKID on SPLINE3) refer to the (x,y) pair sequence number appropriate for the surface (primary, or first or second control) being splined.
4. If cranks and/or control surfaces exist, their locations are given on the PAERO3 entry.
5. The numbering system and coordinate system are shown below:

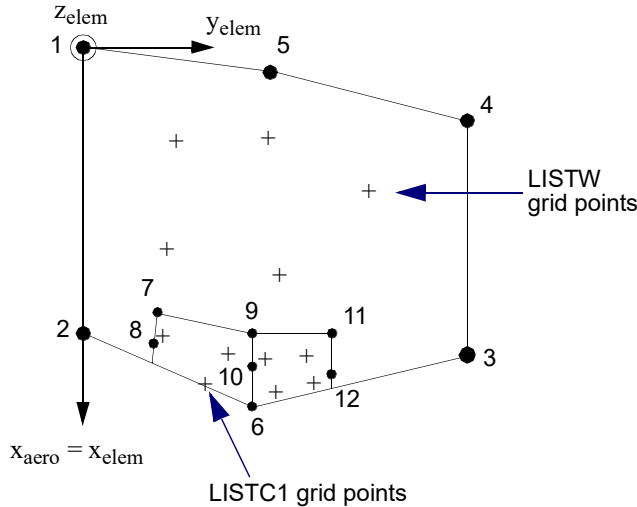


Figure 9-8 CAERO3 Element Configuration

Platform Corners		Control	
1	Leading edge, inboard	7	Hinge line, inboard
2	Trailing edge, inboard	8	On inboard edge (usually at trailing edge)
3	Trailing edge, outboard	9	Hinge line, outboard
4	Leading edge, outboard	10	On outboard edge (usually at trailing edge)
Cranks		Control (if two)	
5	Leading edge	9	Hinge line, inboard
6	Trailing edge	10	On inboard edge (usually at trailing edge)
		11	Hinge line, outboard
		12	On outboard edge (usually at trailing edge)

6. The CAERO3 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO3.

## CAERO4 Aerodynamic Macro-Strip Element Connection

Defines an aerodynamic macro element for Strip theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO4	EID	PID	CP	NSPAN	LSPAN				
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO4	6000	6001	100		315				
	0.0	0.0	0.0	1.0	0.2	1.0	0.0	0.8	

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PAERO4 entry. ( $\text{Integer} > 0$ )
CP	Coordinate system for locating points 1 and 4. ( $\text{Integer} \geq 0$ ; Default = 0)
NSPAN	Number of strips; if a positive value is given, NSPAN equal strips are assumed. If zero or blank, LSPAN must be specified. ( $\text{Integer} \geq 0$ )
LSPAN	ID of an AEFACCT entry containing a list of division points for strips. Used only if NSPAN is zero or blank. ( $\text{Integer} > 0$ )
X1, Y1, Z1 X4, Y4, Z4	Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and greater than structural grid, scalar, and extra point IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN = 3, the division points are 0.0, 0.333, 0.667, and 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.



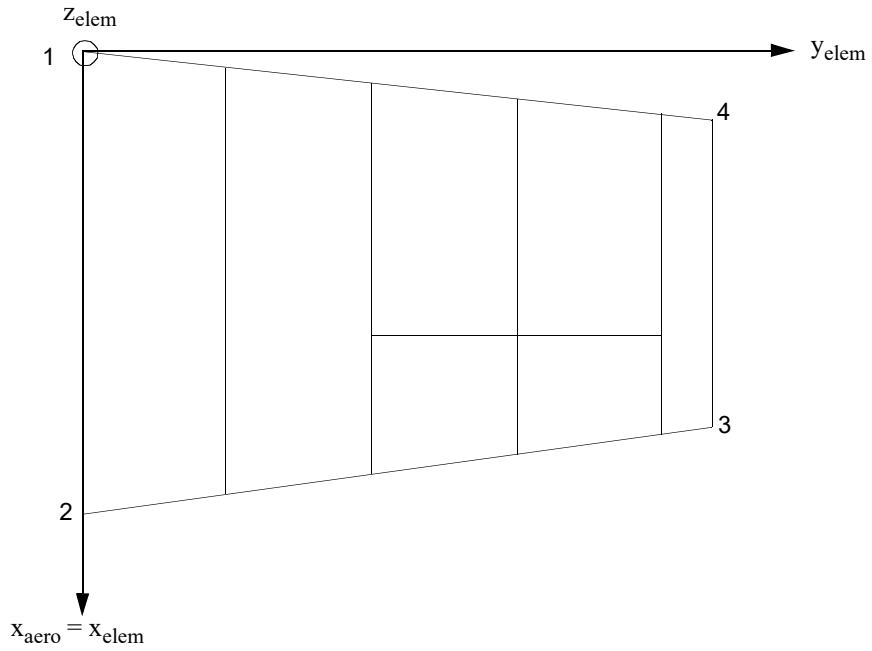


Figure 9-9 CAERO4 Element Connection

4. The CAERO4 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO4.

## CAERO5

## Aerodynamic Panel Element Configuration

Defines an aerodynamic macro element for Piston theory.

Format:

1	2	3	4	5	6	7	8	9	10
CAERO5	EID	PID	CP	NSPAN	LSPAN	NTHRY	NTHICK		
	X1	Y1	Z1	X12	X4	Y4	Z4	X43	

Example:

CAERO5	6000	6001	100		315	0	0		
	0.0	0.0	0.0	1.0	0.2	1.0	0.	0.8	

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PAERO5 entry. (Integer > 0)
CP	Coordinate system for locating points 1 and 4. (Integer ≥ 0; Default = 0)
NSPAN	Number of strips. (Integer. If a positive value is given, equal strips are assumed. If zero or blank, then LSPAN must be specified.)
LSPAN	ID of an AEFAC T entry containing a list of division points for strips. Used only if NSPAN is zero or blank. (Integer > 0)
NTHRY	Parameter to select Piston or van Dyke's theory. (Integer = 0, 1, or 2; Default = 0) Blank or 0   Piston theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ 1            van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with no sweep correction (secΛ = 1.0). 2            van Dyke's theory is used to compute $\bar{C}_1$ and $\bar{C}_2$ with a sweep correction based on the actual Λ.
NTHICK	Parameter to select thickness integrals input. (Integer ≥ 0; Default = 0) Blank or 0   Thickness integrals are computed internally. > 0           Thickness integrals are input directly and are the ID number of an AEFAC T entry that lists the <i>I</i> and/or <i>J</i> integrals.
X1,Y1, Z1 X4,Y4, Z4	} Location of points 1 and 4 in coordinate system CP. (Real)
X12, X43	Edge chord lengths in aerodynamic coordinate system. (Real ≥ 0; X12 and X43 cannot both be zero.)

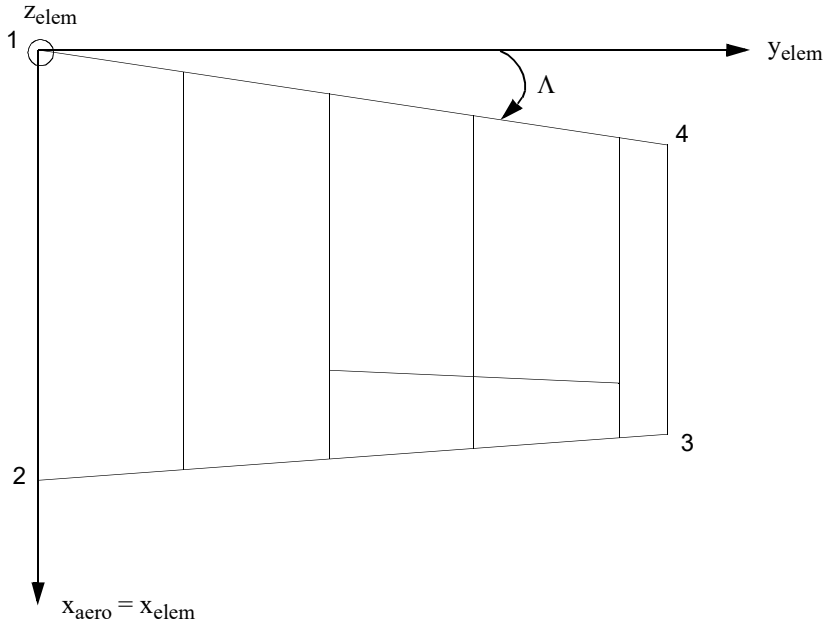


Figure 9-10 CAER05 Element Configuration

Remarks:

1. The strips are numbered sequentially, beginning with EID. The user must ensure that all strip numbers are unique and different from structural grid IDs.
2. The number of division points is one greater than the number of boxes. Thus, if NSPAN=3, the division points are 0.0, 0.333, 0.667, 1.000. If the user supplies division points, the first and last points need not be 0.0 and 1.0 (in which case the corners of the panel would not be at the reference points).
3. A triangular element is formed if X12 or X43 = 0.0.

$$\bar{C}_1 = m / (m^2 - \sec^2 \Lambda)^{1/2}$$

$$\bar{C}_2 = [m^4 (\gamma + 1) - 4 \sec^2 \Lambda (m^2 - \sec^2 \Lambda)] / [4 (m^2 - \sec^2 \Lambda)^2]$$

where:

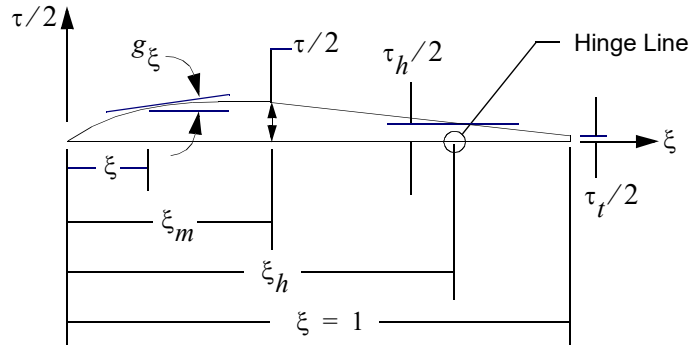
- m = Mach number
- $\gamma$  = Specific heat ratio
- $\Lambda$  = Leading edge sweep angle

When  $\sec \Lambda = 0.0$ , Piston theory coefficients are obtained (NTHRY = 1)

When  $\sec \Lambda = 1.0$ , van Dyke's coefficients are obtained (NTHRY = blank or 0)

When  $\sec \Lambda \neq 0.0$  or  $\neq 1.0$ , sweep corrections are included (NTHRY = 2)

5.  $I$  and  $J$  thickness integral definitions:



$g_\xi \equiv \frac{dg}{d\xi}$  = slope of airfoil semithickness

$$\begin{aligned}
 I_1 &= \int_0^1 g_\xi d\xi & J_1 &= \int_{\xi_h}^1 g_\xi d\xi \\
 I_2 &= \int_0^1 \xi g_\xi d\xi & J_2 &= \int_{\xi_h}^1 \xi g_\xi d\xi \\
 I_3 &= \int_0^1 \xi^2 g_\xi d\xi & J_3 &= \int_{\xi_h}^1 \xi^2 g_\xi d\xi \\
 I_4 &= \int_0^1 g_\xi^2 d\xi & J_4 &= \int_{\xi_h}^1 g_\xi^2 d\xi
 \end{aligned}$$

$$I_5 = \int_0^1 \xi g_\xi^2 d\xi \quad J_5 = \int_{\xi_h}^1 \xi g_\xi^2 d\xi$$

$$I_6 = \int_0^1 \xi^2 g_\xi^2 d\xi \quad J_6 = \int_{\xi_h}^1 \xi^2 g_\xi^2 d\xi$$

Figure 9-11 CAERO5  $I$  and  $J$  Thickness Integral Definitions

- The CAERO5 entry is only supported in SOL 145, SOL 200 with ANALYSIS=FLUT and in SOL 146 with mechanical loads. Gust response in SOL 146 is not supported for the CAERO5.

## CAMPBLL

## Campbell Diagram Parameters

Specifies the parameters for Campbell Diagram generation and mode tracking analysis.

Formats:

1	2	3	4	5	6	7	8	9	10
CAMPBLL	CID	VPARM	DDVALID	TYPE					
	MODTRK	CORU	SWITR	NUMMOD	PRTCOR				

Examples:

CAMPBLL	15	SPEED	22	RPM					
	1	0.75	1		1				

Describer	Meaning
CID	Identification number of entry (Integer > 0; Required).
VPARM	Variable parameter, allowable entry is: 'SPEED'. 'SPEED', reference rotor speed will be varied (rotordynamic option only).
DDVALID	Identification number of DDVAL entry that specifies the values for the variable parameter (Integer > 0; Required).
TYPE	Allowable entries are: 'FREQ' and 'RPM'.
MODTRK	Perform mode tracking using either numerical approach (default) or eigenvector based approach or both. (Default = 0). See remark 4.
CORU	Threshold for mode correlation.
SWITR	Option to use updated mode for mode tracking in case of failure. See remark 7.
NUMMODE	Number of modes to track (Should be less than $0.5 * Neig$ , where $Neig$ is the number of eigenvalues extracted in EIGC). See remark 4.
PRTCOR	Option to print correlation matrix at each rotor speed (Default = 0). See remark 9.

Remarks:

1. CAMPBLL option is supported for SOL 107, SOL110, SOL200 and SOL400 analysis.
2. CAMPBLL option is supported in SOL 200 and SOL 400 for ANALYSIS = DCEIG/MCEIG.
3. Only the modes with positive imaginary part are used for mode tracking (to avoid repeated eigenvalues.)
4. Following methods are available in MSC Nastran for mode tracking:
  - a. Numerical Mode tracking based on second derivative.

Obtain second derivative for the variation of eigenvalue with rotor speed and determine the eigenfrequency with minimum slope.

- b. Mode tracking based on orthogonality of left and right eigenvectors.

This feature is available for analysis in fixed reference frame only.

MODTRK = 0: Perform numerical mode tracking only. (Default)

MODTRK = 1: Perform BOTH numerical and eigenvector based mode tracking.

MODTRK = 2: Perform eigenvector based mode tracking only.

- 5. The variables specified in the second row are relevant for eigenvector-based mode tracking only.
- 6. Ideally, for tracked pair of modes, the value of Normalized Cross Complex Orthogonality (NC2O) parameter is very close to 1. However, for complex models involving large stator component and dynamic reduction, this value can be significantly lower. Thus, the default value for CORU is set to 0.7. Based on the model complexity and NC2O matrix obtained for two rotor speeds, user may change this value.
- 7. In case of mode tracking failure at a particular rotor speed, user has the option to continue mode tracking for rest of the rotors speed by:
  - SWITR = 1: Updating modes corresponding to rotor speed with failed mode tracking, or
  - SWITR = 0: Using the modes from last rotor speed with successful mode tracking. (Default)

In case the Campbell diagram includes multiple mode switching and there are new modes entering the analysis, use of SWITR=1 is recommended.
- 8. Only the complex eigenvalues with positive imaginary part are considered for mode tracking analysis. As a result, the number of eigenvalues available for mode tracking may not be same at all the rotor speeds considered in the analysis. This may lead to mode tracking failure. One way to avoid this issue is to pick NUMMODE to be less than half of the number of complex eigenvalues extracted using EIGC bulk data entry. Thus, the analysis uses only the first NUMMODE frequencies with positive imaginary part for mode tracking at each rotor speed.
- 9. When PRTCOR parameter is turned to 1, the code prints NC2O matrix corresponding to each rotor speed in the F06 file. This information can be very useful in identifying reasons for mode tracking failure.
- 10. For both approaches, damping ratio is also tracked while tracking eigenfrequencies. The results produced from mode tracking can be extracted in OP4 using following assign statements:

```
$ Extract results for numerical mode tracking
ASSIGN OUTPUT4='freq_nmt.op4',UNIT=71,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_nmt.op4',UNIT=72,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_nmt.op4',UNIT=73,FORM=FORMATTED,DELETE $ DAMPING RATIO
$ Extract results for eigenvector based mode tracking
ASSIGN OUTPUT4='freq_evmt.op4',UNIT=81,FORM=FORMATTED,DELETE $ MODE FREQS
ASSIGN OUTPUT4='rot_evmt.op4',UNIT=82,FORM=FORMATTED,DELETE $ ROTOR SPEED
ASSIGN OUTPUT4='dmp_evmt.op4',UNIT=83,FORM=FORMATTED,DELETE $ DAMPING RATIO
```

These OP4 files can be imported in MSC PATRAN to produce Campbell Diagram.

- 11. DMP>1 is not supported for SOL 110 for generating Campbell diagram when ACMS option is not used.

## CAXIFI

## Fluid Element Connections

Defines an axisymmetric fluid element that connects  $i = 2, 3,$  or  $4$  fluid points.

Formats:

1	2	3	4	5	6	7	8	9	10
CAXIF2	EID	IDF1	IDF2			RHO	B		
CAXIF3	EID	IDF1	IDF2	IDF3		RHO	B		
CAXIF4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CAXIF2	11	23	25			0.25E-3			
CAXIF3	105	31	32	33		6.47E-3			
CAXIF4	524	421	425	424	422	0.5E-3	2.5+3		

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
IDFi	Identification numbers of connected GRIDF points. ( $\text{Integer} > 0$ )
RHO	Fluid density in mass units. ( $\text{Real} > 0.0$ or blank)
B	Fluid bulk modulus. ( $\text{Real} \geq 0.0$ or blank)

Remarks:

1. CAXIFI is allowed only if an AXSLOT entry is also present.
2. The element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO or B is blank, then the corresponding RHOD and BD fields must be specified on the AXSLOT entry.
4. Plot elements are generated for these elements. Because each plot element connects two points, one is generated for the CAXIF2 element, three are generated for the CAXIF3 element, and four plot elements are generated for the CAXIF4 element. In the last case the elements connect the pairs of points (1-2), (2-3), (3-4), and (4-1).
5. If  $B = 0.0$ , the fluid is incompressible.



## CAXISYM

## Axisymmetric Line Elements Connection - SOL 400

Defines two or three node axisymmetric thick shell elements.

Formats:

1	2	3	4	5	6	7	8	9	10
CAXISYM	EID	PID	G1	G2	G3	NOFF			

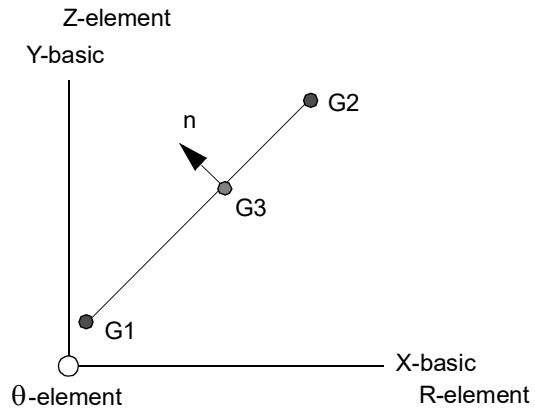
Example:

CAXISYM	22	98	8	16					
---------	----	----	---	----	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PAXISYM entry. (Integers > 0)
G1, G2	Identification numbers at the two end grid points. Required data. (Unique; Integers > 0)
G3	Identification number of the one grid in between G1 and G2. See Remark 2. (Unique Integer ≥ 0 or blank). If G3=0; this is a straight two node element.
NOFF	Offset from the surface of the grid points to the element reference plane. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other identification numbers.
2. It is recommended that G3 be located within the middle third of the element.
3. The element must lie in the x-y plane of the basic system and is oriented as shown below.



## CBAR Simple Beam Element Connection

Defines a simple beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBAR	EID	PID	GA	GB	X1	X2	X3	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	

Example:

CBAR	2	39	7	3	0.6	18.	26.	GOG	
		513							

Alternate Format and Example:

CBAR	EID	PID	GA	GB	G0			OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
CBAR	2	39	7	6	105			GOG	
		513							

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBAR, PBARL or PBRSECT entry. (Integer > 0 or blank*; Default = EID unless BAROR entry has nonzero entry in field 3.)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 8. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
OFFT	Offset vector interpretation flag. (character or blank) See Remark 8.

Descriptor	Meaning
PA, PB	Pin flags for bar ends A and B, respectively. Used to remove connections between the grid point and selected degrees-of-freedom of the bar. The degrees-of-freedom are defined in the element's coordinate system (see Figure 9-12). The bar must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4 is specified, the PBAR entry must have a value for J, the torsional stiffness. (Up to 5 of the unique Integers 1 through 6 anywhere in the field with no embedded blanks; Integer > 0.) Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not allowed in SOL 700.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors $\vec{w}_a$ and $\vec{w}_b$ , respectively (see Figure 9-12) in displacement coordinate systems (or in element system depending upon the content of the OFFT field), at points GA and GB, respectively. See Remark 7. and 8. (Real; Default = 0.0) Offsets are not allowed in SOL 700.

\*See the BAROR entry for default options for field 3 and fields 6 through 9.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Figure 9-12 and Figure 9-13 define bar element geometry with and without offsets:

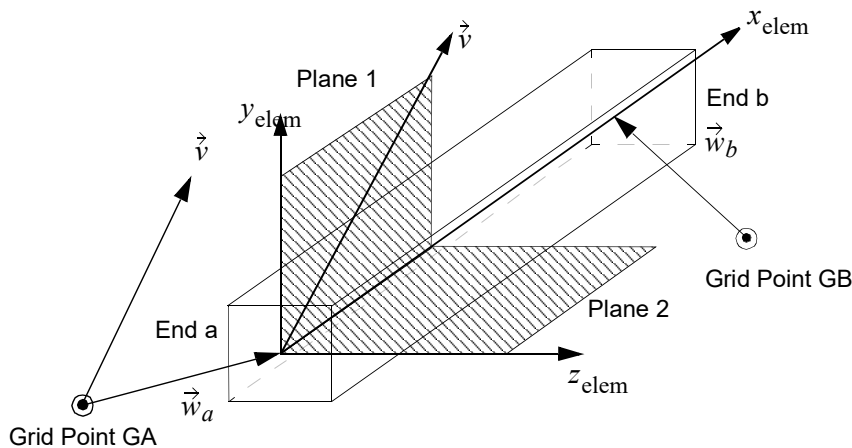


Figure 9-12 CBAR Element Geometry with Offsets

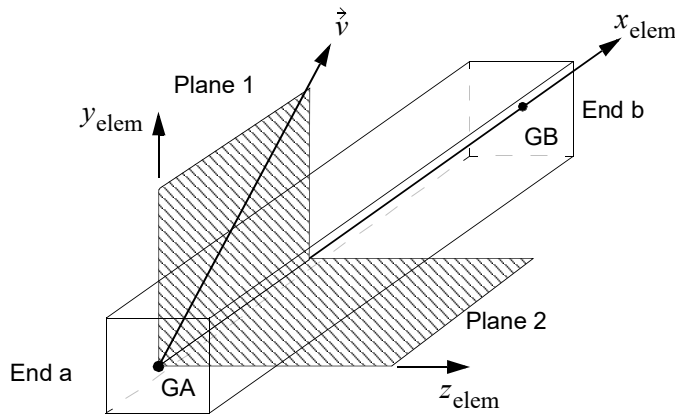


Figure 9-13 CBAR Element Geometry without Offsets

3. Figure 9-14 and Figure 9-15 define the elemental force and moment sign convention.

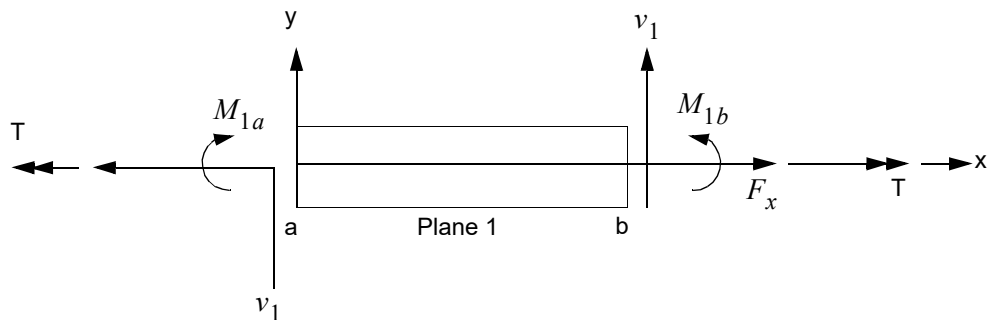


Figure 9-14 CBAR Element Internal Forces and Moments (x-y Plane)

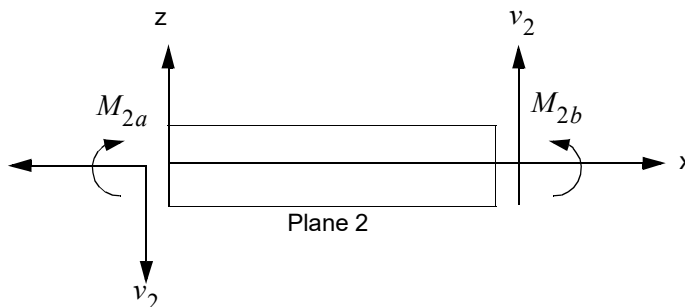


Figure 9-15 CBAR Element Internal Forces and Moments (x-z Plane)

4. The continuation may be omitted if there are no pin flags or offsets.  
 5. For the case where field 9 is blank and not provided by the BAROR entry, if an integer is specified in field 6, then G0 is used; if field 6 is blank or real, then X1, X2, X3 is used.

6. See [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide* for a definition of coordinate system terminology.
7. Offset vectors are treated internally like rigid elements. For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF.
  - Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.
  - The length of offset vectors is not affected by thermal loads. But the thermal load changes due to location changes by offsets are correctly computed if the enhanced method is used.
  - BAR elements with offsets will give correct buckling results if the enhanced method is used.
  - Masses are correctly transformed for offset effects if MDLPRM, OFFDEF, LROFF is used. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.
  - In nonlinear solution sequences, such as SOL 106 or SOL 400, BAR is treated as linear element. If geometric nonlinear effects are required, please use Bulk Data entry MDLPRM,BRTOBM,1 to convert BAR to BEAM.
8. OFFT is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset coordinate system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\vec{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character “O” in the table replaces the obsolete character “E”.)

9. For SOL 600, the BIT field is ignored unless param,MAROFSET is 1 or 2. An extra flag
10. For SOL 700, the BIT field is ignored and a warning is issued.
11. For RC network solver in thermal analysis, the G0, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B and W3B are ignored.

## CBARAO

### Auxiliary Output Points Along Bar Element Axis (CBAR Entry)

Defines a series of points along the axis of a bar element (CBAR entry) for stress and force recovery output. This entry is applicable in static and normal modes analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
CBARAO	EID	SCALE	X1	X2	X3	X4	X5	X6	

Example:

CBARAO	1065	FR	0.2	0.4	0.6	0.8			
--------	------	----	-----	-----	-----	-----	--	--	--

Alternate Format and Example:

CBARAO	EID	SCALE	NPTS	X1	DELTA				
CBARAO	1065	FR	4	0.2	0.2				

Descriptor	Meaning
EID	Element identification of a CBAR entry. (0 < Integer < 100,000,000)
SCALE	Defines scale of Xi values. (Character = "LE" or "FR")
Xi	Series of locations along element axis for stress and force data recovery. (Real > 0.0)
DELTA	Incremental distance along element axis. (Real)
NPTS	Number of stress recovery points, not including the end points. (Integer > 0)

Remarks:

1. This entry defines intermediate locations on the axis of selected CBAR elements for additional data recovery. The values of Xi are actual distances along the length if SCALE = "LE". If SCALE = "FR", the values of Xi are ratios of actual distances to the bar length. A PLOAD1 Bulk Data entry for the CBAR element in question must be present to obtain intermediate data recovery.
2. When the alternate format is used, a series of locations  $X_i = X_{[i-1]} + \text{DELTA}$ ,  $i = 1, 2, \dots, \text{NPTS}$  is generated.
3. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location Xi and output as a separate line. The force and stress values at the end points of the beam will always be output. This output format will be used for all beam and bar elements.
4. Intermediate loads on the element defined by the PLOAD1 entry will be accounted for in the calculation of element stresses and forces. If no PLOAD1 entry is defined for the element, the shear forces are constant, the moments are linear, and it is not necessary that the user define additional points.



5. For each bar element, either the basic format or the alternate format, but not both, may be used. A maximum of six internal points can be specified with the basic form. The end points must not be listed because data will be generated for them, as explained in Remark 3. If more than six unequally spaced internal points are desired, it is advisable to subdivide the bar into two or more elements.

## CBEAM

### Beam Element Connection

Defines a beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM	EID	PID	GA	GB	X1	X2	X3	OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

Example:

CBEAM	2	39	7	13	8.2	6.1	-5.6	GOG	
		513		3.0					
	8	5							

Alternate Format and Example:

CBEAM	EID	PID	GA	GB	G0			OFFT	
	PA	PB	W1A	W2A	W3A	W1B	W2B	W3B	
	SA	SB							

CBEAM	2	39	7	13	105			GOG	
		513							

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM, PBCOMP, PBEAML or PBMSECT entry. (Integer > 0; Default = EID)*
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA (Default), or in the basic coordinate system. See Remark 8. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0; G0 ≠ GA or GB)
OFFT	Offset vector interpretation flag. See Remark 8. (Character or blank)

Describer	Meaning
PA, PB	Pin flags for beam ends A and B, respectively; used to remove connections between the grid point and selected degrees-of-freedom of the beam. The degrees-of-freedom are defined in the element's coordinate system and the pin flags are applied at the offset ends of the beam (see <a href="#">Figure 9-16</a> ). The beam must have stiffness associated with the PA and PB degrees-of-freedom to be released by the pin flags. For example, if PA = 4, the PBEAM entry must have a nonzero value for J, the torsional stiffness. (Up to five of the unique Integers 1 through 6 with no embedded blanks.) Pin flags combined with offsets are not allowed for SOL 600. Pin flags are not presently allowed in SOL 700. Also, Pig flags should not be used in nonlinear analysis when there is large displacement.
W1A, W2A, W3A W1B, W2B, W3B	Components of offset vectors from the grid points to the end points of the axis of the shear center. See Remarks 7., 8. and 8. (Real; Default = 0.0)
SA, SB	Scalar or grid point identification numbers for the ends A and B, respectively. The degrees-of-freedom at these points are the warping variables $d\theta/dx$ . (Integers $\geq 0$ or blank)

\*See the BEAMOR entry for default options for field 3 and fields 6 through 9.

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For an additional explanation of the beam element, see the [Beam Element \(CBEAM\)](#) in the *MSC Nastran Reference Guide*. [Figure 9-16](#) defines beam element geometry:

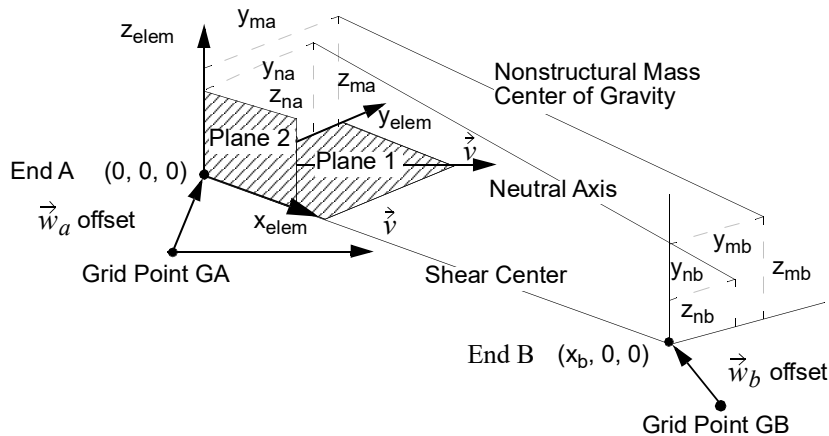


Figure 9-16 CBEAM Element Geometry System

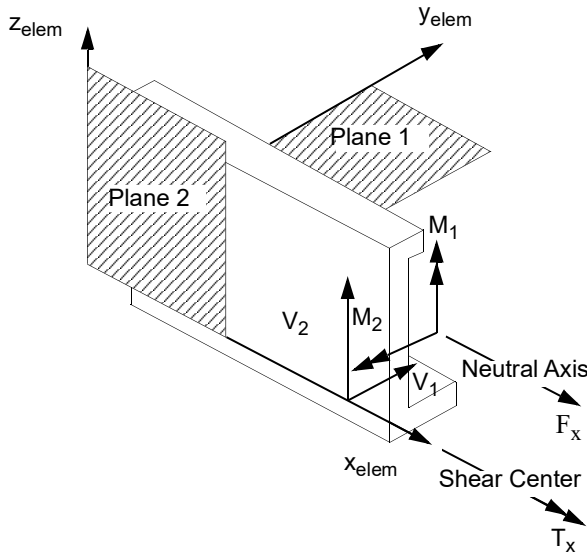


Figure 9-17 CBEAM Internal Element Forces and Moments

3. If field 6 is an integer, then G0 is used. If field 6 is blank or real, then X1, X2, X3 is used.
4. G0 cannot be located at GA or GB.
5. The rules for the continuations entries are:
  - Both continuations may be omitted if there are no pin flags, offsets, or warping variables.
  - If the second continuation is used, then the first continuation must be included, even if all fields are blank.
  - If the second continuation is omitted, torsional stiffness due to warping of the cross section will not be considered.
6. If warping is allowed ( $SA$  and  $SB > 0$ ), then  $SA$  and  $SB$  must be defined with SPOINT or GRID entries. If GRID entries are used, the warping degree-of-freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for  $SA$  and  $SB$  if GROUNDCHECK is requested in case control.
7. Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1.

8. OFFT in field 9 is a character string code that describes how the offset and orientation vector components are to be interpreted. By default (string input is GGG or blank), the offset vectors are measured in the displacement coordinate systems at grid points A and B and the orientation vector is measured in the displacement coordinate system of grid point A. At user option, the offset vectors can be measured in an offset system relative to grid points A and B, and the orientation vector can be measured in the basic system as indicated in the following table:

String	Orientation Vector	End A Offset	End B Offset
GGG	Global	Global	Global
BGG	Basic	Global	Global
GGO	Global	Global	Offset
BGO	Basic	Global	Offset
GOG	Global	Offset	Global
BOG	Basic	Offset	Global
GOO	Global	Offset	Offset
BOO	Basic	Offset	Offset

Any attempt to specify invalid combinations results in a bulk data entry input error message. For example, a value of OOO (indicating offset and orientation vectors are specified in an offset reference system) results in a fatal error since the orientation vector cannot be specified in an offset system. The offset system x-axis is defined from GA to GB. The orientation vector  $\vec{v}$  and the offset system x-axis are then used to define the z and y axes of the offset system. A vector is formed from a cross product of a vector going from Grid A to Grid B and the orientation vector to create the offset coordinate z-direction. To obtain a nonzero cross product the orientation vector must not be parallel to both vectors from Grid A to Grid B for the offset coordinate system and End A and End B for the element coordinate system. (Note: The character “O” in the table replaces the obsolete character “E”.)

9. For RC network solver in thermal analysis, the X1, X2, X3, OFFT, PA, PB, W1A, W2A, W3A, W1B, W2B, W3B, SA and SB are ignored.

## CBEAM3

### Three-Node Beam Element Connection

Defines a three-node beam element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEAM3	EID	PID	GA	GB	GC	X1	X2	X3	
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

Example:

CBEAM3	101	2	201	332	1000	1.0	3.5	-2.0	
		3.0		3.0	2.2	-1.0			
	2.5	10.	15.	20.0	206	301	312		

Alternate Format and Example:

CBEAM3	EID	PID	GA	GB	GC	G0			
	W1A	W2A	W3A	W1B	W2B	W3B	W1C	W2C	
	W3C	TWA	TWB	TWC	SA	SB	SC		

CBEAM3	101	2	201	332	1000	105			
		3.0			2.2	1.0			
	2.5	10.	15.	20.0	206	301	312		

Descriptor	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PBEAM3 or PBMSECT entries. See Remark 8. (Integer > 0; Required)
GA, GB, GC	Grid point identification numbers of connection points. GA and GB are grid point identification numbers at the two ends of the beam element while GC is the one at the grid point in between. (Integer > 0 or blank; GA, GB and GC must be distinct from each other. See Remark 6.)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. The direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then transferred to End A. (Integer > 0; G0 ≠ GA or GB or GC)

Describer	Meaning
WiA, WiB, WiC	Components of offsets vectors, measured in the displacement coordinate systems at grid points A, B, and C, from the grid points to the points on the axis of shear center. See Remark 9. (Real; Default = 0.0)
TWA, TWB, TWC	Pretwist angles in degrees at A, B, and C, respectively. (Real; Default = 0.0)
SA, SB, SC	Scalar or grid point identification numbers for A, B, and C, respectively. The degrees of freedom at these points are warping variables. (Integer > 0 or blank)

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. If field 7 is an integer, then G0 is used. If field 7 is blank or real, then X1, X2, X3 are used.
3. G0 cannot be located at GA or GB or GC.
4. If warping effect is included in the analysis (SA, SB and SC > 0), then SA, SB, and SC must be defined with either SPOINT or GRID entries. If GRID entries are used, the warping degree of freedom is attached to the first (T1) component. In addition, SPOINT ID is recommended for SA, SB and SC if GROUNDCHECK is requested in case control.
5. BEAMOR cannot be used to set up default options for field 3 and fields 6 through 8 for CBEAM3 entries.
6. If GC is left blank, then the element degenerates to a formulation similar to the two-node straight beam element. The two-node straight beam formulation is not the CBEAM formulation and will result in overly stiff results with warping and shear center offset. This is because the CBEAM3 is a Variational Asymptotic Beam Section that generalizes Timoshenko beam theory and requires the mid node for accuracy.
7. This entry is not available in SOL 600.
8. For CBEAM3 referencing PBMSECT, the CORE/LAYER keywords are required or a Fatal message is issued. For this composite case, it is recommended that CBEAM3 has 3 nodes and 3 warping DOFs.

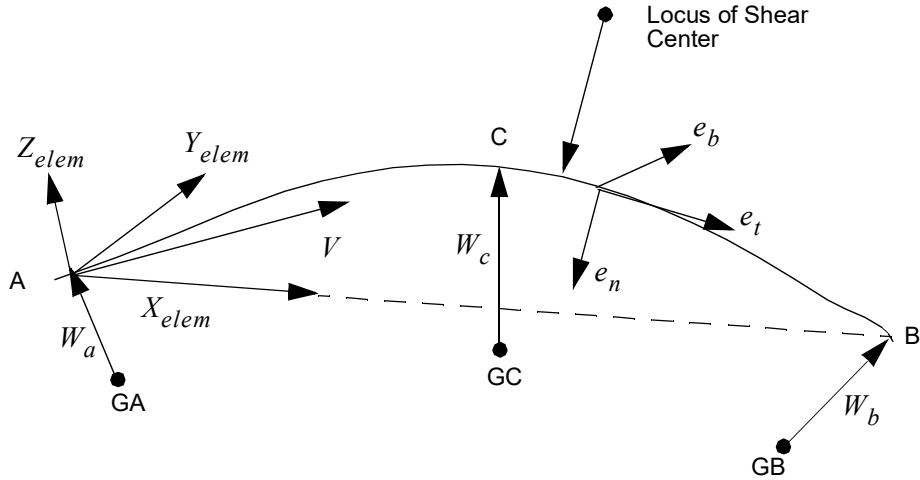


Figure 9-18 CBEAM3 Element Geometry System

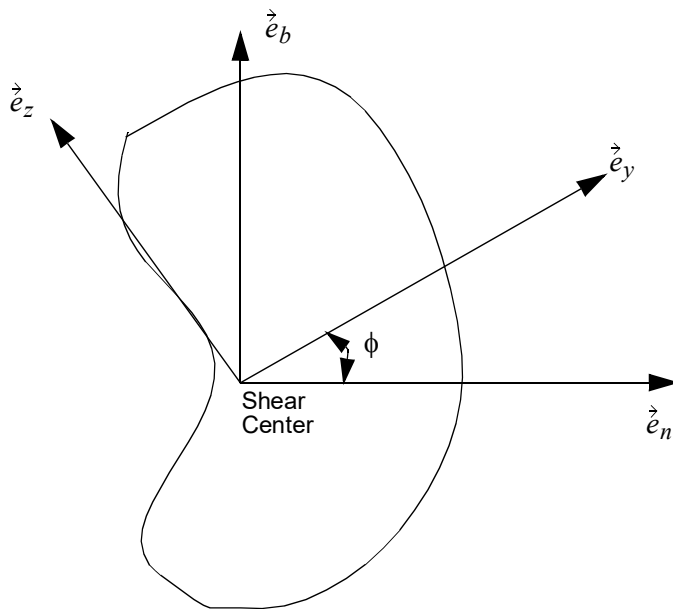


Figure 9-19 Local Coordinate System on Beam Cross-Section

- Offset vectors are treated internally like rigid elements. Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.



For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, the user is required to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

## CBEND Curved Beam or Pipe Element Connection

Defines a curved beam, curved pipe, or elbow element.

Format:

1	2	3	4	5	6	7	8	9	10
CBEND	EID	PID	GA	GB	X1	X2	X3	GEOM	

Example:

CBEND	32	39	17	19	6.2	5.1	-1.2	3	
-------	----	----	----	----	-----	-----	------	---	--

Alternate Format and Example:

CBEND	EID	PID	GA	GB	G0			GEOM	
CBEND	32	39	17	19	106			3	

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PBEND entry. (Integer > 0; Default = EID)
GA, GB	Grid point identification numbers of connection points. (Integer > 0; GA ≠ GB)
X1, X2, X3	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)
G0	Alternate method to supply the orientation vector $\vec{v}$ using grid point G0. Direction of $\vec{v}$ is from GA to G0. $\vec{v}$ is then translated to End A. (Integer > 0; G0 ≠ GA or GB)
GEOM	Flag to select specification of the bend element. See Remark 3. (1 ≤ Integer ≤ 4; No Default)

Remarks:

1. Element identification numbers must be unique with respect to all other element identification numbers.

2. For an additional explanation of the CBEND element, see the PBEND entry description. [Figure 9-20](#) and [Figure 9-21](#) define the element coordinate system and internal forces and moments.

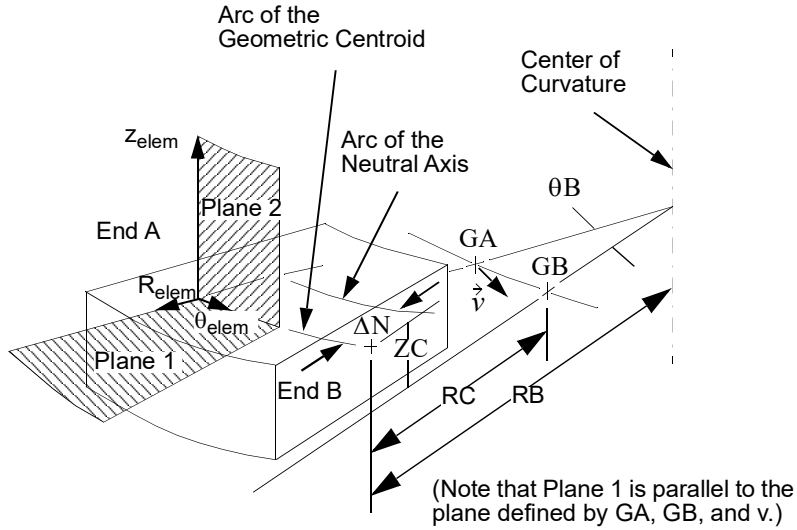


Figure 9-20 CBEND Element Coordinate System

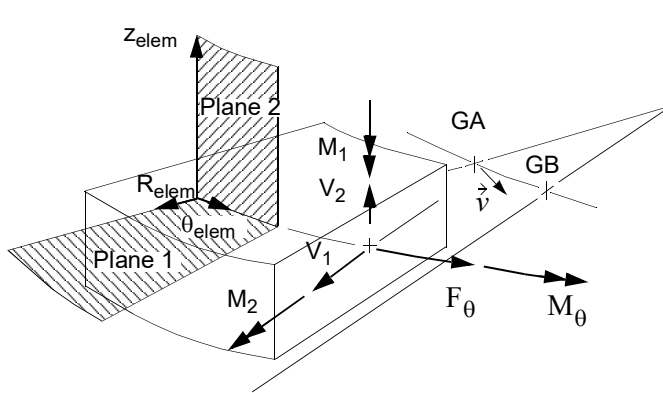
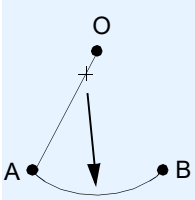
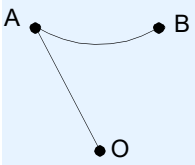
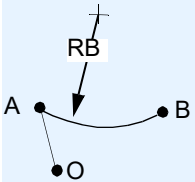
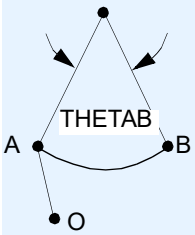


Figure 9-21 CBEND Element Internal Forces and Moments

3. The options for element connection to GA, GB using GEOM are the following.

Table 2 GEOM Options

Configuration	GEOM	Description
	1	The center of curvature lies on the line AO (or its extension) or vector $\vec{v}$ .
	2	The tangent of centroid arc at end A is parallel to line AO or vector $\vec{v}$ . Point O (or vector $\vec{v}$ ) and the arc AB must be on the same side of the chord AB.
	3	The bend radius (RB) is specified on the PBEND entry: Points A, B, and O (or vector $\vec{v}$ ) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$ ) lies on the opposite side of line AB from the center of the curvature.
	4	THETAB is specified on the PBEND entry. Points A, B, and O (or vector $\vec{v}$ ) define a plane parallel or coincident with the plane of the element arc. Point O (or vector $\vec{v}$ ) lies on the opposite side of line AB from the center of curvature.

4. For RC network solver in thermal analysis, the X1, X2, X3 and GEOM are ignored.

## CBUSH Generalized Spring-and-Damper Connection

Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH	EID	PID	GA	GB	GO/X1	X2	X3	CID	
	S	OCID	S1	S2	S3				

**Example 1:** Noncoincident grid points.

CBUSH	39	6	1	100	75				
-------	----	---	---	-----	----	--	--	--	--

**Example 2:** GB not specified.

CBUSH	39	6	1					0	
-------	----	---	---	--	--	--	--	---	--

**Example 3:** Coincident grid points ( $GA \neq GB$ ).

CBUSH	39	6	1					6	
-------	----	---	---	--	--	--	--	---	--

**Example 4:** Noncoincident grid points with fields 6 through 9 blank and a spring-damper offset.

CBUSH	39	6	1	600					
	0.25	10	0.	10.	10.				

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PBUSH entry. ( $\text{Integer} > 0$ ; Default = EID)
GA, GB	Grid point identification number of connection points. See Remark 6. ( $GA > 0$ , $GB \geq 0$ or blank)
Xi	Components of orientation vector $\vec{v}$ , from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply vector $\vec{v}$ using grid point GO. Direction of $\vec{v}$ is from GA to GO. $\vec{v}$ is then transferred to End A. See Remark 3. ( $\text{Integer} > 0$ )
CID	Element coordinate system identification. A 0 value means the basic coordinate system will be used. If CID is blank, then the element coordinate system is determined from GO or Xi. See Figure 9-22 and Remark 3. ( $\text{Integer} \geq 0$ or blank)
S	Location of spring damper. See Figure 9-22. (Real; Default = 0.5)

Describer	Meaning
OCID	Coordinate system identification of spring-damper offset. See Remark 9. (Integer $\geq -1$ ; Default = -1, which means the offset point lies on the line between GA and GB according to Figure 9-22)
S1, S2, S3	Components of spring-damper offset in the OCID coordinate system if OCID $\geq 0$ . See Figure 9-23 and Remark 9. (Real)

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Figure 9-22 shows the bush element geometry. In geometric nonlinear analysis (SOL400), the element axis (line GA to GB) follows the deformation of grids GA and GB.
3.  $CID \geq 0$  overrides GO and Xi. Then the element x-axis is along T1, the element y-axis is along T2, and the element z-axis is along T3 of the CID coordinate system. If the CID refers to a cylindrical coordinate system or a spherical coordinate system, then grid GA is used to locate the system. If for cylindrical or spherical coordinate, GA falls on the z-axis used to define them, it is recommended that another CID be selected to define the element x-axis.
4. For noncoincident grids ( $GA \neq GB$ ), when GO or (X1, X2, X3) is given and no CID is specified, the line AB is the element x-axis and the orientation vector  $\vec{v}$  lies in the x-y plane (similar to the CBEAM element).
5. For noncoincident grids ( $GA \neq GB$ ), if neither GO or (X1, X2, X3) is specified and no CID is specified, then the line AB is the element x-axis. This option is valid only when K1 (or B1) or K4 (or B4) or both on the PBUSH entry are specified (but K2, K3, K5, K6 or B2, B3, B5, B6 are not specified). If K2, K3, K5, or K6 (or B2, B3, B5, or B6) are specified, a fatal message will be issued.
6. If the distance between GA and GB is less than .0001, or if GB is blank, then CID must be specified. GB blank implies that B is a grounded terminal, a grounded terminal is a point with a displacement that is constrained to zero.
7. If PID references a PBUSHT entry, then the CBUSH element may only be defined in the residual structure and cannot be attached to any omitted degrees-of-freedom.
8. Element impedance output is computed in the CID coordinate system. The impedances in this system are uncoupled.
9. If OCID = -1 or blank (default) then S is used and S1, S2, S3 are ignored. If OCID  $\geq 0$ , then S is ignored and S1, S2, S3 are used.

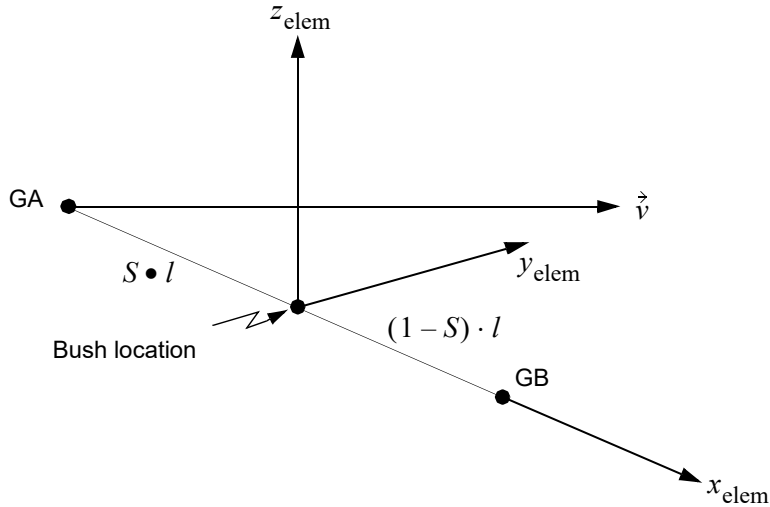
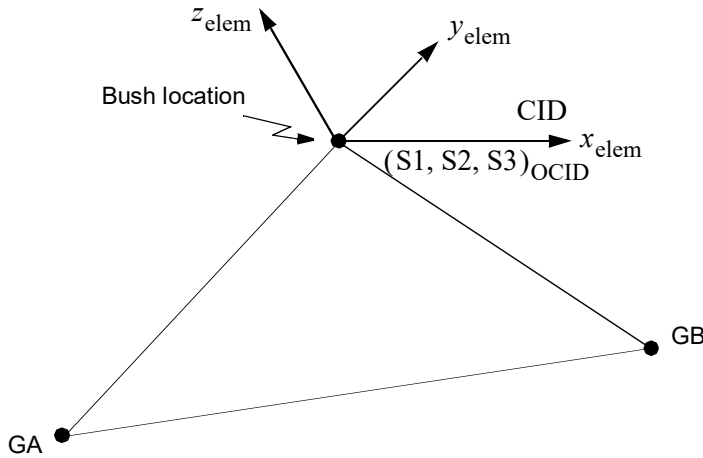


Figure 9-22 CBUSH Element



The material stiffness and damping properties of the elastomer are located at  $(S1, S2, S3)$ .

Figure 9-23 Definition of Offset S1, S2, S3

- When  $CID \geq 0$ , the element x-axis is set as in Remark 3. This means that the element force is always computed as  $K_e \cdot (UB - UA)$ ; if  $UA > UB$ , a compressive force will result. This is unlike the GO or Xi options, where relative positive elongation in tension and relative negative elongation is compression.

11. The CBUSH element is designed to satisfy rigid body equilibrium requirements. For noncoincident grids, internal rigid links connect the bush location to the grid locations. This results in coupling between translational and rotational degrees-of-freedom at the grids even when no rotational springs or dampers are specified on the PBUSH.
12. CBUSH elements are not supported in thermal analysis.



## CBUSH1D

### Rod Type Spring-and-Damper Connection

Defines the connectivity of a one-dimensional spring and viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH1D	EID	PID	GA	GB	CID				

Example:

CBUSH1D	35	102	108	112					
---------	----	-----	-----	-----	--	--	--	--	--

Describer	Meaning	Default Values
EID	Element identification number. (0 < Integer < 100,000,000)	Required
PID	Property identification number of a PBUSH1D entry. (Integer > 0)	EID
GA	Grid point id of first grid.	Required
GB	Grid point id of second grid	blank
CID	Coordinate system id. (Integer ≥ 0)	blank

Remarks:

1. For noncoincident grids  $GA \neq GB$  and if CID is blank, the line GA to GB is the element axis. In geometric nonlinear analysis, the element axis (line GA to GB) follows the deformation of grids GA and GB. See [Figure 9-24](#).
2. If  $CID \geq 0$  is specified, the x-axis of the CID coordinate system is the element axis. In geometric nonlinear analysis, the element axis (x-axis of CID) remains fixed.
3. If GA and GB are coincident or if GB is blank, then  $CID \geq 0$  must be specified and the element axis is the x-axis of CID.

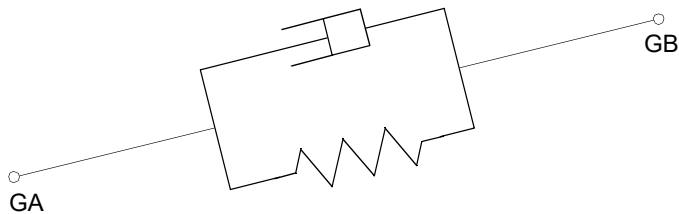


Figure 9-24 Spring and Damper Element

## CBUSH2D

## 2-D Linear-Nonlinear Connection

Defines the connectivity of a two-dimensional Linear-Nonlinear element.

Format:

1	2	3	4	5	6	7	8	9	10
CBUSH2D	EID	PID	GA	GB	CID	PLANE			

Example:

CBUSH2D	100	101	1001	2001	0	XY			
---------	-----	-----	------	------	---	----	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000; Required)
PID	Property identification number of a PBUSH2D. (Integer > 0; Required)
GA	Inner grid. (Integer > 0; Required)
GB	Outer grid. (Integer > 0; Required)
CID	Coordinate system used to define 2-D plane. (Integer ≥ 0; Default = 0)
PLANE	Orientation plane in CID: XY, YZ, ZX, see Remark 1. (Character; Default = 'XY')

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacements coordinates of GA and GB. The planes correspond to directions 1 and 2. GA and GB should be coincident grids with parallel displacement coordinate systems. The coordinate systems are not checked. Wrong answers will be produced if this rule is not followed.
2. The behavior is different if there is no rotor in the model and a CBUSH2D is used. When this happens, for most cases, if there is no rotor the nominal stiffness values from the PBUSH2D will be used, with no frequency-dependence. However, if an ELEMUDS is used, the CBUSH2D will be ignored if there is no rotor in the model. Also, an ELEMUDS with CBUSH2DA call will require to have a rotor in the model. A FATAL message will be used if there is no rotor, for ELEMUDS having CBUSH2DA call.

## CCONEAX

### Axisymmetric Shell Element Connection

Defines a conical shell element.

Format:

1	2	3	4	5	6	7	8	9	10
CCONEAX	EID	PID	RA	RB					

Example:

CCONEAX	1	2	3	4					
---------	---	---	---	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PCONEAX entry. (Integer > 0; Default = EID)
RA	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)
RB	Identification number of a RINGAX entry. (Integer > 0; RA ≠ RB)

Remarks:

1. This element has limited capabilities. See the *MSC Nastran Reference Guide*, Section 5.3.3.
2. This entry is allowed only if an AXIC entry is also present.
3. In order to reference this entry on a SET Case Control command, the ID must be modified by

$$ID_n = ID \cdot 1000 + n$$

where n is the harmonic number plus one and ID<sub>n</sub> is the value specified on the SET entry.

## CDAMP1 Scalar Damper Connection

Defines a scalar damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1	EID	PID	G1	C1	G2	C2			

Example:

CDAMP1	19	6	0		23	2			
--------	----	---	---	--	----	---	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PDAMP property entry. ( $\text{Integer} > 0$ ; Default = EID)
G1, G2	Geometric grid point identification number. ( $\text{Integer} \geq 0$ )
C1, C2	Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point.)

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2), must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.

## CDAMP1D

## Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP1D	1001	101	55	1					
---------	------	-----	----	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMPn entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer ≥ 0)
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.

Remark:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2), must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- When CDAMP1 is used in heat transfer analysis, it generates a lumped heat capacity.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

## CDAMP2

## Scalar Damper Property and Connection

Defines a scalar damper element without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2	EID	B	G1	C1	G2	C2			

Example:

CDAMP2	16	2.98	32	1					
--------	----	------	----	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; blank or zero if scalar point.)

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2), must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
8. RC network solver does not support CDAMP2 for thermal analysis.

## CDAMP2D

## Scalar Damper Connection for SOL 700 Only

Defines a scalar damper connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP2D	EID	B	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CDAMP2D	1001	101	55	1					
---------	------	-----	----	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
B	Value of the scalar damper. (Real)
G1, G2	Geometric grid point identification number. ( $\text{Integer} \geq 0$ )
C1, C2	Component number. ( $0 \leq \text{Integer} \leq 6$ ; 0 or up to six unique integers, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. ( $\text{Integer} \geq 0$ )
FOLLOW	Method to update the direction vector in which the damper acts: FOLLOW=CORD: direction vector follows the motion of the coordinate system as specified under CORD.

Remark:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2), must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASI, CMASSI, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- When CDAMP2 is used in heat transfer analysis, it generates a lumped heat capacity.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry. Available in SOL 700 only.

## CDAMP3 Scalar Damper Connection to Scalar Points Only

Defines a scalar damper element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP3	EID	PID	S1	S2					

Example:

CDAMP3	16	978	24	36					
--------	----	-----	----	----	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PDAMP entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. When CDAMP3 is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP3 for thermal analysis.



## CDAMP4

## Scalar Damper Property and Connection to Scalar Points Only

Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP4	EID	B	S1	S2					

Example:

CDAMP4	16	-2.6	4	9					
--------	----	------	---	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
B	Scalar damper value. (Real)
S1, S2	Scalar point identification numbers. (Integer $\geq$ 0; S1 $\neq$ S2 )

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar damper element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. If this entry is used in heat transfer analysis, it generates a lumped heat capacity.
6. A scalar point specified on this entry need not be defined on an SPOINT entry.
7. RC network solver does not support CDAMP4 for thermal analysis.

## CDAMP5 Scalar Damper with Material Property

Defines a damping element that refers to a material property entry and connection to grid or scalar points. This element is intended for heat transfer analysis only.

Format:

1	2	3	4	5	6	7	8	9	10
CDAMP5	EID	PID	G1	G2					

Example:

CDAMP5	1	4	10	20					
--------	---	---	----	----	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Identification number of a PDAMP5 property entry. ( $\text{Integer} > 0$ ; Default = EID)
G1, G2	Grid or scalar point identification numbers. ( $\text{Integer} \geq 0$ and $G1 \neq G2$ )

Remarks:

1. G1 or G2 may be blank or zero indicating a constraint.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. CDAMP5 generates a lumped heat capacity in heat transfer analysis.
4. A scalar point specified on CDAMP5 need not be defined on an SPOINT entry.
5. This entry is not supported in SOL 600.
6. RC network solver does not support CDAMP5 for thermal analysis.

## CDUMi Dummy Element Connection

Defines a dummy element ( $3 \leq i \leq 7$ ).

Format:

1	2	3	4	5	6	7	8	9	10
CDUMi	EID	PID	G1	G2	G3	G4	-etc.-		
	A1	A2	-etc.-						

Example:

CDUM3	114	108	2	5	6	8	11		
	2.4		3.E4	2		50			

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PDUMi entry. See Remark 2. ( $\text{Integer} > 0$ )
Gi	Grid point identification numbers of connection points. ( $\text{Integer} > 0$ ; $G1 \neq G2$ $\dots \neq GN$ )
Ai	Additional fields. (Real or Integer)

Remarks:

1. The user must write the associated element subroutines for matrix generation, stress recovery, etc., and perform a link edit to replace the dummy routines. See the *MSC Nastran Programmer's Manual*.
2. If no property entry is required, PID may contain the material identification number.
3. Additional entries are defined in the user-written element routines.
4. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran" Other than the EID field, all field checking is the responsibility of the user supplied code.

## CELAS1      Scalar Spring Connection

Defines a scalar spring element.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1	EID	PID	G1	C1	G2	C2			

Example:

CELAS1	2	6			8	1			
--------	---	---	--	--	---	---	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PELAS entry. ( $\text{Integer} > 0$ ; Default = EID)
G1, G2	Geometric grid point identification number. ( $\text{Integer} \geq 0$ )
C1, C2	Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point.)

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2) must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
- It is recommended that for most applications the grids connected by CELAS1 entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

## CELAS1D

## Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS1D	EID	PID	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CELAS1D	1001	101	55	1	8	1			
---------	------	-----	----	---	---	---	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer ≥ 0) (SOL 700 only)
FOLLOW	Method to update the direction vector in which the spring acts: For SOL 700: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.

Remarks:

1. Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS3 entry.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. The two connection points (G1, C1) and (G2, C2) must be distinct.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

6. If  $G_i$  refers to a grid point then  $C_i$  refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
7. Available in SOL 700 only.
8. It is recommended that for most applications the grids connected by CELAS1D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

## CELAS2      Scalar Spring Property and Connection

Defines a scalar spring element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2	EID	K	G1	C1	G2	C2	GE	S	

Example:

CELAS2	28	6.2+3	32		19	4			
--------	----	-------	----	--	----	---	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point or scalar identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 blank or zero if scalar point.)
GE	Damping coefficient. See Remarks 6. and 8. (Real)
S	Stress coefficient. (Real)

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2) must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters](#).
- If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.
- To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.

## CELAS2D

## Scalar Spring Connection for SOL 700 Only

Defines a scalar spring connection for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS2D	EID	K	G1	C1	G2	C2			
	CORD	FOLLOW							

Example:

CELASD2	1001	101	55	1	8	1			
---------	------	-----	----	---	---	---	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
G1, G2	Geometric grid point identification number. (Integer ≥ 0)
C1, C2	Component number. (0 ≤ Integer ≤ 6; 0 or up to six unique, 1 through 6 may be specified in the field with no embedded blanks. 0 applies to scalar points and 1 through 6 applies to grid points.)
CORD	Number of a coordinate system in which the degree-of-freedom (C1,C2) is defined. (Integer ≥ 0)
FOLLOW	Method to update the direction vector in which the spring acts: FOLLOW=CORD: Direction vector follows the motion of the coordinate system as specified under CORD.

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CELAS4 entry.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2) must be distinct.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom in the displacement coordinate system specified by CD on the GRID entry.



7. It is recommended that for most applications the grids connected by CELAS2D entries be coincident and the displacement coordinate systems of each grid be congruent. Connecting non-coincident grids and/or non-congruent coordinate systems can lead to models that do not pass strain energy checks (see the [GROUNDCHECK \(Case\)](#) Case Control command).

## CELAS3      Scalar Spring Connection to Scalar Points Only

Defines a scalar spring element that connects only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS3	EID	PID	S1	S2					

Example:

CELAS3	19	2	14	15					
--------	----	---	----	----	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PELAS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar spring element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CELAS4**

Scalar Spring Property and Connection to Scalar Points Only

Defines a scalar spring element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CELAS4	EID	K	S1	S2					

Example:

CELAS4	42	6.2-3	2						
--------	----	-------	---	--	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
K	Stiffness of the scalar spring. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2, but not both, may be blank or zero indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. A structural damping coefficient is not available with CELAS4. The value of g is assumed to be 0.0.
4. No stress coefficient is available with CELAS4.
5. Only one scalar spring element may be defined on a single entry.
6. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
7. A scalar point specified on this entry need not be defined on an SPOINT entry.

## CFAST A Shell Patch Fastener Connection

Defines a fastener with material orientation connecting two surface patches. Large displacement and large rotational effects are supported when in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
CFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS						

Example using PROP:

CFAST	3	20	PROP	21	24	206			
-------	---	----	------	----	----	-----	--	--	--

Example using ELEM:

CFAST	7	70	ELEM	27	74	707			
-------	---	----	------	----	----	-----	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default = EID)
TYPE	Specifies the surface patch definition: (Character) If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. See Remark 1. and Figure 9-25. If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. See Remark 1. and Figure 9-25.
IDA,IDB	Property id (for PROP option) or Element id (for ELEM option) defining patches A and B. IDA ≠ IDB (Integer > 0)
GS	Grid point defining the location of the fastener. See Remark 2. (Integer > 0 or blank)
GA,GB	Grid ids of piercing points on patches A and B. See Remark 2. (Integer > 0 or blank)
XS,YS,ZS	Location of the fastener in basic. Required if neither GS nor GA is defined. See Remark 2. (Real or blank)

Remarks:

1. The CFAST defines a flexible connection between two surface patches. The number of unique physical grids connected depends on the location for the piercing points GA and GB and the size of the diameter D (see PFAST).

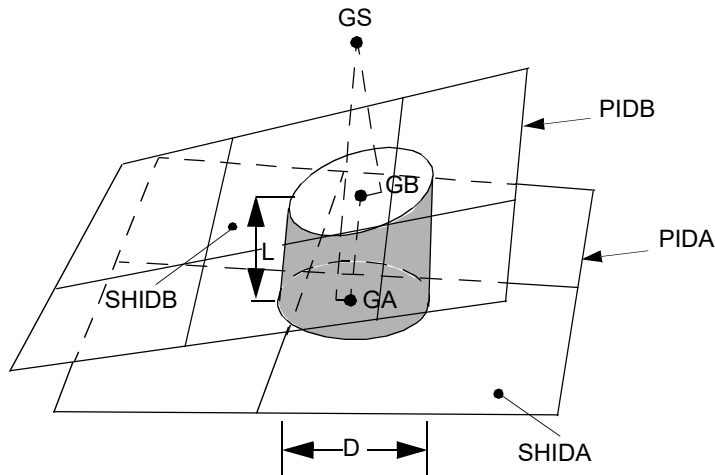


Figure 9-25 Patches Defined with TYPE= 'PROP' or TYPE = 'ELEM'

- GS defines the approximate location of the fastener in space. GS is projected onto the surface patches A and B. The resulting piercing points GA and GB define the axis of the fastener. GS does not have to lie on the surfaces of the patches. GS must be able to project normals to the two patches. GA can be specified in lieu of GS, in which case GS will be ignored. If neither GS nor GA is specified, then (XS, YS, ZS) in basic must be specified.

If both GA and GB are specified, they must lie on or at least have projections onto surface patches A and B respectively. If GA and GB are both specified, GS is ignored. By default, the locations of user specified GA and GB will not be changed. If the user specifies "SWLDPRM, MOVGAB, 1," then the locations will be corrected so that they lie on the surface patches A and B within machine precision. The length of the fastener is the final distance between GA and GB. If the length is zero, the normal to patch A is used to define the axis of the fastener.

Diagnostic printouts, checkout runs and control of search and projection parameters are requested on the SWLDPRM Bulk Data entry.

- The use of param,cfdiagp,yes and param,cfrandel,real\_fraction\_value allows for the random removal of a percentage of CFAST elements for failure studies.
- This entry is not supported in SOL 700.
- If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
- In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
- If partitioned superelements are present, then CFAST is supported in the main Bulk Data section only.

**CFLUIDi****Fluid Element Connections**

Defines three types of fluid elements for an axisymmetric fluid model.

Formats:

1	2	3	4	5	6	7	8	9	10
CFLUID2	EID	IDF1	IDF2			RHO	B		
CFLUID3	EID	IDF1	IDF2	IDF3		RHO	B		
CFLUID4	EID	IDF1	IDF2	IDF3	IDF4	RHO	B		

Examples:

CFLUID2	100	11	14			.025	0.0		
CFLUID3	110	15	13	12		1.2			
CFLUID4	120	11	15	12	14				

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
IDFi	Identification number of a RINGFL entry. (Integer > 0; IDF1 ≠ IDF2 ≠ IDF3 ≠ IDF4; all IDF <sub>i</sub> < 500000)
RHO	Mass density. (Real > 0.0; Default is the value of DRHO on the AXIF entry)
B	Bulk modulus, pressure per volume ratio. (Real; Default is the value of DB on the AXIF entry)

Remarks:

1. CFLUIDi is allowed only if an AXIF entry is also present.
2. Element identification number must be unique with respect to all other fluid, scalar, and structural elements.

- The volume defined by IDFi is a body of revolution about the polar axis of the fluid coordinate system defined by AXIF. CFLUID2 defines a thick disk with IDF1 and IDF2 defining the outer corners as shown in [Figure 9-26](#):

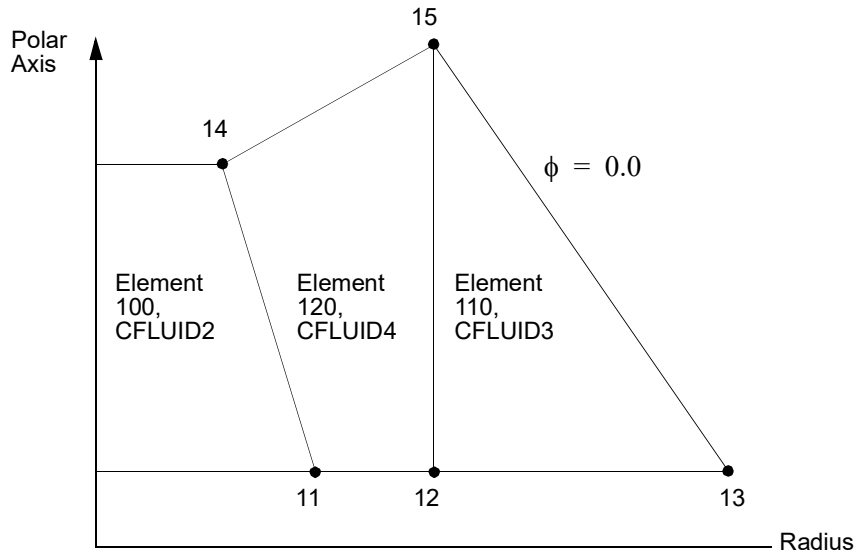


Figure 9-26 CFLUIDi Examples

- All interior angles must be less than  $180^\circ$ .
- The order of connected RINGFL points is arbitrary.
- If  $B = 0.0$ , the fluid is incompressible.

## CGAP

## Gap Element Connection

Defines a gap or friction element.

Format:

1	2	3	4	5	6	7	8	9	10
CGAP	EID	PID	GA	GB	X1	X2	X3	CID	

Example:

CGAP	17	2	110	112	5.2	0.3	-6.1		
------	----	---	-----	-----	-----	-----	------	--	--

Alternate Format and Example:

CGAP	EID	PID	GA	GB	GO			CID	
CGAP	17	2	110	112	13				

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PGAP entry. (Integer $> 0$ ; Default = EID)
GA, GB	Connected grid points at ends A and B. (Integers $> 0$ ; $GA \neq GB$ )
X1, X2, X3	Components of the orientation vector $\bar{v}$ , from GA, in the displacement coordinate system at GA. (Real)
GO	Alternate method to supply the orientation vector $\bar{v}$ using grid point GO. Direction of $\bar{v}$ is from GA to GO. (Integer $> 0$ )
CID	Element coordinate system identification number. CID must be specified if GA and GB are coincident (distance from GA to GB $< 10^{-4}$ ). See Remark 6. (Integer $\geq 0$ or blank)

Remarks:

1. The CGAP element is intended for the nonlinear solution sequences 106, 129, 153, 159, and 400. However, it will produce a linear stiffness matrix for the other solutions, but remains linear with the initial stiffness. The stiffness used depends on the value for the initial gap opening (U0 field in the PGAP entry).
2. The gap element coordinate system is defined by one of two following methods:
  - If the coordinate system (CID field) is specified, the element coordinate system is established using that coordinate system, in which the element x-axis is in the T1 direction and the y-axis in the T2 direction. The orientation vector  $\bar{v}$  will be ignored in this case.
  - If the CID field is blank and the grid points GA and GB are not coincident (distance from A to B  $\geq 10^{-4}$ ), then the line AB is the element x-axis and the orientation vector  $\bar{v}$  lies in the x-y plane (like the CBEAM element).



3. The element coordinate system does not rotate as a result of deflections.
4. Initial gap openings are specified on the PGAP entry and not derived from the separation distance between GA and GB.
5. Forces are requested with the FORCE or NLSTRESS Case Control command. The NLSTRESS command is only for nonlinear solutions, and the output also includes the gap STATUS. Forces are output in the element coordinate system. The force  $F_x$  is positive for compression.
6. If CID is being used to define the element coordinate system and the CID refers to either a cylindrical or spherical coordinate system then grid GA will be used to locate the system. If grid GA lies on the z-axis of the cylindrical or spherical coordinate system it is recommended that a different coordinate system be used for this element.
7. See PARAM,CDITER for an alternative approach.

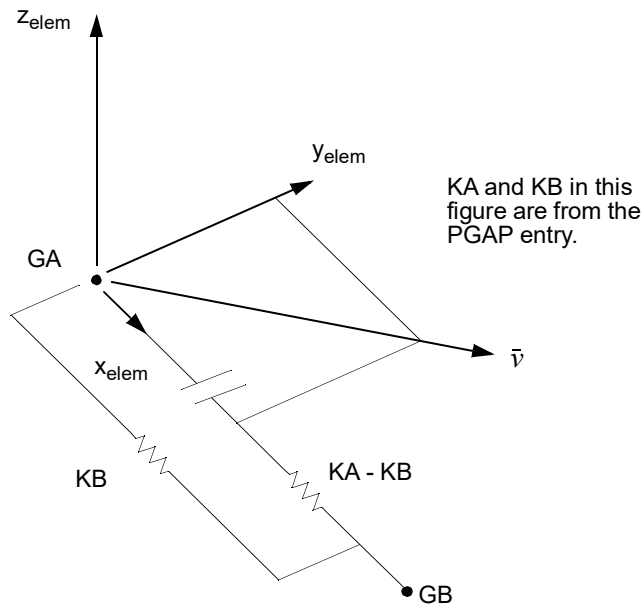


Figure 9-27 CGAP Element Coordinate System

8. Since a large stiffness is used for KA (the closed GAP stiffness), param,g damping should be avoided. Instead damping should be specified on the MATi entries and PARAM,W4 set.

## CHACAB

## Acoustic Absorber Element Connection

Defines the acoustic absorber element in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CHACAB	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACAB	95	12	1	2	5	7	8	9	
	24	23							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PACABS entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0 or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grid points G1 through G4 and G9 through G12 is assumed to be in contact with the structure.

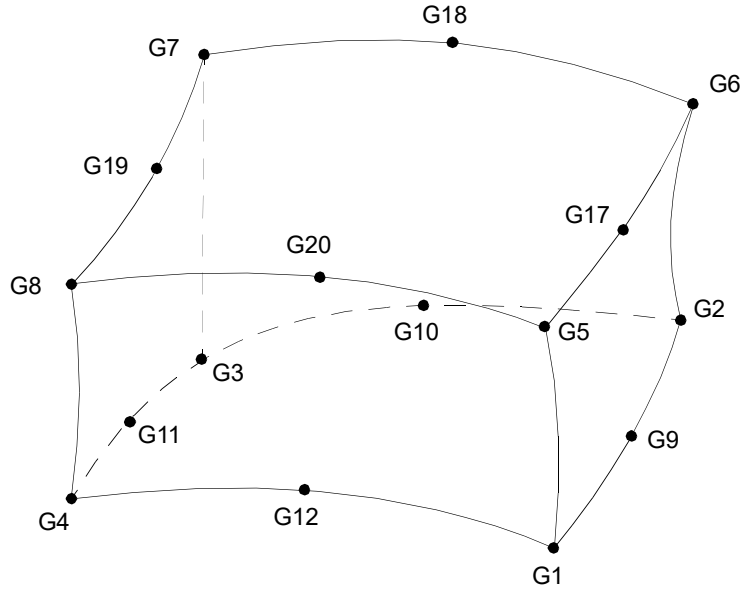


Figure 9-28 CHACAB Element Connection

6. The mass is lumped to the face formed by grid points G5 through G8 and G17 through G20 and defined to be in contact with the fluid. The opposite face has no mass contribution due to the absorber element. Also, the face in contact with the fluid has only translational stiffness in the direction normal to the face.

## CHACBR

## Acoustic Barrier Element Connection

Defines the acoustic barrier element.

Format:

1	2	3	4	5	6	7	8	9	10
CHACBR	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12			
			G17	G18	G19	G20			

Example:

CHACBR	95	12	1	2	5	7	8	9	
	24	23							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PACBAR entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. The edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted.
4. It is recommended that the edge points be located within the middle third of the edge.
5. The face consisting of grids G1 through G4 and G9 through G12 is assumed to be the backing that corresponds to MBACK on the PACBAR entry.
6. The face consisting of grid points G5 through G8 and G17 through G20 is assumed to be the septum that corresponds to MSEPTM on the PACBAR entry.

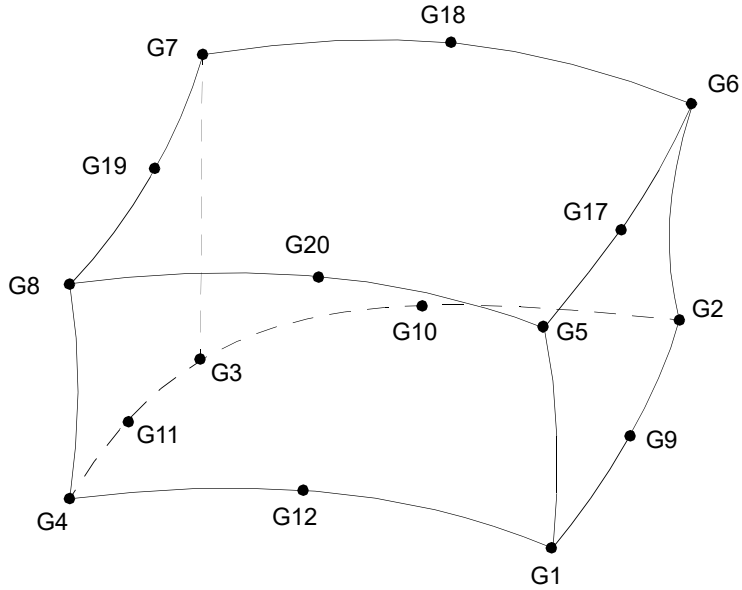


Figure 9-29 CHACBR Element Connection

7. The face in contact with the fluid is defined to be the face formed by grid points G5 through G8 and G17 through G20 and has only translational stiffness in the direction normal to the face.

## CHBDYE

## Geometric Surface Element Definition (Element Form)

Defines a boundary condition surface element with reference to a heat conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		

Example:

CHBDYE	2	10	1	3	3	2	2		
--------	---	----	---	---	---	---	---	--	--

Describer	Meaning
EID	Surface element identification number for a specific side of a particular element. See Remarks 1. and 9. (Unique (0 < Integer < 100,000,000) among all elements.)
EID2	A heat conduction element identification number. (Integer > 0)
SIDE	A consistent element side identification number. See Remark 6. ( $1 \leq \text{Integer} \leq 6$ )
IVIEWF	A VIEW entry identification number for the front face of surface element. See Remark 2. for default. (Integer $\geq 0$ )
IVIEWB	A VIEW entry identification number for the back face of surface element. See Remark 2. for default. (Integer $\geq 0$ )
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. (Integer $\geq 0$ )
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. (Integer $\geq 0$ )

Remarks:

1. EID is a unique elemental ID associated with a particular surface element. EID2 identifies the general heat conduction element being considered for this surface element.
2. The defaults for IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank both on the CHBDYE entry and the BDYOR entry, then the default is zero.
3. For the front face of shell elements, the right-hand rule is used as one progresses around the element surface from G1 to G2 to ... Gn. For the edges of shell elements or the ends of line elements, an outward normal is used to define the front surface.
4. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
5. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.

6. Side conventions for solid elements.

The sides of the solid elements are numbered consecutively according to the order of the grid point numbers on the solid element entry. The sides of solid elements are either quadrilaterals or triangles. For each element type, tabulate the grid points (gp) at the corners of each side.

**8-node or 20-node CHEXA**

side	gp	gp	gp	gp
1	4	3	2	1
2	1	2	6	5
3	2	3	7	6
4	3	4	8	7
5	4	1	5	8
6	5	6	7	8

**CPENTA**

side	gp	gp	gp	gp
1	3	2	1	
2	1	2	5	4
3	2	3	6	5
4	3	1	4	6
5	4	5	6	

**CTETRA**

side	gp	gp	gp
1	3	2	1
2	1	2	4
3	2	3	4
4	3	1	4

**CPYRAM**

side	gp	gp	gp	gp
1	3	2	5	
2	1	4	5	
3	2	1	5	
4	3	5	4	
5	4	1	2	3

7. Side conventions for shell elements.

Side 1 of shell elements (top) are of an AREA type, and additional sides (2 through a maximum of 5 for a QUAD) are of LINE type. (See CHBDYG for surface type definition.)

Area Type Sides -- The first side is that given by the right-hand rule on the shell elements grid points.

Line Type Sides -- The second side (first line) proceeds from grid point 1 to grid point 2 of the shell element, and the remaining lines are numbered consecutively. The thickness of the line is that of the shell element, and the normal to the line is outward from the shell element in the plane of the shell. Note that any midside nodes are ignored in this specification.

For 3-D heat shell elements when used in combination with linear or quadratic nodal temperature (see option TEMPP of the NLMOPTS entry), SIDE=6 refers to BOT surface. SIDE=1 refers to TOP surface. The IVIEW and RADM should be filled in consistently in relation with the SIDE entry. E.g., when SIDE=1 or 6 then only IVIEWF makes sense.

8. Side conventions for line elements.

LINE elements have one linear side (side 1) with geometry that is the same as that of the element and two POINT-type sides corresponding to the two points bounding the linear element (first grid point-side 2; second grid point-side 3).

The TUBE-type element has two linear sides of type TUBE. The first side represents the outside with diameters equal to that of the outside of the tube. The second side represents the inside with diameters equal to that of the inside of the tube.

Point Sides -- Point sides may be used with any linear element. The direction of the outward normals of these points is in line with the element axis, but pointing away from the element. The area assigned to these POINT-type sides is consistent with the element geometry.

Rev Sides -- The CTIRAX6 element has associated with it three REV sides. The first side is associated with Grid Points G1, G2, and G3. The positive face identification normals point away from the element.



9. Application of boundary conditions to CHBDYE is referenced through the EID. Boundary conditions can reference either the front or back face of the CHBDYE by specifying +EID or -EID respectively. Correspondingly, the back face is minus the normal vector of the front face. Similarly, IVIEWF and RADMIDF are associated with +EID and IVIEWB and RADMIDB with -EID. For radiation problems, if the RADMIDF or RADMIDB is zero, default radiant properties assume perfect black body behavior.
10. Starting with MSC Nastran 2004, axisymmetric view factors are supported CHBDYG of TYPE=REV, but not supported CHBDYE. If CHBDYE is used for this, axisymmetric view factors are not calculated.

## CHBDYG Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

Describer	Meaning
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all elemental entries)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	A VIEW entry identification number for the front face. See Remark 2. for default. (Integer ≥ 0)
IVIEWB	A VIEW entry identification number for the back face. See Remark 2. for default. (Integer ≥ 0)
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. (Integer ≥ 0)
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. (Integer ≥ 0)
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

Remarks:

- EID is a unique ID associated with a particular surface element as defined by the grid points.
- The defaults for TYPE, IVIEWF, IVIEWB, RADMIDF, and RADMIDB may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYG entry and the BDYOR entry, then the default is zero.
- TYPE specifies the kind of element surface; allowed types are: REV, REV1, AREA3, AREA4, AREA6, and AREA8. See [Figure 9-30](#), [Figure 9-31](#), and [Figure 9-32](#).
  - TYPE = REV

The “REV” type has two primary grid points that must lie in the x-z plane of the basic coordinate system with  $x > 0$ . A midside grid point G3 is optional and supports convection or heat flux from the edge of the six-noded CTRIAX6 element. The defined area is a conical section with z as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.

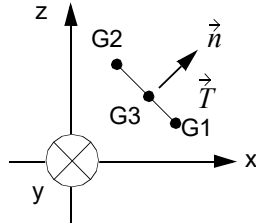


Figure 9-30 Normal Vector for CHBDYG Element of Type “REV”

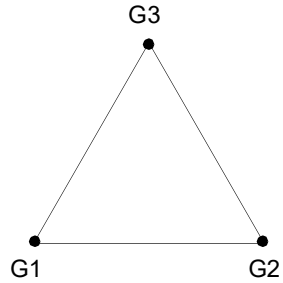
The unit normal lies in the x-z plane, and is given by

$$\vec{n} = (\vec{e}_y \times \vec{T}) / |\vec{e}_y \times \vec{T}|.$$

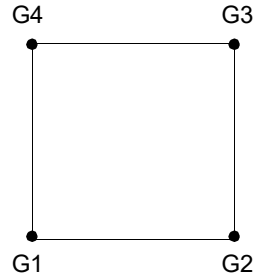
$\vec{e}_y$  is the unit vector in the y direction.

- TYPE = REV1. The “REV1” type has two primary grid points that must lie in the x-y plane of the basic coordinate system with  $x > 0$ . A midside grid point G3 is optional and supports convection or heat flux from the edge of the CQUADX (4 or 8 node) and CTRIAX (3 or 6 node) element. The defined area is a conical section with y as the axis of symmetry. A property entry is required for convection, radiation, or thermal vector flux.
- TYPE = AREA3, AREA4, AREA6, or AREA8

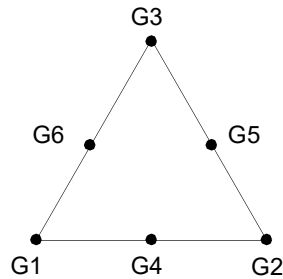
These types have three and four primary grid points, respectively, that define a triangular or quadrilateral surface and must be ordered to go around the boundary. A property entry is required for convection, radiation, or thermal vector flux.



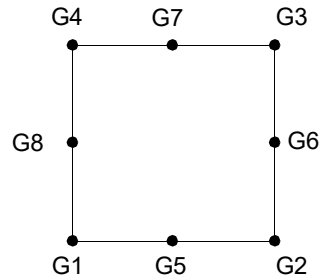
AREA3



AREA4



AREA6  
(Grid points G4 through G6 optional)



AREA8  
(Grid points G5 through G8 optional)

Figure 9-31 TYPE Examples

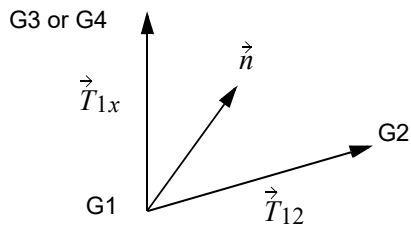


Figure 9-32 Normal Vector for CHBDYG Element of Types "AREAi"

The unit normal vector is given by

$$\vec{n} = \frac{(\vec{T}_{12} \times \vec{T}_{1x})}{|\vec{T}_{12} \times \vec{T}_{1x}|}$$

(G3 is used for triangles, and G4 is used for quadrilaterals.)

- For defining the front face, the right-hand rule is used on the sequence G1 to G2 to ... Gn of grid points.

5. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
6. All conduction elements to which any boundary condition is to be applied must be individually identified with one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP.
7. See Remark 9. of CHBDYE for application of boundary conditions using CHBDYG entries and a discussion of front and back faces.

## CHBDYP

## Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

Example:

CHBDYP	2	5	POINT	2	2	101		500	
	3	3			0.0	0.0	1.0		

Describer	Meaning
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all element identification numbers.)
PID	PHBDY property entry identification numbers. (Integer > 0)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	VIEW entry identification number for the front face. (Integer ≥ 0 or blank)
IVIEWB	VIEW entry identification number for the back face. (Integer ≥ 0 or blank)
G1, G2	Grid point identification numbers of grids bounding the surface. (Integer > 0)
G0	Orientation grid point. (Integer ≥ 0; Default = 0)
RADMIDF	RADM entry identification number for front face. (Integer ≥ 0 or blank)
RADMIDB	RADM entry identification number for back face. (Integer ≥ 0 or blank)
GMID	Grid point identification number of a midside node if it is used with the line type surface element.
CE	Coordinate system for defining orientation vector. (Integer ≥ 0; Default = 0)
Ei	Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank)

Remarks:

1. EID is a unique ID associated with a particular surface element as defined by the grid point(s).
2. The defaults for PID, TYPE, IVIEWF, IVIEWB, GO, RADMIDF, RADMIDB, CE, and Ei may be specified on the BDYOR entry. If a particular field is blank on both the CHBDYP entry and the BDYOR entry, then the default is zero.

3. TYPE specifies the kind of element surface; the allowed types are: “POINT,” “LINE,” “ELCYL,” “FTUBE,” and “TUBE.” For TYPE = “FTUBE” and TYPE = “TUBE,” the geometric orientation is completely determined by G1 and G2; the GO, CE, E1, E2, and E3 fields are ignored.

- TYPE = “POINT”

TYPE = “POINT” has one primary grid point, requires a property entry, and the normal vector  $\vec{V}_i$  must be specified if thermal flux is to be used.

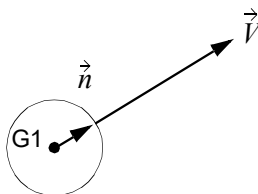


Figure 9-33 Normal Vector for CHBDYP Element of Type “POINT”  
(See Remarks 4. and 5.)

The unit normal vector is given by  $\vec{n} = \vec{V} / |\vec{V}|$  where  $\vec{V}$  is specified in the Ei field and given in the basic system at the referenced grid point. See Remarks 4. and 5. for the determination of  $\vec{V}$ .

- TYPE = “LINE,” “FTUBE,” or “TUBE”

The TYPE = “LINE” type has two primary grid points, requires a property entry, and the vector is required. TYPE = “FTUBE” and TYPE = “TUBE” are similar to TYPE = “LINE” except they can have linear taper with no automatic view factor calculations. GMID is an option for the TYPE = “LINE” surface element only and is ignored for TYPE = “FTUBE” and “TUBE”.

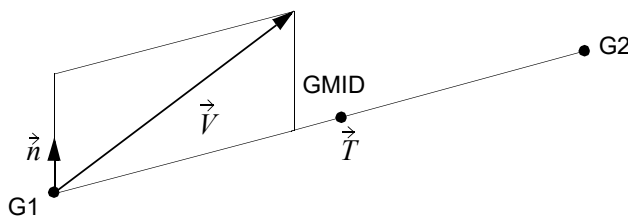


Figure 9-34 Normal Vector for CHBDYP Element with TYPE=“LINE”,  
TYPE=“FTUBE”, or TYPE=“TUBE” (See Remarks 4. and 5.)

The unit normal lies in the plane  $\vec{V}$  and  $\vec{T}$ , is perpendicular to  $\vec{T}$ , and is given by:

$$\vec{n} = \frac{\vec{T} \times (\vec{V} \times \vec{T})}{|\vec{T} \times (\vec{V} \times \vec{T})|}$$

- TYPE = “ELCYL”

TYPE = “ELCYL” (elliptic cylinder) has two connected primary grid points and requires a property entry. The vector must be nonzero.

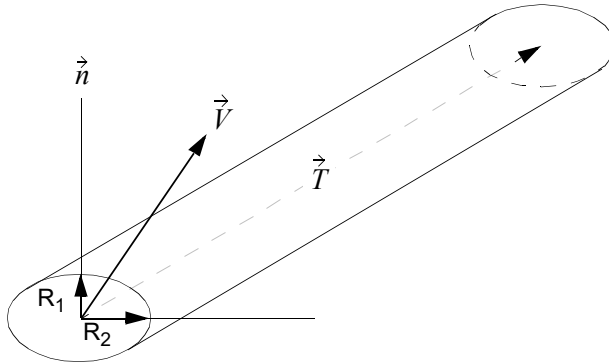


Figure 9-35 Normal Vector for CHBDYP Element of TYPE=“ELCYL”  
(See Remarks 4. and 5.)

The same logic is used to determine  $\vec{n}$  as for TYPE = LINE. The “radius”  $R_1$  is in the  $\vec{n}$  direction, and  $R_2$  is the perpendicular to  $\vec{n}$  and  $\vec{T}$  (see fields 7 and 8 of PHBDY entry).

4. For TYPE = “POINT,” TYPE = “LINE,” and TYPE = “ELCYL,” geometric orientation is required. The required information is sought in the following order:
  - If  $GO > 0$  is found on the CHBDYP entry, it is used.
  - Otherwise, if a nonblank CE is found on the CHBDYP continuation entry, this CE and the corresponding vectors E1, E2, and E3 are used.
  - If neither of the above, the same information is sought in the same way from the BDYOR entry.
  - If none of the above apply, a warning message is issued.
5. The geometric orientation can be defined by either GO or the vector E1, E2, E3.
  - If  $GO > 0$ :  
For a TYPE = “POINT” surface, the normal to the front face is the vector from G1 to GO. For the TYPE = “LINE” surface, the plane passes through G1, G2, GO and the right-hand rule is used on this sequence to get the normal to the front face. For TYPE = “ELCYL” surface the first axis of the ellipse lies on the G1, G2, GO plane, and the second axis is normal to this plane. For TYPE = “FTUBE” or “TUBE” surface, no orientation is required, and GO is superfluous.
  - If GO is zero:  
For a TYPE = “POINT” surface, the normal to the front face is the orientation vector. For the TYPE = “LINE” surface, the plane passes through G1, G2, and the orientation vector; the front face is based on the right-hand rule for the vectors G2-G1 and the orientation vector. For TYPE = “ELCYL” surface, the first axis of the ellipse lies on the G1, G2, orientation vector plane, and the second axis is normal to this plane.
6. The continuation entry is optional.



7. If the surface element is to be used in the calculation of view factors, it must have an associated VIEW entry.
8. All conduction elements to which any boundary condition is to be applied must be individually identified with the application of one of the surface element entries: CHBDYE, CHBDYG, or CHBDYP entries.
9. For RC network solver in thermal analysis, the G0, GMID, CE, E1, C2 and E3 are ignored.

## CHEXA Six-Sided Solid Element Connection

Defines the connections of the six-sided solid element with eight to twenty grid points or the six-sided solid shell element with eight grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

CHEXA	71	4	3	4	5	6	7	8	
	9	10	0	0	30	31	53	54	
	55	56	57	58	59	60			

Descriptor	Meaning	Type	Default
EID	Element identification number.	(0 < Integer < 100,000,000)	Required
PID	Property identification number of a PSOLID, PLSOLID or PCOMPLS entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

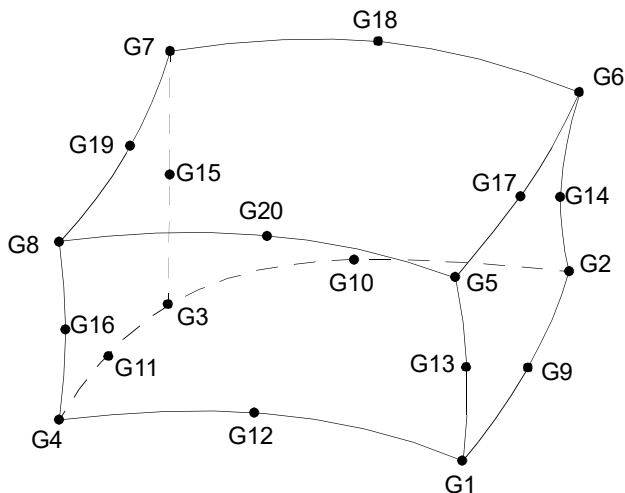


Figure 9-36 CHEXA Element Connection

**Remarks:**

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be given in consecutive order about one quadrilateral face. G5 through G8 must be on the opposite face with G5 opposite G1, G6 opposite G2, etc.
3. For Nastran conventional element, the edge points, G9 to G20, are optional. Any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero (as for G9 and G10 in the input example), the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element (with shear correction) in all cases.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. Either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system. The material coordinate system is defined on the PSOLID entry.
5. The second continuation is optional.
6. For nonhyperelastic and non-composite elements, the element coordinate system for the CHEXA element is defined in terms of the three vectors R, S, and T, which join the centroids of opposite faces.  
R vector joins the centroids of faces G4-G1-G5-G8 and G3-G2-G6-G7.  
S vector joins the centroids of faces G1-G2-G6-G5 and G4-G3-G7-G8.  
T vector joins the centroids of faces G1-G2-G3-G4 and G5-G6-G7-G8.

The origin of the coordinate system is located at the intersection of these vectors. The X, Y, and Z axes of the element coordinate system are chosen as close as possible to the R, S, and T vectors and point in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that if the R, S, and T vectors are described in the element coordinate system a 3 x 3 positive-definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

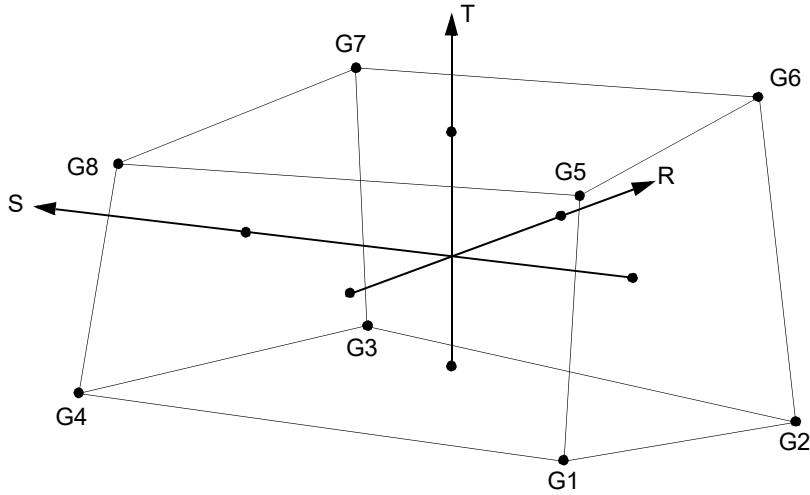


Figure 9-37 CHEXA Element R, S, and T Vectors

7. It is recommended that the edge points be located within the middle third of the edge.
8. For hyperelastic elements, the plot codes are specified under the CHEXAFD element name in [Item Codes](#).
9. By default, all the twelve edges of the element are considered straight unless, any of G9 through G20 are specified.
10. This element may be used:

Element Type	Property	BEHi Code	Integration Scheme	Solution Type
Solid	PSOLID	N/A	Refer to IN and ISOP	All Solution Sequences
	Nastran conventional element. Typically used for small strain, small / large displacement solutions.			
Below are advanced nonlinear elements.				
Solid	PSOLID+ PSLDN1	SOLID	L, Q, LRIH and QRI	SOL 400
	Typically used for large strain, large displacement solutions in conjunction with MATEP, MATS1, MATHE, etc.			
Solid Shell	PSOLID+ PSLDN1	SLCOMP	ASTN	SOL 400
	Typically used for bending dominated large strain, large displacement solutions.			
3D Solid Gasket	PSOLID+ PSLDN1	SLCOMP	L	SOL 400
	Used for gaskets with MATG.			

Element Type	Property	BEHi Code	Integration Scheme	Solution Type
Solid Composite	PCOMPLS	SLCOMP	L and Q	SOL 400 and SOL 101 to SOL 112
	Typically used for solid composite small strain / large strain / small displacement / large displacement solutions.			
Solid Shell Composite	PCOMPLS	SLCOMP	L and Q	SOL 400 and SOL 101 to SOL 112
	Typically used for bending dominated solid composite small strain / large strain / small displacement / large displacement solutions.			

- When this element is used as a three-dimensional eight-node solid shell element or three-dimensional eight-node composite solid shell element, the user should keep in mind that the layer orientation is required to be in the element T-direction, when specifying grid order.
- The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-38](#).

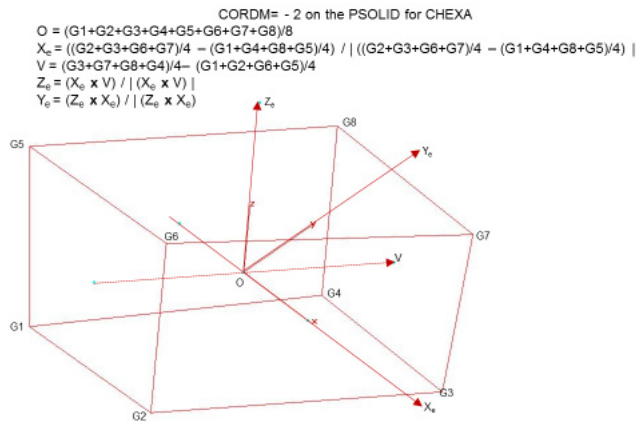


Figure 9-38 PSOLID for CHEXA

## CIFHEX

## Solid InterFace Cohesive Zone Modeling Element

Linear/Quadratic, twenty-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIFHEX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20			

Example:

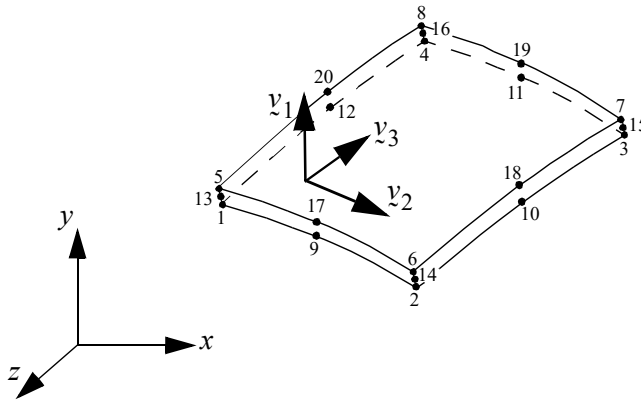
CIFHEX	700	701	456	357	882	889	443	447	
	162	911							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G8	Identification number of connected corner grid points. Required data for all eight corner grid points. (Unique Integer > 0)
G9-G12 G17-G20	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G13-G16	Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with twenty-noded hexahedral elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points  $G_i$  must be numbered as shown in the following figure. Specify either G1-G8, or all G1-G20.
3. The element is typically used to model the interface between different materials, where G1, G9, G2, G10, G3, G11, G4 and G12 correspond to one side (called the bottom) and G5, G17, G6, G18, G7, G19, and G20 correspond to the other side (called the top). The stress components are one normal and two shear tractions. When only G1-G8 are specified, the element is linear. When in addition to G1-G8, G9-G12, G17-G20 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.

- The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G4 and G5-G8 may coincide.



- This element does not support thermal load. No thermal strain will be generated.

## CIPENT

## Solid InterFace Cohesive Zone Modeling Element

Linear/Quadratic, fifteen-node, three-dimensional interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIPENT	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CIPENT	700	701	456	357	882	889	443	447	
--------	-----	-----	-----	-----	-----	-----	-----	-----	--

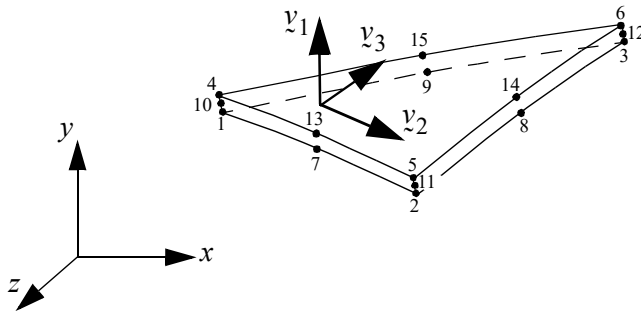
Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G6	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G7-G9 G13-G15	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G10-G12	Identification number of connected midside grid points. Optional data for midside grid points used only to make the element compatible with fifteen-noded pentahedral elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points  $G_i$  must be numbered as shown in the following figure. Specify either G1-G6 or all G1-G15.
3. The element is typically used to model the interface between different materials, where G1, G7, G2, G8, G3 and G9 correspond to one side (called the bottom) and G4, G13, G5, G14, G6, and G15 correspond to the other side (called the top). The stress components are one normal and one shear tractions. When only G1-G6 are specified, the element is linear. When in addition to G1-G6, G7-G9, G13-G15 are specified, the element is quadratic.
4. The corresponding deformations are relative displacements between the top and bottom edge of the element.



- The element is allowed to be infinitesimally thin; in which case edges defined by grids G1-G3 and G4-G6 may coincide.



- This element does not support thermal load. No thermal strain will be generated.

## CIFQDX

## Axisymmetric InterFace Cohesive Zone Modeling Element

Linear/Quadratic, eight-node, axisymmetric interface element used to simulate the onset and progress of delamination in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
CIFQDX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

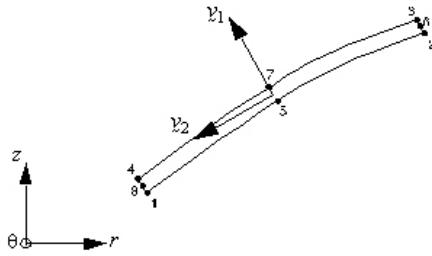
Example:

CIFQDX	700	701	456	357	882	889	443	447	
	1612	911							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G4	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G5, G7	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G6, G8	Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral axisymmetric elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points  $G_i$  must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where  $G_1$ ,  $G_5$ , and  $G_2$  correspond to one side (called the bottom) and  $G_3$ ,  $G_7$ , and  $G_4$  correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only  $G_1$ - $G_4$  are specified, the element is linear. When in addition to  $G_1$ - $G_4$ ,  $G_5$  and  $G_7$  are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges  $G_1$ - $G_5$ - $G_2$  and  $G_3$ - $G_7$ - $G_4$  may coincide.
6. The element must lie in the x-y plane of the basic system. Coordinate  $r$  is parallel to the x-basic and coordinate  $z$  is parallel to y-basic.



7. This element does not support thermal load. No thermal strain will be generated.

## CIFQUAD

## Planar InterFace Cohesive Zone Modeling Element

Linear/Quadratic, eight-noded planar interface element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
CIFQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

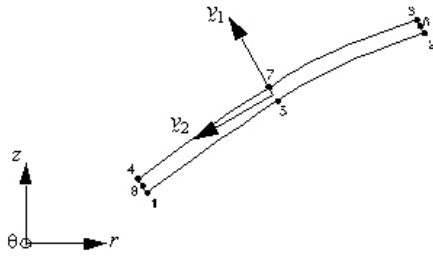
CIFQUAD	700	701	456	357	882	889	443	447	
	1612	911							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property number of a PCOHE entry. (Integer > 0)
G1-G4	Identification number of connected corner grid points. Required data for all four corner grid points. (Unique Integer > 0)
G5, G7	Identification number of connected edge grid points. Optional data for bottom and top edge grid points. (Unique Integer > 0)
G6, G8	Identification number of connected edge grid points. Optional data for side grid points used only to make the element compatible with eight-noded quadrilateral elements. (Unique Integer > 0)

Remarks:

1. Element identification numbers should be unique with respect to all other element ID's of any kind.
2. Grid points  $G_i$  must be numbered as shown in the following figure.
3. The element is typically used to model the interface between different materials, where  $G_1$ ,  $G_5$ , and  $G_2$  correspond to one side (called the bottom) and  $G_3$ ,  $G_7$ , and  $G_4$  correspond to the other side (called the top). The stress components are one normal and one shear traction. Then only  $G_1$ - $G_4$  are specified, the element is linear. When in addition to  $G_1$ - $G_4$ ,  $G_5$  and  $G_7$  are specified, the element is quadratic.
4. The corresponding deformation are relative displacements between the top and bottom edge of the element.
5. The element is allowed to be infinitesimally thin; in which case edges  $G_1$ - $G_5$ - $G_2$  and  $G_3$ - $G_7$ - $G_4$  may coincide.

- The element must lie in the x-y plane of the basic system.



- This element does not support thermal load. No thermal strain will be generated.

## CINTC Line Interface Element Connection

Defines a line interface element with specified boundaries.

Format:

1	2	3	4	5	6	7	8	9	10
CINTC	EID	TYPE							
LIST = (BID1(INTP1), BID2(INTP2),...,BIDn(INTPn))									

Example:

CINTC	1001	GRDLIS T							
LIST=(101,102(Q),-103(Q),104(L))									

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
TYPE	Connectivity. If TYPE = "GRDLIST" or blank (Default), the user will specify the boundaries via Bulk Data entry, GMBNDC. See Remark 2. (Character; Default = "GRDLIST")
BIDi	Boundary curve identification number, referenced to Bulk Data entry, GMBNDC. See Remark 2. (Integer ≠ 0)
INTPi	Interpolation scheme. (Character; Default = "L") INTP = "L": Linear interpolation; INTP = "Q": Quadratic interpolation.

Remarks:

1. Line interface element identification numbers must be unique with respect to all other line interface elements.
2. There must be at least two BIDi specified. If all BIDi are positive, by default, the degrees of freedom associated with the grids on the boundary represented by the first BID will be taken as the independent (n-set), and the degrees of freedom with the grids on the rest of boundaries are taken as the dependent (m-set). If there is a single negative BID, the degrees of freedom associated with the grids on the boundary represented by this BID will be taken as the independent (n-set), and the rest of the degrees of freedom with other boundaries are used as the dependent (m-set). If there are two or more negative BIDs, the degrees of freedom with the first negative one will be taken as the independent.
3. Forces of multipoint constraints may be recovered with the MPCFORCE Case Control command.
4. The m-set degrees of freedom specified on the boundary grids by this entry may not be specified by other entries that define mutually exclusive sets.

**CLOAD****Static Load Combination for Superelement Loads (Superposition)**

Defines a static load as a linear combination of previously calculated superelement loads defined by the LSEQ entry in nonlinear static analysis (SOLs 106 or 153).

Format:

1	2	3	4	5	6	7	8	9	10
CLOAD	CID	S	S1	IDV1	S2	IDV2	S3	IDV3	
	S4	IDV4	-etc.-						

Example:

CLOAD	25	1.0	25.0	10	-1.0	101	2.2-1	604	
	-62.0	62							

Describer	Meaning
CID	Combination identification number. (Integer > 0)
S	Scale factor. (Real)
Si	Scale factors. (Real)
IDVi	Identification numbers of load vectors (EXCITEID of a selected LSEQ entry) calculated for a superelement loads entry. (Integer > 0)

Remarks:

- The CLOAD entry must be selected in the residual solution subcases of the Case Control with CLOAD = CID and must be used if loads are applied to upstream superelements in SOL 106 or 153.
- The load vector defined is given by  $\{P\} = S \sum_i S_i \{P_{IDV_i}\}$
- The IDVi field refers to a previously calculated load vector for the superelement via the LSEQ approach. That is, a LOADSET keyword must have been selected in Case Control that in turn refers to one or more LSEQ entries in the Bulk Data Section. The IDVi refers to the EXCITEID of such LSEQ entries. For more details, see the Case Control commands [LSEQ](#) Bulk Data entry and the [LOADSET \(Case\)](#).
- In the CID or IDV fields, a CLOAD entry may not reference an identification number defined by another CLOAD entry.

## CMARKB2

## Two-Noded Marker Connectivity Definition

Defines a 2-noded marker beam element by means of connecting two grid points. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CMARKB2	ID	PID	G1	G2					

Example:

CMARKB2	7	1	9	10					
---------	---	---	---	----	--	--	--	--	--

Describer	Meaning
ID	Unique element number. (Integer > 0; Required)
PID	Property ID referring to a PMARKER entry. (Integer > 0; Required)
G1	Grid point number connectivity 1. (Integer > 0; Required)
G2	Grid point number connectivity 2. (Integer > 0; Required)

Remarks:

1. A CMARKB2 element may refer to two types of grid points:
  - a. Structural grid points, thus grid points that are part of the connectivity of an element
  - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.



## CMARKN1

## One-Noded Marker Connectivity Definition

Defines a 1-noded marker element on a grid point. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CMARKN1	ID	PID	G1						

Example:

CMARKN1	7	1	9						
---------	---	---	---	--	--	--	--	--	--

Describer	Meaning
ID	Unique element number. (Integer > 0; Required)
PID	Property ID referring to a PMARKER entry. (Integer > 0; Required)
G	Grid point number. (Integer > 0; Required)

Remarks:

1. A CMARKB2 element may refer to two types of grid points:
  - a. Structural grid points, thus grid points that are part of the connectivity of an element
  - b. Free grid points in space. These grid points do not have mass associated with them. The motion of these grid points is specified by the PMARKER property
2. The ID must be unique in the model and may not be used as structural element ID.

## CMASS1      Scalar Mass Connection

Defines a scalar mass element.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS1	EID	PID	G1	C1	G2	C2			

Example:

CMASS1	32	6	2	1					
--------	----	---	---	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
PID	Property identification number of a PMASS entry. ( $\text{Integer} > 0$ ; Default = EID)
G1, G2	Geometric grid or scalar point identification number. ( $\text{Integer} \geq 0$ )
C1, C2	Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point)

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS3 entry.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2) must not be coincident.
- For a discussion of the scalar elements, see the *MSC Nastran Reference Guide*, Section 5.6.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
- Scalar elements input coupled mass matrices when the second pair of fields is entered. When uncoupled point masses are desired input only the first pair of fields. When a coupled mass matrix is requested the submatrix added has M on the diagonal, and -M on the off-diagonal. The element is not checked for internal constraints, which is the user's responsibility if desired. There are instances where elements with internal constraints are desired, although not frequently. To identify the presence of internal constraints caused by coupled mass, inspect GPWG output, OLOAD output due to GRAV loads, and rigid body modes of free structures. Some forms of coupled mass will cause coupling of rigid body translational mass terms in GPWG output, and poor rigid body modes in modal analysis.

## CMASS2      Scalar Mass Property and Connection

Defines a scalar mass element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS2	EID	M	G1	C1	G2	C2			

Example:

CMASS2	32	9.25	6	1					
--------	----	------	---	---	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ )
M	Value of the scalar mass. (Real)
G1, G2	Geometric grid or scalar point identification number. ( $\text{Integer} \geq 0$ )
C1, C2	Component number. ( $0 \leq \text{Integer} \leq 6$ ; blank or zero if scalar point)

Remarks:

- Scalar points may be used for G1 and/or G2, in which case the corresponding C1 and/or C2 must be zero or blank. Zero or blank may be used to indicate a grounded terminal G1 or G2 with a corresponding blank or zero C1 or C2. A grounded terminal is a point with a displacement that is constrained to zero. If only scalar points and/or ground are involved, it is more efficient to use the CMASS4 entry.
- Element identification numbers should be unique with respect to all other element identification numbers.
- The two connection points (G1, C1) and (G2, C2) must be distinct. Except in unusual circumstances, one of them will be a grounded terminal with blank entries for Gi and Ci.
- For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
- A scalar point specified on this entry need not be defined on an SPOINT entry.
- If Gi refers to a grid point then Ci refers to degrees-of-freedom(s) in the displacement coordinate system specified by CD on the GRID entry.
- See Remark 7 for [CMASS1](#).

## CMASS3 Scalar Mass Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS3	EID	PID	S1	S2					

Example:

CMASS3	13	42	62						
--------	----	----	----	--	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PMASS entry. (Integer > 0; Default = EID)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

## CMASS4      Scalar Mass Property and Connection to Scalar Points Only

Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMASS4	EID	M	S1	S2					

Example:

CMASS4	23	14.92		23					
--------	----	-------	--	----	--	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
M	Scalar mass value. (Real)
S1, S2	Scalar point identification numbers. (Integer ≥ 0; S1 ≠ S2)

Remarks:

1. S1 or S2 may be blank or zero, indicating a constrained coordinate. This is the usual case.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one scalar mass element may be defined on a single entry.
4. For a discussion of the scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. A scalar point specified on this entry need not be defined on an SPOINT entry.

**CMREBAI** Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR with INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Unlike the CMREBAR element, this element can span multiple matrix CHEXA elements (for example, could be allied to all elements on the bottom of a flat surface modeled with several layers of CHEXA matrix elements through the thickness).

**Format:**

1	2	3	4	5	6	7	8	9	10
CMREBAI	ID	IP	G1	G2	G3	G4	G5	G6	
	G7	G8							

**Example:**

CMREBAI	100	2	31	32	88	87			
CMREBAI	250	8	1	2	3	4	5	6	
	7	8							

Describer	Meaning
ID	Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)
IP	Property identification of a matching PMRBAR entry. (Integer; Required; no Default)
G1-G4	Grid point identification numbers of the four corner points. (Integer > 0; all unique)
G5-G8	Grid point identification numbers of the four mid-side nodes similar to G5-G8 for the CQUAD8 element. (Integer > 0, or blank. If any of G5-G8 are not blank, all nodes in the range G5 to G8 must be defined and must be unique among the range G1-G8.)

**Remarks:**

1. This entry makes use of Marc's REBAR and INSERT capabilities for membrane element types 147 and 148. If G5-G8 are blank, Marc element 147 with 4 nodes is used. If G5-G8 are not blank, Marc element 148 with 4 corner nodes and 4 mid-side nodes is used. Entry CMREBAR makes use of Marc's REBAR capability (without INSERT) and uses rebar elements 23 and 146.
2. The grid ID's do not have to correspond to those of any matrix CHEXA element. The PMREABI entry is used to describe the matrix CHEXA elements that these rebar elements will be inserted into.
3. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforced composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea

of rebar layer concept contains (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by embedding membrane rebar elements into solid matrix elements using Marc's INSERT option. Membrane rebar elements types 147 and 148 are available with this option. They are empty 4-node or 8-node quadrilaterals. You can place reinforcing cord layers within these empty elements. These elements are then embedded into their corresponding solid elements representing the matrix materials. Independent meshes can be used for the rebar membrane elements and the matrix elements. Marc's INSERT option is automatically invoked by the CMREBAI elements and used to enforce the compatibility between two different meshes.

4. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSERT option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
5. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
6. See MPREBAI for additional information and figures defining these rebar elements.
7. Only CHEXA elements may be used for the matrix elements.

**CMREBAR**

Defines Rebar Elements and Matching "Matrix" Solid Elements using the Marc REBAR without INSERT Method in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. CMREBAR elements require that the rebar be placed in matching CHEXA matrix elements on a one-to-one basis. For a similar capability where the rebar can span multiple CEHXA matrix elements, see the CMREBAI entry.

Format:

1	2	3	4	5	6	7	8	9	10
CMREBAR	ID	IP	ID2	IDD	ID22				

Example:

CMREBAR	100	2	1						
CMREBAR	1001	50	101	1100	200				

Describer	Meaning
EID	Rebar element ID should be distinct between all element ID's. (Integer; Required; no Default)
IP	Property identification of a matching PMRBAR entry. (Integer; Required; no Default)
ID2	CHEXA (8 node or 20 node) "matrix" element that the rebar will be added to. (Integer; Required; no Default)
IDD	If more than one rebar element in a continuous range is to be added to a continuous range of rebar elements, IDD represents the first rebar element identification number in the range. (Integer; Required; Default = 0, IDD must be larger than ID)
ID22	If more then one rebar element in a continuous range is to be added to a continuous range of "matrix" elements, ID22 represents the last CHEXA matrix element identification number in the range. (Integer; Required; Default = 0, ID22 must be larger then ID2)

Remarks:

1. This entry makes use of Marc's REBAR capability for element types 23 and 146. Entry CMREBAI makes use of Marc's REBAR and INSERT capabilities and uses membrane rebar elements 147 and 148.
2. If IDD is entered, IDD2 must also be entered and the difference IDD-ID must match the difference IDD2-ID2.



3. The same grid ID's are used to define element ID and ID2. Similarly the same grid ID's define the other elements if a range of elements are used. Element ID is an "empty shell element" and up to 5 rebar layers (each containing multiple rebar) are placed in the empty shell as specified by the PMREBAR property entries.
4. Cord-reinforced composites are characterized by a group of reinforcing cords with arbitrary spatial orientations embedded in various matrix materials. The different constituents may have different mechanical properties. Two typical examples of the cord-reinforced composites are tires and cord-reinforced concretes. In modeling such materials, the rebar technique is very useful. The basic idea of rebar layer concept contains that (1) the reinforcing cords and the matrix materials of the composites are represented independently by different types of elements along with different constitutive models, (2) the reinforcing cords within the elements modeling these cords (the so-called rebar elements) are assumed to be in the form of layers, and (3) the rebar elements are then embedded into the matrix elements. The compatibility between the cord elements and the matrix elements is enforced by superimposing solid rebar elements on corresponding solid matrix elements using the same element connectivity. The rebar elements are empty 8-node or 20-node CHEXA elements derived from the matching matrix elements. The reinforcing cord layers are placed within the elements. Each solid rebar element is then superimposed on a solid matrix element. The two elements share the same space with the same element connectivity (therefore, the same element nodes). The compatibility condition between the reinforcements and the matrix materials is then automatically enforced.
5. The major difference between the CMREBAR and CMREBAI elements is that CMREBAR elements share the same grids as the matrix CHEXA elements while CMREBAI elements typically have different grid ID's than the matrix CHEXA elements. Marc's INSERT option automatically adds tyings (MPC's) between the CMREBAI grids and the CHEXA grids.
6. CMREBAI elements are preferred over the CMREBAR elements when re-meshing is involved.
7. See PMREBAR for additional information and figures defining these rebar elements.
8. Only CHEXA elements may be used for the matrix elements.

## Entries CO - CY

### COHESIV

Defines Data for Cohesive Materials in SOL 600

This option allows you to define material properties for interface elements, that may be used to simulate the onset or progress of delamination, and to associate these material properties with a list of element numbers. The cohesive material is defined using the cohesive energy (also called critical energy release rate), that equals the area below the equivalent traction versus equivalent relative displacement curve. The shape of this curve can be bilinear, exponential, or combined linear-exponential. Mixed mode delamination is incorporated by converting the normal and shear components of the relative displacements into an equivalent using the normal shear weighting factor.

As an alternative to the standard linear, exponential, and liner-exponential model, the user can also utilize this option to trigger the call to the UCOHESIVE user-subroutine. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
COHESIV	MID	ITYPE	IACT	NAME					
	EN	COD	MOD	NSW	SNW	DECAY			
	VISC	RATE	STIFF						
	ISET								

Example:

COHESIV	101	1	1	material	number10	1			
	2000.	.01	.015	23.	24.	1.			
	.05	0.0	1.0						
	101								

Describer	Meaning
MID (3,1)	Material ID - Must match a MATXXX entry. (Integer; no Default)
ITYPE (3,2)	Type of cohesive model. (Integer > 0; Default = 1) 1 Bilinear model (Default) 2 Exponential modes 3 Combined linear-exponential model -1 User-defined using user-subroutine UCOHESIVE
IACT (3,3)	Option to deactivate elements and output to t16 file. (Integer; Default = 0) 0 Elements remain active regardless of the damage level. 1 Deactivate the elements if the maximum damage in all the element integration points has been reached. Do not remove the elements from the t16 file.

Describer	Meaning
	2 Deactivate the elements if the maximum damage in all the element integration points has been reached. Remove the elements from the t16 file.
NAME (3,5)	Name of the material. (Character up to 40 characters; no Default; optional entry)
EN (4,1)	Cohesive energy. (Real; no Default; Required value)
COD (4,2)	Critical opening displacement. (Real; no Default; Required value)
MOD (4,3)	Maximum opening displacement (Bilinear model only). (Real; no Default)
NSW (4,4)	Normal-shear weighting factor, beta. (Real; no Default; Required value)
SNW (4,5)	Shear-normal weighting factor. (Real; Default = 1.0)
DECAY (4,6)	Exponential decay factor. (Linear-Exponential model only).(Real; Default = 1.0)
VISC (4,7)	Viscous energy dissipation factor (zero implies no viscous energy dissipation). (Real; Default = 0.0)
RATE (4,8)	Relative displacement reference rate. Only used if viscous energy dissipation is by setting VISC to a non-zero value. (Real; Default = 0.0)
STIFF (4,9)	Stiffening factor in compression. (Real; Default = 1.0)
ISET (5)	ID of a SET3 entry defining the elements associated with this cohesive material. (Integer, no Default, Required value)

#### Remarks:

1. Values in parenthesis (i,j) refer to Marc's COHESIVE entry. (Datablock, field)
2. For solid elements, this entry may be used to add a "layer" or interface between the solid elements (CHEXA, CTETRA or CPENTA). This interface can fail or delaminate depending on the properties entered. It is used in conjunction with Marc element types 188, 192 and 193 and provides the material properties for these element types.
3. For shell elements (CQUADi, CTRIAi) this entry may be used to add a layer between the edges of adjacent shells. It is used in conjunction with Marc element types 186 and 187 and provides the material properties for these element types.
4. Solid or shell elements with a COHESIV MID will automatically be assigned Marc element types 186, 187, 188, 189, 192, 193 as appropriate to the type and number of grids defined for that element. These solid elements must either only have corner nodes or have the full parabolic number of nodes (for example, CHEXA must either have 8 nodes or 20 nodes).
5. For SOL 600, cohesive behavior is not available for axisymmetric or plane strain analyses.
6. MID must not be used by any other material such as MAT1, MAT2, etc.
7. All continuation lines are required.

**COHFRIC****Cohesive Friction**

Allows friction and sticking during tensile conditions at the coupling surface. Use SOL700 only.

Format:

COHFRIC	CID	MAXSTRS	FRIC	REFVEL					
---------	-----	---------	------	--------	--	--	--	--	--

Example:

COHFRIC	112	8e+10	8e+5	2					
---------	-----	-------	------	---	--	--	--	--	--

Describer	Meaning
CID	Unique number of a COHFRIC entry. (Integer>0; required)
MAXSTRS	Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real ≥ 0.0; default=0.0)
FRIC	Friction stress under tensile conditions. (Real ≥ 0.0; default=0.0)
REFVEL	Reference value for velocity. (Real ≥ 0.0; default=0.0)

Remarks:

1. If the cohesive friction parameters is uniform across the coupling surface then DYPARAM, COHESION can be used instead.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals  $Fric \times \min\left(1, \frac{V_{REL, tangential}}{REFVEL}\right)$ . This is a viscous-like friction law.
3. This shear force opposes the relative tangential movement along the coupling surface.

**COMPUDS**

Orthotropic Failure Model that allows to model Property Degradation

Defines an orthotropic failure model for shell composites specified by a user subroutine. Used in SOL700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
COMPUDS	MID	GROUP	UNAME						+
+	E1	E2	N12	G12	G1,Z	G2,Z	RHO		+
+	NV	S	ALPHA	TRSFAL	F12				+
+	XT	XT	YT	YC	PFD	VALUE	PFDST		+
+	FBTEN	FBCOM	MXTEN	MXCOM	MXSHR				+
+	PRDFT	PRDFC	PRDMT	PRDMC	PRDSH				+

**Example:**

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE excomp1 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
In Bulk Data:
```

COMPUDS	1	EXCOMP1							+
+	30E6	1.0E6	0.3	2.0E6	3.0E6	1.5E6	0.056		+
+	5	100							
+	200	150	100	110					

Field	Contents
MID	Unique material identification number. (Integer > 0; required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; required).
UNAME	User subroutine name associated with the entry. (Character; default = blank).
E1	Modulus of elasticity in longitudinal direction (also defined as fiber direction or one-direction). (Real > 0.0; required)
E2	Modulus of elasticity in lateral direction (also defined as matrix direction or two-direction). (Real > 0.0; required)
N12	Poisson's ratio ( $\nu$ / 1 for uniaxial loading in one-direction). Note that $\nu_{21} = \nu_{12} E_2 / E_1$ for uniaxial loading in two-direction is related to $\nu_{12}$ , E1, E2 by the relation $\nu_{12} E_2 = \nu_{21} E_1$ . (Real > 0.0; required)
G12	In-plane shear modulus R. (Real > 0.0; required)
G1,Z	Transverse shear modulus for shear in 1-Z plane. (Real > 0.0; default= G12)
RHO	Mass density. (R > 0.0; required)



Field	Contents
NV	Number of additional history variables for a user model. See Remark 8. ( $0 < \text{Integer} < 10$ ; default=0)
S	Failure stress for in-plane shear. (Real $\geq 0.0$ ; default=0.0)
ALPHA	Nonlinear shear coefficient. (Real $\geq 0.0$ ; default=0.0)
TRSFAIL	Transverse shear failure. (Character; default = SUBL)
	ELEM: Failure if element fails
	SUBL: Failure if sublayer fails
F12	Interaction term in Tsai-Wu theory. (Real; default=0.0)
XT, XC	Tensile compressive failure stress in the large structural direction. (Real $> 0.0$ ; required)
YT, YC	Tensile compressive failure stress in the lateral direction. (Real $> 0.0$ ; required)
PFD	Post-failure degradation model. (Character; default = STEPS)
	STEPS: Degrade stresses by time steps
	TIME: Degrade stresses by time
	VELOC: Degrade stresses by velocity
VALUE	Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Integer or Real; default=100)
PFDST	Post-failure degradation start. (Character; default=INDV)
	INDV Stresses are degraded per distinct failure mode.
	ALL Stresses are degraded if all elastic constants are zero.
FBTEN,FBCO N,MXTEN,M XCOM,MXS HR	Failure modes in fiber, matrix direction, and theory failure. (Character; default=blank)
PRDFT	Property degradation due to fiber-tension failure. (Integer; default=1111)
PRDFC	Property degradation due to fiber-compression failure. (Integer; default=1010)
PRDMT	Property degradation due to matrix-tension failure. (Integer; default=0110)
PRDMC	Property degradation due to matrix-compression failure. (Integer; default=0110)
PRDSH	Property degradation due to in-plane shear failure. (Integer; default=0001)

Remarks:

11. Please check MAT8 and MAT8A cards about the details of each field. The field definition is identical as MAT8 and MAT8A.
12. UNAME can be:

Subroutine Name	Function
EXCOMP	User defined orthotropic material models for shell composite

**CONCTL** Parameter SWLDPRM override for CFAST, CSEAM, and CWELD Connector Elements

This entry provides local connector search algorithm control to override SWLDPRM values.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
CONCTL	SET3ID		PARAM1	VALUE1	PARAM2	VALUE2	PARAM3	VALUE3	
	PARAM4	VALUE5	PARAM4	VALUE5	-etc.-				

Example:

CONCTL	75		PROJTOL	0.2	PRTSW	1	GSMOVE	3	
	NREDIA	2							
CONCTL	83		MOVGAB	1					

Describer	Meaning
SET3ID	The ID of a SET3 entry using (DES= ELEM) to identify a group of connector elements. (Integer>0)
PARAMi	Name of the connector parameter. Allowable names are listed in Table 8-62 of SWLDPRM. (Character)
VALUEi	Value of the parameter. See Table 8-62 of SWLDPRM (Real or Integer)

Remarks:

- Multiple CONCTL entries are allowed and ALL appearing will be used. If the user supplies a PARAMi, VALUEi that refers to specific Element on one CONCTL entry and a same PARAMi with VALUEj that refers to the same specific Element on another CONCTL, a fatal message will be issued.
- For a given connector element ID, this entry will override the global default set by the SWLDPRM entry.
- The parameter CHKRUN can only be set on SWLDPARM and will be ignored on CONCTL entries.
- A user fatal will be issued if DES on the SET3 entry is not "ELEM".



## CONM1 Concentrated Mass Element Connection, General Form

Defines a 6 x 6 symmetric mass matrix at a geometric grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM1	EID	G	CID	M11	M21	M22	M31	M32	
	M33	M41	M42	M43	M44	M51	M52	M53	
	M54	M55	M61	M62	M63	M64	M65	M66	

Example:

CONM1	2	22	2	2.9	6.3				
	4.8	28.6							
		28.6						28.6	

Descriptor	Meaning
EID	Unique element identification number. ( $0 < \text{Integer} < 100,000,000$ ). See Remark 2.
G	Grid point identification number. ( $\text{Integer} > 0$ ). See Remark 3.
CID	Coordinate system identification number for the mass matrix. ( $\text{Integer} \geq 0$ )
Mij	Mass matrix values. (Real)

Remarks:

1. For a less general means of defining concentrated mass at grid points, see the CONM2 entry description.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Unlike the CONM2 entry, the CONM1 entry does not allow for the specification of concentrated mass at harmonic structural grid points. Therefore, grid point G must necessarily be a non-harmonic structural grid point.

**CONM2** Concentrated Mass Element Connection, Rigid Body Form

Defines a concentrated mass at a grid point.

Format:

1	2	3	4	5	6	7	8	9	10
CONM2	EID	G	CID	M	X1	X2	X3		
	I11	I21	I22	I31	I32	I33			

Example:

CONM2	2	15	6	49.7					
	16.2		16.2			7.8			

Describer	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ ). See Remark 1.
G	Grid point identification number. ( $\text{Integer} > 0$ ). See Remark 7.
CID	Coordinate system identification number. For CID of -1; see X1, X2, X3 below. ( $\text{Integer} \geq -1$ ; Default = 0). See Remarks 4., 6. and 7.
M	Mass value. (Real). See Remarks 5. and 7.
X1, X2, X3	Offset distances from the grid point to the center of gravity of the mass in the coordinate system defined in field 4, unless CID = -1, in which case X1, X2, X3 are the coordinates, not offsets, of the center of gravity of the mass in the basic coordinate system. (Real). See Remarks 4., 5., 6. and 7.
Iij	Mass moments of inertia measured at the mass center of gravity in the coordinate system defined by field 4. If CID = -1, the basic coordinate system is implied. (Real). See Remarks 4., 5. and 7.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For a more general means of defining concentrated mass at grid points, see the CONM1 entry description.
3. The continuation is optional.
4. If CID = -1, offsets are internally computed as the difference between the grid point location and X1, X2, X3. The grid point locations may be defined in a nonbasic coordinate system. In this case, the values of Iij must be in a coordinate system that parallels the basic coordinate system.
5. The form of the inertia matrix about its center of gravity is taken as:

$$\begin{bmatrix} M & & & & & & & & \\ & M & & & & & & & \\ & & \text{symmetric} & & & & & & \\ & & & I_{11} & & & & & \\ & & & -I_{21} & & I_{22} & & & \\ & & & -I_{31} & & -I_{32} & & I_{33} & \\ & & & & & & & & \end{bmatrix}$$

where

$$M = \int \rho dV$$

$$I_{11} = \int \rho(x_2^2 + x_3^2) dV$$

$$I_{22} = \int \rho(x_1^2 + x_3^2) dV$$

$$I_{33} = \int \rho(x_1^2 + x_2^2) dV$$

$$I_{21} = \int \rho x_1 x_2 dV$$

$$I_{31} = \int \rho x_1 x_3 dV$$

$$I_{32} = \int \rho x_2 x_3 dV$$

and  $x_1, x_2, x_3$  are components of distance from the center of gravity in the coordinate system defined in field 4. The negative signs for the off-diagonal terms are supplied automatically. A warning message is issued if the inertia matrix is non-positive definite, since this may cause fatal errors in dynamic analysis modules.

6. If  $CID \geq 0$ , then X1, X2, and X3 are defined by a local Cartesian system, even if CID references a spherical or cylindrical coordinate system. This is similar to the manner in which displacement coordinate systems are defined.
7. MSC Nastran contains two kinds of structural grid points, namely, non-harmonic structural grid points and harmonic structural grid points. The former have the standard three translational and three rotational degrees of freedom while the latter have three symmetric components and three anti-symmetric components dependent on the harmonic value.

Harmonic grid points are identified automatically as the grid points listed on CQUADX and CTRIAX element entries that are associated with PAXSYMH entries.

For a harmonic grid point, the mass value  $M$  is the total mass. This value is not to be multiplied by  $2\pi$ . The mass matrix computed for harmonic grids is dependent on the harmonic value associated with the PAXSYMH entry and will be automatically determined by the code. Only harmonic values of 0 and 1 have contributions to grid point weight generator type calculations.

Any values specified in the CID, X1, X2, X3, I11, I21, I22, I31, I32, or I33 fields are ignored for a harmonic grid.

**CONROD****Rod Element Property and Connection**

Defines a rod element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CONROD	EID	G1	G2	MID	A	J	C	NSM	

Example:

CONROD	2	16	17	4	2.69				
--------	---	----	----	---	------	--	--	--	--

Describer	Meaning
EID	Unique element identification number. (0 < Integer < 100,000,000)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)
MID	Material identification number. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient for torsional stress determination. (Real)
NSM	Nonstructural mass per unit length. (Real)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
4. For RC network solver in thermal analysis, the J, C and NSM are ignored.

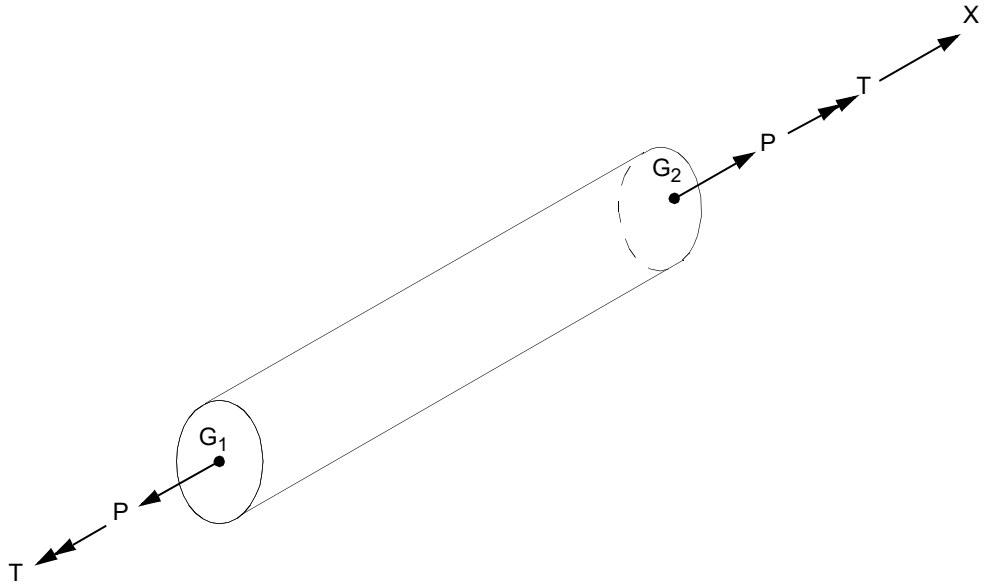


Figure 9-39 CONROD Element Forces and Moments

## CONTRLT Thermal Control Element for Heat Transfer Analysis

Defines the control mechanism for QVECT, QVOL, QBDY3, RADBC, CONV, and CONVM in heat transfer analysis (SOL 159 and SOL 400 with analysis=htran).

Format:

1	2	3	4	5	6	7	8	9	10
CONTRLT	ID	SENSOR	SFORM	CTYPE	PL	PH	PTYPE	PZERO	
	DT	DELAY	TAUC				FCTMIN	FCTMAX	
	GAIN1	GAIN2	GAIN3						

Example:

CONTRLT	100	20	T	TSTAT	68.	73.	7	1.	
	0.1	0.01	0.				0.	1.	
	0.	0.	0.						

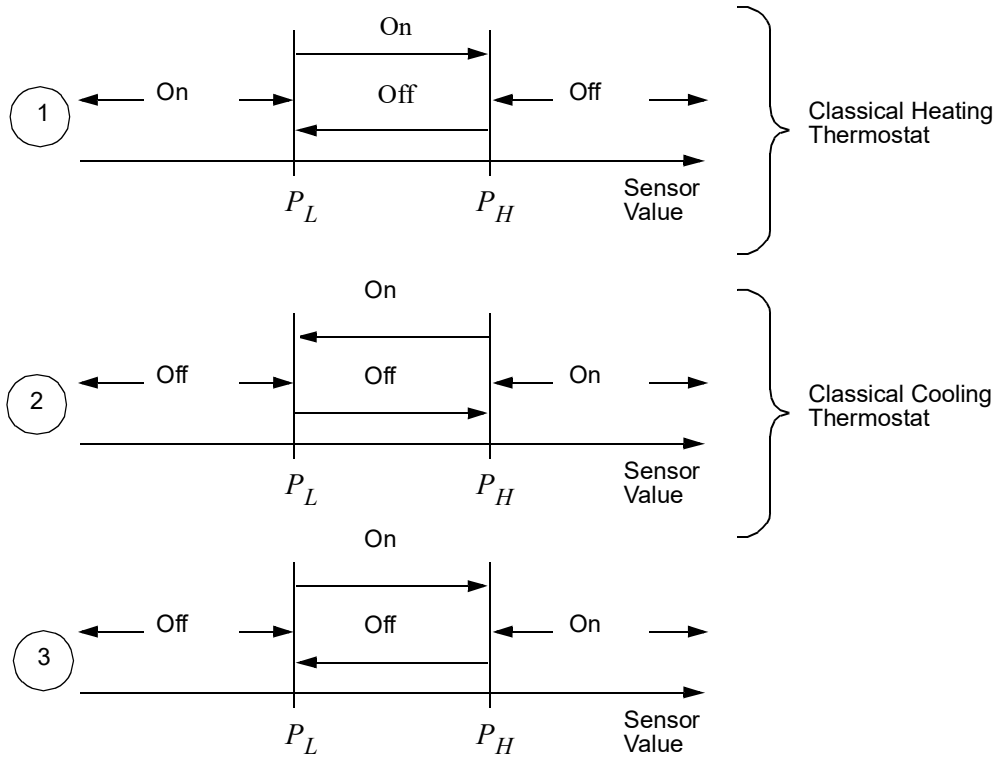
Describer	Meaning
ID	Controller identification node. See Remark 1. (Integer > 0; Required)
SENSOR	Grid or scalar point ID of sensor. See Remark 2. (Integer > 0; Required)
SFORM	Sensor output form. See Remark 3. (Character, T; Default = T)
CTYPE	Control type. See Remark 4. (Character, Default TSTAT)
PL	Minimum temperature. For PTYPE 9 through 11, set temperature. See Remark 5. (Real; Required)
PH	Maximum temperature. For PTYPE 9 through 11, unused. See Remark 5. (Real; Required)
PTYPE	Process type. See Remark 5. (Integer 1 through 11, 1 through 6 already defined; no Default)
	7      Bang-bang (a thermostat that is either on or off, a special case of a bang-bang controller)
	8      Proportional
	9      Steady-state (special routine for steady state models)
	10     Classical PID (proportional-integral-differential) and does not include the time derivative of the set point )
	11     Standard PID (Includes the time derivative of the set point)
PZERO	Initial controller value. See Remark 4. (0. < Real < 1.; Default = 0)
DT	Monitoring time interval or sampling period. (Real > 0.; Default = 0)

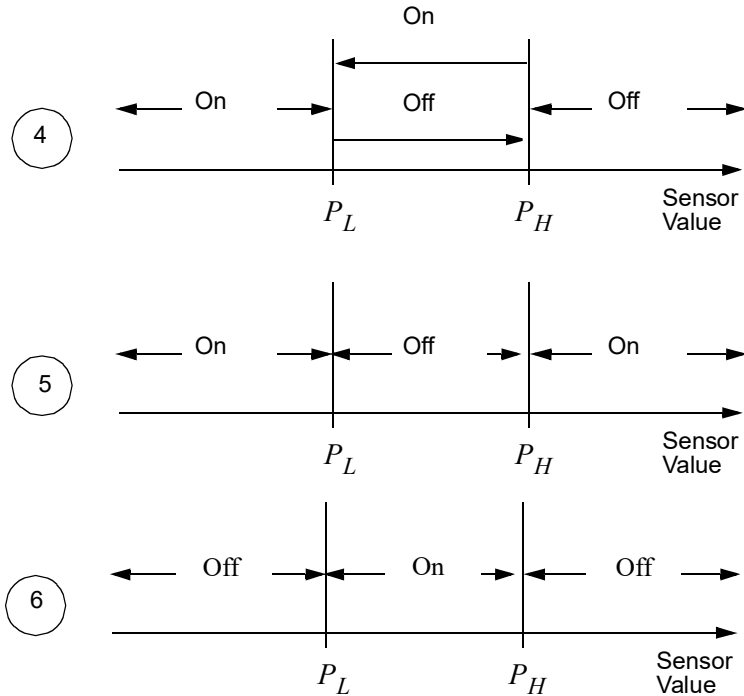
Describer	Meaning
DELAY	Time delay after the switch is triggered or time for delayed control action in PID control. (Real > 0.; Default = 0.0)
TAUC	Decay time constant for actuator response. (Real > 0.; Default = 0.0)
FCTMIN/MAX	Minimum factor/Maximum factor on load to apply. (Real; Default = 0.0)
GAINx	Gain variables for heat controller. Use GAIN1 for P value, GAIN2 for I value, and GAIN3 for D value. (Real; Default = 0.0)

**Remarks:**

1. The CONTRLT ID is referenced by CNTRLND entry identified on any of the CONV, CONVM, RADBC, QVECT, QVOL, and QBDY3 Bulk Data entries. If any grid or scalar point ID is the same as the CONTRLT ID, then the combined logic associated with the controller and the control node will be in force for the LBC referenced. Any number of CONTRLT statements may exist in a single model.
2. Sensor point, where a feedback temperature or rate of change of temperature is measured. May be a dependent DOF in a MPC relationship.
3. Sensor output may only be temperature (T)
4. Control type can only be TSTAT. The PZERO field cannot have any other value but 0.0 or 1.0.
5. The upper and lower limit values (PL and PH) define a dead band for a thermostat. The available thermostat controller (TSTAT) formats are (PTYPE = 1 through 6).
6. RC network solver only supports PTYPE=1 and PTYPE=2 thermostat controller, SFORM, CTYPE, PZERO, DT, DELAY, TAUC and TA8 are ignored.







7. The last 5 parameters are for RC Network solver only.
8. RC Network solver only supports controllers (thermostats) on source data. This entry can be used in either steady-state (static) or transient state (dynamic). For more details about these parameters, please reference the *MSC SINDA User's Guide and Library Reference*.

## CONV Heat Boundary Element Free Convection Entry

Specifies a free convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
	TA5	TA6	TA7	TA8					

Example:

CONV	2	101	3	201	301				
------	---	-----	---	-----	-----	--	--	--	--

Describer	Meaning
EID	CHBDYG, CHBDYE, or CHBDYP surface element identification number. (0 < Integer < 100,000,000)
PCONID	Convection property identification number of a PCONV entry. (Integer > 0)
FLMND	Point for film convection fluid property temperature. (Integer ≥ 0; Default = 0)
CNTRLND	Control point for free convection boundary condition. (Integer ≥ 0; Default = 0)
TAi	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2 through TA8; Default for TA2 through TA8 is TA1.)

Remarks:

- The basic exchange relationship can be expressed in one of the following forms:
  - $q = H \cdot (T - T_{AMB})^{EXPF} (T - T_{AMB})$ , CNTRLND = 0
  - $q = (H \cdot u_{CNTRLND})(T - T_{AMB})^{EXPF} (T - T_{AMB})$ , CNTRLND ≠ 0
  - $q = H(T^{EXPF} - T_{AMB}^{EXPF})$ , CNTRLND = 0
  - $q = (H \cdot u_{CNTRLND})(T^{EXPF} - T_{AMB}^{EXPF})$ , CNTRLND ≠ 0

EXPF is specified on the PCONV entry.  
(See [PCONV](#) entry for additional clarification of forms.)
- The continuation entry is not required.
- CONV is used with an CHBDYi (CHBDYG, CHBDYE, or CHBDYP) entry having the same EID.

4. The temperature of the film convection point provides the look up temperature to determine the convection film coefficient. If  $FLMND=0$ , the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperature, or the ambient temperature, as defined in the FORM field of the PCONV Bulk Data entry.
5. If only one ambient point is specified then all the ambient points are assumed to have the same temperature. If midside ambient points are missing, the temperature of these points is assumed to be the average of the connecting corner points.
6. See the Bulk Data entry, [PCONV](#), for an explanation of the mathematical relationships involved in free convection and the reference temperature for convection film coefficient.
7. RC network solver only supports CNTRLND node defined by SPC or SPCD entries, FLMND is ignored.

## CONVM Heat Boundary Element Forced Convection Entry

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONVM	EID	PCONID	FLMND	CNTMDOT	TA1	TA2	Mdot		

Example:

CONVM	101	1	201	301	20	21			
-------	-----	---	-----	-----	----	----	--	--	--

Describer	Meaning
EID	CHBDYP element identification number. (0 < Integer < 100,000,000)
PCONID	Convection property identification number of a PCONVM entry. (Integer > 0)
FLMND	Point used for fluid film temperature. (Integer ≥ 0; Default = 0)
CNTMDOT	Control point used for controlling mass flow. (Integer ≥ 0 or Blank). Blank or zero is only allowed when Mdot > 0.0. See Remark 3.
TA1, TA2	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2; Default for TA2 is TA1.)
Mdot	A multiplier for mass flow rate in case there is no point associated with the CNTRLND field. (Real > 0.0; Default = 1.0 if CNTMDOT > 0). See Remark 3.)

Remarks:

1. CONVM is used with an CHBDYP entry of type FTUBE having the same EID.
2. The temperature of the fluid film point may be specified to determine the material properties for the fluid. If FLMND=0, the reference temperature has several options. It can be the average of surface and ambient temperatures, the surface temperatures, or the ambient temperature, as defined in the FORM field of the PCONVM Bulk Data entry.
3. The CNTMDOT has a dual function. It can reference the ID of the CONTRLT Bulk Data entry to activate a thermostat controlled flux transfer. It can also reference a GRID or a scalar point which is set to the desired mass flow rate (mdot) to effect the advection of energy downstream at an  $mdot \cdot C_p \cdot T$  rate. If CNTMDOT is zero or is not specified (blank field), then the mass flow rate must be specified on the Mdot field (8th field).

Case 1: Define a thermostat controller using the CONTRLT Bulk Data entry.

Total mass flow rate = CONTRLT · Mdot · CNTMDOT

where CONTRLT is the grid ID on the 2nd field of the CONTRLT Bulk Data entry that ties the mass flow rate through this sensor grid.

Mdot is the 8th field of the CONVM Bulk Data entry.

CNTMDOT is the grid ID that has a mass flow rate associated with it.

Example: The thermostat controller controls GRID 5 (sensor grid) from 250 degrees to 300 degrees. If the temperature exceeds 300 degrees, then the mass flow is turned ON. If the temperature is below 250 degrees, then the mass flow is turned OFF.

```
CONTRLT, 5, 2, T, TSTAT, 250.0, 300.0, 1, 1.0 +
+, 0.0, 0.0, 0.0
CONVM, 100001, 1, , 5, 4
```

Note that GRID 5 is also specified on the 5th field of the CONVM Bulk Data entry.

Case 2: There is no CONTRLT Bulk Data entry (more general case).

Total mass flow rate = Mdot · CNTMDOT

If the CNTMDOT (the 5th field of the CONVM Bulk Data entry) is specified, then the default Mdot value is 1.0 and the mass flow rate is defined through CNTMDOT.

Example: Define a constant mass flow rate of 0.5 through CNTMDOT.

```
CONVM, 100001, 1, , 5, 4
SPC, 1, 5, 1, 0.5
```

To specify a time-varying mass flow rate, replace the SPC Bulk Data entry by TEMPBC,TRAN for SOL 159 or by SPC1 and SPCD for SOL 400.

For SOL 159:

```
TEMPBC, 2, TRAN, 1.0, 5
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , , 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

For SOL 400:

```
SPC = 111 (Case Control command)
SPC1, 111, , 5
SPCD, 2, 5, , 1.0
DLOAD, 700, 1.0, 1.0, 9
TLOAD1, 9, 2, , 1, 121
TABLED1, 121
, 0.0, 0.3, 100.0, 0.5, endt
```

If the CNTMDOT is not specified (zero or blank), then the mass flow rate must be constant and must be defined on Mdot (the 8th field of the CONVM Bulk Data entry).

Example: Define a mass flow rate of 0.5 directly on Mdot without specifying CNTMDOT.

CONVM,100001,1,,,,99,,0.5

4. If only the first ambient point is specified then, the second ambient point is assumed to have the same temperature.
5. See the Bulk Data entry, [PCONVM](#), for an explanation of the mathematical relationships available for forced convection and the reference temperature for fluid material properties.
6. RC network solver does not support CONVM for thermal analysis, it uses PRJCON for forced convection.

## CORD1C

## Cylindrical Coordinate System Definition, Form 1

Defines a cylindrical coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1C	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1C	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Descriptor	Meaning
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A ; G1B ≠ G2B ≠ G3B )

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in [Figure 9-40](#)) in this coordinate system is given by (R, θ, Z) where θ is measured in degrees.



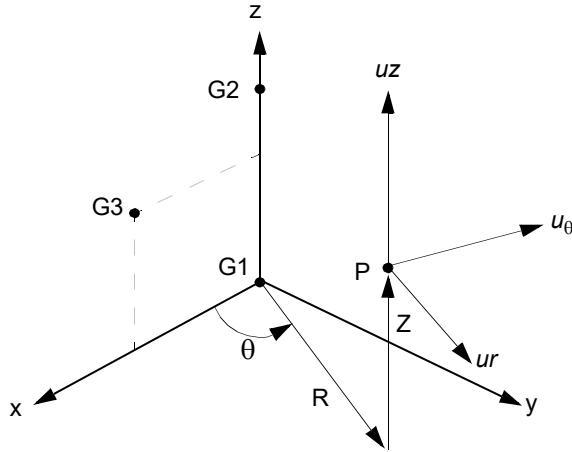


Figure 9-40 CORD1C Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_z)$ .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide*.

If CYLINDRICAL and point exactly on z-axis ( $r = 0.0$  exactly) then R,  $\theta$ , Z triad coordinates are exactly aligned with the local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

## CORD1R Rectangular Coordinate System Definition, Form 1

Defines a rectangular coordinate system using three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1R	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

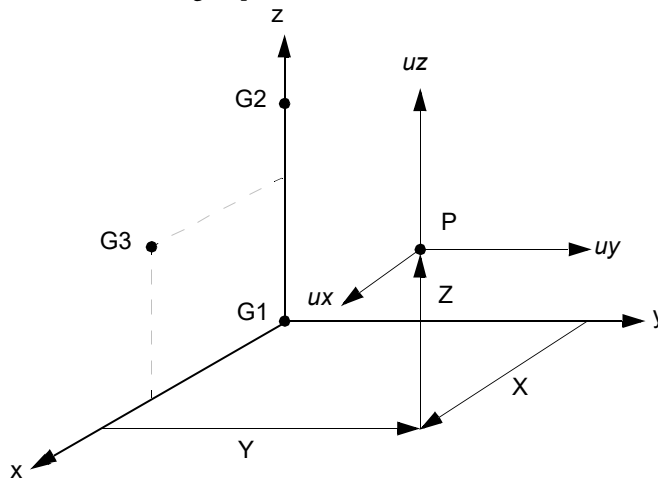
Example:

CORD1R	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Describer	Meaning
CIDA, CIDB	Coordinate system identification number. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with definitions that do not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the x-z plane. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in [Figure 9-41](#)) in this coordinate system is given by (X, Y, Z).



## Figure 9-41 CORD1R Definition

5. The displacement coordinate directions at P are shown above by  $(u_x, u_y, u_z)$ .

## CORD1S

## Spherical Coordinate System Definition, Form 1

Defines a spherical coordinate system by reference to three grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD1S	CIDA	G1A	G2A	G3A	CIDB	G1B	G2B	G3B	

Example:

CORD1S	3	16	32	19					
--------	---	----	----	----	--	--	--	--	--

Describer	Meaning
CIDA, CIDB	Coordinate system identification numbers. (Integer > 0)
GiA, GiB	Grid point identification numbers. (Integer > 0; G1A ≠ G2A ≠ G3A and G1B ≠ G2B ≠ G3B)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
2. One or two coordinate systems may be defined on a single entry.
3. GiA and GiB must be defined in coordinate systems with a definition that does not involve the coordinate system being defined. The first point is the origin, the second lies on the z-axis, and the third lies in the plane of the azimuthal origin. The three grid points GiA (or GiB) must be noncolinear and not coincident.
4. The location of a grid point (P in [Figure 9-42](#)) in this coordinate system is given by (R, θ, φ) where θ and φ are measured in degrees.

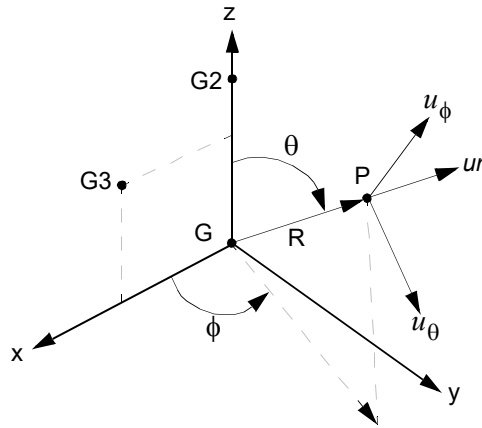


Figure 9-42 CORD1S Definition

5. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_\phi)$ .
6. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide*.

If SPHERICAL and  $R = 0.0$  (point at origin) then  $R, \theta, \phi$  triad coordinates are exactly aligned with the local defining system triad. User caution: if  $R \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

If SPHERICAL and r in azimuth plane is  $r = 0.0$  and  $z \neq 0.0$  then the coordinate system is defined as  $R$  global along  $\pm Z$ - local defining system triad according if  $z > 0.$  or  $z < 0.$ ,  $\theta$  -global along X-local defining system triad and  $\phi$  -global along Y-local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

## CORD2C

## Cylindrical Coordinate System Definition, Form 2

Defines a cylindrical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2C	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2C	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Describer	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

- Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S and CORD3G entries must be unique.
- The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
- The continuation entry is required.
- If RID is zero or blank, the basic coordinate system is used.
- The location of a grid point (P in [Figure 9-43](#)) in this coordinate system is given by (R, θ, Z), where θ is measured in degrees.

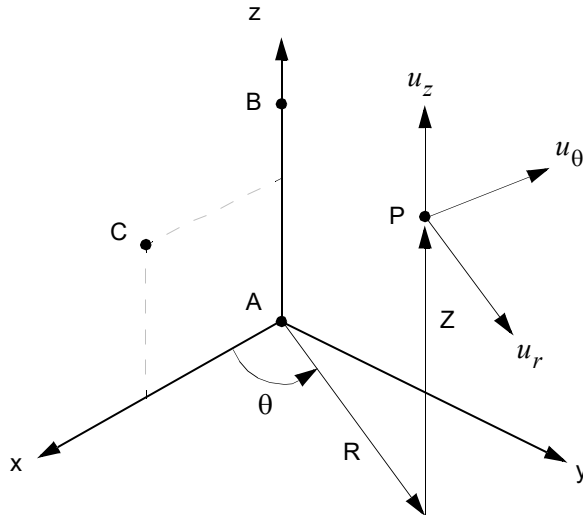


Figure 9-43 CORD2C Definition

6. The displacement coordinate directions at P are dependent on the location of P as shown above by  $(u_r, u_\theta, u_z)$ .
7. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of cylindrical coordinate systems in [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide*. If CYLINDRICAL and point exactly on z-axis ( $r = 0.0$  exactly) then R,  $\theta$ , Z triad coordinates are exactly aligned with the local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.
9. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#).

## CORD2R

## Rectangular Coordinate System Definition, Form 2

Defines a rectangular coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2R	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2R	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Describer	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer ≥ 0; Default = 0; which is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second defines the direction of the z-axis. The third point defines a vector which, with the z-axis, defines the x-z plane. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#).
6. The location of a grid point (P in the [Figure 9-44](#)) in this coordinate system is given by (X, Y, Z).



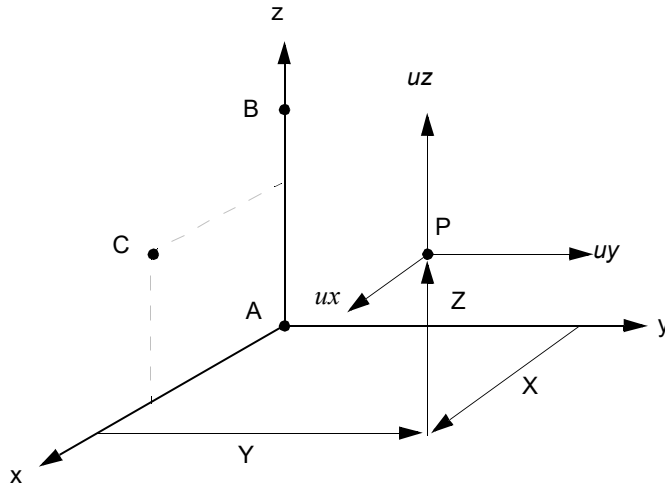


Figure 9-44 CORD2R Definition

7. The displacement coordinate directions at P are shown by  $(u_x, u_y, u_z)$ .
8. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

## CORD2S

## Spherical Coordinate System Definition, Form 2

Defines a spherical coordinate system using the coordinates of three points.

Format:

1	2	3	4	5	6	7	8	9	10
CORD2S	CID	RID	A1	A2	A3	B1	B2	B3	
	C1	C2	C3						

Example:

CORD2S	3	17	-2.9	1.0	0.0	3.6	0.0	1.0	
	5.2	1.0	-2.9						

Describer	Meaning
CID	Coordinate system identification number. (Integer > 0)
RID	Identification number of a coordinate system that is defined independently from this coordinate system. (Integer $\geq$ 0; Default = 0 is the basic coordinate system.)
Ai, Bi, Ci	Coordinates of three points given with respect to the coordinate system defined by RID. (Real)

Remarks:

1. Coordinate system identification numbers on all CORD1R, CORD1C, CORD1S, CORD2R, CORD2C, CORD2S, and CORD3G entries must all be unique.
2. The three points [(A1, A2, A3), (B1, B2, B3), (C1, C2, C3)] must be unique and noncolinear. Noncolinearity is checked by the geometry processor. The first point defines the origin. The second point defines the direction of the z-axis. The third lies in the plane of the azimuthal origin. The reference coordinate system must be independently defined.
3. The continuation entry is required.
4. If RID is zero or blank, the basic coordinate system is used.
5. An example of defining a coordinate system with respect to another coordinate system (RID) may be found in the Linear Static Analysis manual [Example](#).
6. The location of a grid point (P in [Figure 9-45](#)) in this coordinate system is given by (R,  $\theta$ ,  $\phi$ ), where  $\theta$  and  $\phi$  are measured in degrees.

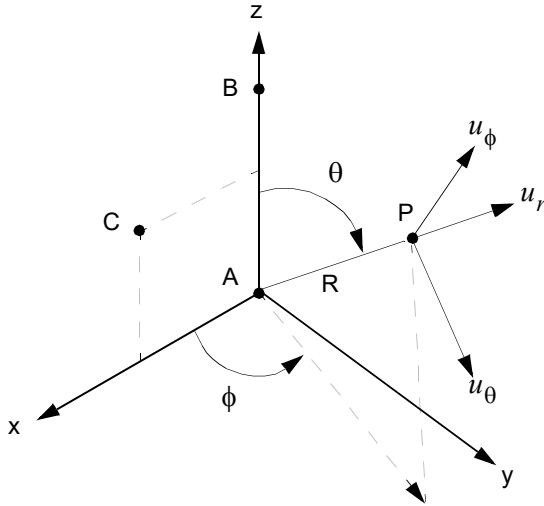


Figure 9-45 CORD2S Definition

7. The displacement coordinate directions at P are shown above by  $(u_r, u_\theta, u_\phi)$ .
8. It is recommended that points on the z-axis not have their displacement directions defined in this coordinate system. See the discussion of spherical coordinate systems in [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide*.

If SPHERICAL and  $R = 0.0$  (point at origin) then  $R, \theta, \phi$  triad coordinates are exactly aligned with the local defining system triad. User caution: if  $R \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

If SPHERICAL and  $r$  in azimuth plane is  $r = 0.0$  and  $z \neq 0.0$  then the coordinate system is defined as  $R$  global along  $\pm Z$ -local defining system triad according if  $z > 0.$  or  $z < 0.$ ,  $\theta$ -global along  $X$ -local defining system triad and  $\phi$ -global along  $Y$ -local defining system triad. User caution: if  $r \approx 0.0$  but not exactly 0.0 should be avoided as unexpected direction cosine matrix may result.

9. If any CORD2C, CORD2R, or CORD2S entry is changed or added on restart, then a complete re-analysis is performed. Therefore, CORD2C, CORD2R, or CORD2S changes or additions are not recommended on restart.

## CORD3G

## General Coordinate System

Defines a general coordinate system using three rotational angles as functions of coordinate values in the reference coordinate system. The CORD3G entry is used with the MAT9 entry to orient material principal axes for 3-D composite analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3G	CID	METHOD	FORM	THETAID1	THETAID2	THETAID3	CIDREF		

Example:

CORD3G	100	E313	EQN	110	111	112	0		
--------	-----	------	-----	-----	-----	-----	---	--	--

Descriptor	Meaning
CID	Coordinate system identification number. See Remark 1. (Integer > 0)
METHOD	E313 or S321 for Euler angle rotation in 313 sequence or space-fixed rotation in 321 sequence. See Remark 2. (Character; Default = "E313")
FORM	Specifies the Bulk Data entry which defines angles. FORM = "EQN" for DEQATN entry or FORM = "TABLE" for TABLE3D entry. (Character; Default = "EQN")
THETAID	Identification number for DEQATN or TABLE3D Bulk Data entry which defines the three angles (in radians) measured from reference coordinates to the general material coordinate system. See Remark 3. (Integer > 0)
CIDREF	Identification number for the coordinate system from which the orientation of the general coordinate system is defined. (Integer ≥ 0; Default = 0)

Remarks:

1. CID must be unique with respect to all other coordinate systems. CID cannot be referenced on GRID entries.
2. Three Euler angles specify the rotation of the CORD3G coordinate axes (xyz) with respect to the local Cartesian coordinate axes (XYZ) in CIDREF as follows: first rotate about Z-axis by  $\theta_1$ , next rotate about rotated x-axis by  $\theta_2$ , and then rotate about rotated z-axis by  $\theta_3$ . On the other hand, the space-fixed rotations in 321 sequence specify all the rotations about the fixed coordinate axes: first rotate about Z by  $\theta_1$ , next about Y by  $\theta_2$ , then about X by  $\theta_3$ .
3. The three rotations define a coordinate transformation which transforms position vectors in the reference coordinate system into the general coordinate system.
4. The DEQATN option must have three arguments representing the three axes of CIDREF, although not all arguments are necessarily needed in the equation.

5. If FORM = EQN is used with CIDREF pointing to any CORD1k or CORD2k,  $k = C, S$ , and a spatial relationship is required, then the equations are written using  $R, \theta, \varphi$  as input variables. Non spatial fixed values such as THETA1 (A,B,C)=PI(1.) may of course be entered for any type of coordinate defined by the CIDREF.

If FORM = TABLE3D is used with CIDREF pointing to any CORD1k or CORD2k,  $k = C, S$ , and a spatial relationship is required then the equations are written using the variables X,Y, Z defined at the origin of the CIDREF as input values and not the variables  $R, \theta, Z$  or  $R, \theta, \varphi$  input values. For example, if a CORD1S is used then the table input value is X internally computed as  $R \cdot \sin \theta \cdot \cos \varphi$ .

Whether the EQN or TABLE3D FORM is used, the equation must return an angle measure in radians to have any meaning.

## CORD3R

## Moving Rectangular Coordinate System

Defines a moving rectangular coordinate system using three points in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CORD3R	CID	N1	N2	N3	CID	N1	N2	N3	

Example:

CORD3R	1001	1	144	300					
--------	------	---	-----	-----	--	--	--	--	--

Describer	Meaning
CID	Unique coordinate system number. (Integer > 0)
N1, N2, N3	Grid point numbers (must be unique). (Integer > 0)

Remarks:

1. Available in SOL 700 only.
2. Two different coordinate systems may be defined on one entry.
3. The grid points must be defined in an independent coordinate system.
4. The first grid point is the origin, the second lies on the z-axis and the third lies in the x-z plane.
5. The position and orientation of the coordinate system is updated as the grid points move.
6. The three grid points must not be colinear.

## COSMGRP

## Define the coupled area in Co-Simulation service

COSMGRP defines the coupled area where Nastran will exchange input and output data with co-simulation partner in Co-simulation analysis. It is supported in SOL 400 only.

### Format 1: TYPE=SURFACE

1	2	3	4	5	6	7	8	9	10
COSMGRP	GRPID	TYPE							
	EID	FaceID	EID	FaceID	EID	FaceID	EID	FaceID	
	EID	FaceID	...						

### Examples 1: TYPE=SURFACE

1	2	3	4	5	6	7	8	9	10
COSMGRP	202	SURFACE							
	1001	5	1002	4					

### Format 2: TYPE=POINT

1	2	3	4	5	6	7	8	9	10
COSMGRP	GRPID	TYPE							
	GridID1	GridID2	...						
	GridIDi		...						

### Examples 2: TYPE=POINT

1	2	3	4	5	6	7	8	9	10
COSMGRP	202	POINT							
	2001	2002	31002	31003					

Describer	Meaning
GRPID	Identification number of COSMGRP referenced by a COSMSRV Bulk Data Entry. (Integer > 0)
TYPE	The definition type of coupled area (Character; Default is SURFACE) SURFACE: the couple area is defined by the element ID and its face ID POINT: the coupled area is defined by a list of grid IDs
EID	Element identification number
FACEID	Face identification number of element. Refer to the remarks 6. and 7. for the element face identification of CHBDYE entry.
GRIDID	Grid point identification numbers

**COSMINP**

## Define the physical quantities of input in Co-Simulation service

Defines the physical quantities of input which Nastran will receive from the co-simulation partner in Co-simulation analysis. It is supported in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
COSMINP	PHYIN								
	PHYQUA1	PHYQUA2							

Example:

1	2	3	4	5	6	7	8	9	10
COSMINP	405								
	FORCE								

Describer	Meaning
PHYIN	Identification number of COSMINP referenced by a COSMSRV Bulk Data Entry (Integer > 0).
PHYQUAi	Name of physical quantities which will be provided to Nastran as input in co-simulation analysis (Character; Default: FORCE)

Remarks:

1. Currently, it supports FORCE and TEMP (temperature load) from CoSIM only.



## COSMOUT

### Define the physical quantities of output in Co-Simulation service

Defines the physical quantities of output which Nastran will yield and provide to the co-simulation partner in Co-simulation analysis. It is supported only in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
COSMOUT	PHYOUT								
	PHYQUA1	PHYQUA2	PHYQUA3						

Example:

1	2	3	4	5	6	7	8	9	10
COSMOUT	506								
	DISP	ACCE							

Describer	Meaning
PHYOUT	Identification number of COSMOUT referenced by a COSMSRV Bulk Data Entry (Integer > 0).
PHYQUAi	Name of physical quantities which will be provided to partner software as input in co-simulation analysis (Character)

One or any combination of the following physical quantities can be used:

- DISP
- VELO
- ACCE

Remarks:

1. Currently, it supports only one or combination of DISP, VELO, and ACCE.
2. The displacement, velocity and acceleration from Nastran to co-simulation are always in basic coordinate system, even these degree of freedom are defined in local rectangular coordinate system like CORD2R. Note that only rectangular local coordinate system is supported; the cylindrical and spherical local coordinate systems are not supported in co-simulation analysis.

## COSMSEL

## Specify the Co-Simulation service

Specifies the selected co-simulation service for analysis through the COSMSRV. It is supported only in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
COSMSEL	SID								
	CSRVID1	CSRVID2	CSRVID3						

Example:

1	2	3	4	5	6	7	8	9	10
COSMSEL	1								
	21								

Describer	Meaning
-----------	---------

SID Identification number referenced by a COSMSEL Case Control command (Integer > 0).

CSRVIDi Identification number of COSMSRV entry (Integer > 0).

Remarks:

1. Currently, it supports only one COSMSRV.

## COSMSRV

## Define the Co-Simulation service

Defines the co-simulation service for analysis on a coupled group and exchanged physical quantities. It is supported only in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
COSMSRV	CSRVID	SERV							
	GRPID1	PHYINP	GRPID2	PHYOUT					

Example:

1	2	3	4	5	6	7	8	9	10
COSMSRV	21	scFlow							
	202	305	202	406					

Describer	Meaning
CSRVID	Identification number referenced by a COSMSEL Bulk Data Entry (Integer > 0).
SERV	Cosim service identifier (Character) (support scFLOW only now).
GRPID1	Identification number of COSMGRP Bulk Data Entry (Integer > 0). It defines the coupled area in which Nastran receives the input from co-simulation partner.
GRPID2	Identification number of COSMGRP Bulk Data Entry (Integer > 0). It defines the coupled area in which Nastran provides output of analysis to co-simulation partner.
PHYINP	Identification number of COSMINP Bulk Data Entry (Integer > 0). It defines the physical quantities which Nastran receives from co-simulation partner.
PHYOUT	Identification number of COSMOUT Bulk Data Entry (Integer > 0). It defines the physical quantities which Nastran provides to co-simulation partner.

Remarks:

1. Currently, it supports one set of coupled area only.
2. At present, GRPID1 should be same as GRPID2.
3. SERV is a mandatory input. Currently, any characters are acceptable; scFLOW is recommended.

## COUCOHF

## Cohesive Friction Model To Be Used For a COUPLE Entry

Defines a cohesive friction model suited for Euler Coupled analyses. The friction model is defined as part of the coupling surface. Use SOL700 only.

Format:

COUCOHF	CID	COHFRID	SUBID	COHFRICID					
---------	-----	---------	-------	-----------	--	--	--	--	--

Example:

COUCOHF	112	14	12	12					
---------	-----	----	----	----	--	--	--	--	--

Describer	Meaning
CID	Unique number of a COUCOHF entry. (Integer $\geq 0$ ; required)
COHFRID	Number of a set of COUCOHF entries. COUCOHF must be referenced from a COUPLE entry. (Integer $\geq 0$ ; required)
SUBID	BSURF, BCSEG or BCPROP id, which must be a part of the BSID field of the COUPLE entry. See Remark 2. (Integer $\geq 0$ ; default = blank)
COHFRICID	Cohesive friction ID (Integer $\geq 0$ ; required)

Remarks:

1. One couple entry can reference more than one COUCOHF entry. This allows a cohesive friction that varies along the coupling surface.
2. When SUBID is left blank, then the cohesive friction applies to the whole coupling surface.
3. A coupling surface segment can only have one porosity or one cohesive friction definition assigned.

## COUOPT

## Coupling Surface Pressure Definition

Defines the interaction factor and a pressure load from the covered side acting on a BSURE, BCPROP, BCMATL, BCSEG, BCBOX. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUOPT	CID	OPTID	SUBID	FACTOR	FACTORV				
	PLCOVER	PLCOVERV							

Example:

COUOPT	1	80	42	CONSTANT					
	CONSTANT	1.E5							

Describer	Meaning
CID	Unique number of a COUOPT entry. (Integer > 0; Required)
OPTID	Number of a set of COUOPT entries. OPTID must be referenced from a COUPLE entry. (Integer > 0; Required)
SUBID	<p>&gt; 0            Number of a BSURE, BCBOX, BCPROP, BCMATL or BCSEG, which must be part of the surface as defined in the COUPLE entry. (Integer <math>\geq</math> 0, 0)</p> <p>= 0            COUOPT definitions used for the entire surface as defined in the COUPLE entry.</p>
FACTOR	<p>Method of defining the interaction FACTORV with which the Eulerian pressure acting on the surface is multiplied. (Character, CONSTANT)</p> <p>CONSTANT    The FACTOR is constant and specified in FACTORV</p>
FACTORV	The interaction factor. (Real, 1.)
PLCOVER	<p>Method of defining the pressure load exerted on the faces of the surface from the covered side. The pressure load is applied only when the Eulerian pressure is greater than zero. (Character, CONSTANT)</p> <p>CONSTANT    The PLCOVER is constant and specified in PLCOVERV.</p> <p>TABLE        The PLCOVER varies with time. PLCOVERV is the number of a TABLED1 entry giving the variation of the PLCOVER (y-value) with time (x-value).</p>
PLCOVERV	The pressure load or the number of a TABLED1 entry depending on the PLCOVER entry. (Real $\geq$ 0, 0.)

Remarks:

1. The effect of specifying an interaction FACTOR is similar to specifying a porosity coefficient on a COUPOR entry. The difference is that in this case the surface still acts as a wall boundary for the Eulerian material.
2. Applying a PLCOVER instead of applying a pressure load on the faces through either a PLOAD, PLOAD4, or DAREA entry gives the following differences:
  - a. PLCOVER is applied only when there is a balancing Eulerian pressure greater than zero.
  - b. Possible porosity as defined on a COUPOR entry is taken into account when applying the PLCOVER.
  - c. With PARAM,PLCOVCUT you can define a cut-off time that is applied to PLCOVER.
3. The covered side of a surface lies on the side where there is no Eulerian material.

## COUP1FL

## Coupling Surface Failure

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUP1FL	CFID	RHO	SIE	XVEL	YVEL	ZVEL	PRESSURE	MATERIAL	

Example:

COUP1FL	3	1.225	204082.	900.					
---------	---	-------	---------	------	--	--	--	--	--

Describer	Meaning
CFID	Unique ID of a COUP1FL entry referenced from the COUPLE entry. (Integer > 0; Required)
RHO	Surrounding density. See Remark 2. (Real > 0)
SIE	Surrounding specific internal energy. See Remark 2. (Real)
XVEL	Surrounding x-velocity. See Remark 2. (Real)
YVEL	Surrounding y-velocity. See Remark 2. (Real)
ZVEL	Surrounding z-velocity. See Remark 2. (Real)
PRESSURE	Surrounding pressure. See Remark 4. (Real)
MATERIAL	MATDEUL ID. Only used when the multi-material Euler solver is active. (Blank)

Remarks:

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. For restrictions on the use of COUP1FL refer to param,flow-method. The coupling surface failure is associated with the element failure of the structure to which the surface is connected. Therefore, you have to define a failure model for the structure for the entry to take effect in the analysis.
2. For the first order Euler solvers no restriction apply to the values of the surrounding variables. For the Roe solver at least one of the surrounding variables should be defined. The default value of the density (RHO) will be set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
3. The coupling surface must only consist of CQUAD and/or CTRIA elements.
4. The field PRESSURE has to be left blank in combination with the Roe solver.
5. In combination with multi-material Euler only outflow of material is allowed. Each material in an outflow Euler element is transported. The materials are transported in proportion to their relative volume fractions.

## COUPINT

## Coupling Surface Interaction

Defines the surrounding variables when a segment of a coupling surface fails. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUPINT	CID	CID1	CID2						

Example:

COUPINT	33	2							
---------	----	---	--	--	--	--	--	--	--

Describer	Meaning
CIID	Unique number of a COUPINT entry.
CID1	Number of COUPLE entry 1.
CID2	Number of COUPLE entry 2.

Remarks:

1. This entry can only be used in combination with DYPARAM,FASTCOUP, ,FAIL and with either the HYDRO, MMHYDRO or MMSTREN Euler Solver. The interaction will be activated when failure of a Lagrangian structure with which the coupling surface is associated occurs. Therefore, you have to define a failure model for the material of the structure.
2. The coupling surface must consist of CQUAD and/or CTRIA elements.



## COUPLE

## General Euler-Lagrange Coupling Surface

Defines a coupling surface that acts as the interface between an Eulerian (finite volume) and a Lagrangian (finite element) domain. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
COUPLE	CID	BSID	COVER	REVERSE	CHECK	PORID	OPTID	CTYPE	
	INFID	HTRID	FS	FK	EXP	INTID			
	SET1ID	MESHID	TDEAC	COUP1FL	HYDSTAT	SKFRIC	COHFRI D		

Example:

COUPLE	100	37	INSIDE	ON	ON				
				0.3	0.0				

Describer	Meaning
CID	Unique number of a COUPLE entry. (Integer > 0; Required)
BSID	Number of a BSURE, BCBOX, BCPROP, BCMATL or BCSEG entry defining the coupling surface. (Integer > 0; Required)
COVER	The processing strategy for Eulerian elements inside and outside of the coupling surface. (Character, INSIDE)
	INSIDE      The Eulerian elements inside the closed volume of the coupling surface are not processed.
	OUTSIDE     The Eulerian elements outside the closed volume of the coupling surface are not processed
	NONE        The Eulerian elements on both sides of the coupling surface are processed.
REVERSE	Auto-reverse switch for the coupling surface segments. (Character, ON)
	ON            If required, the normals of the coupling surface segments are automatically reversed so that they all point in the same general direction as to give a positive closed volume.
	OFF          The segments are not automatically reversed. The user is responsible for defining the correct general direction of the segment normals.
CHECK	Check switch for coupling surface segments. (Character, ON)
	ON            The normals of the segments are checked to verify that they all point in the same general direction and yield a positive closed volume.

Describer	Meaning
	<p>OFF The segments are not checked. It the responsibility of the user to ensure that the direction of the segment normals yield a positive closed volume.</p> <p>When “REVERSE” is set to “ON”, the “CHECK” option will be automatically activated.</p>
PORID	Number of a set of “LEAKAGE” entries that define the porosity for the BSURF entries. (Integer > 0, 0 (no porosity))
OPTID	Not available for the Roe solver Number of a set of “COUOPT” entries that define special options for the BSURF entries (Integer ≥ 0, 0 (no special options))
CTYPE	Not available for the Roe solver Coupling surface type definition. (Character, STANDARD)
	<p>STANDARD Standard Euler-Lagrange interaction.</p> <p>AIRBAG Coupling for airbag applications. It is equivalent to the standard coupling algorithm with the following exceptions that tailor the solution for airbag applications: Inflow through a porous (sub)-surface will only occur when there is already some material (gas) in the Eulerian element. Almost empty Eulerian elements will be automatically eliminated. The standard algorithm redistributes the small mass to the most suitable neighbor elements.</p>
INFID	Number of a set of “ABINFL” entries that define the inflator(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer > 0, 0 (no inflators))
HTRID	Number of a set of “HEATLOS” entries that define the heat transfer definition(s) on the subsurface(s) of the coupling surface. Not available for the Roe solver. (Integer ≥ 0, 0 (no heat transfer))
FS	Static friction coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0, 0.0)
FK	Kinetic friction coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0, 0.0)
EXP	Exponential decay coefficient. See Remark 6. Not available for the Roe solver. (Real ≥ 0.0, 0.0)
INTID	ID of an INITGAS entry specifying initial gas composition for the Euler mesh. See Remark 7. (Integer ≥ 0, 0 (no initial gas composition))
SET1ID	The number of a SET1 entry, which defines the Eulerian elements associated with this coupling surface. See Remark 8. (Integer ≥ 0; Default = blank)
MESHID	ID of a MESH entry, which defines the Eulerian mesh associated with this coupling surface. See Remark 8. (Integer ≥ 0; Default = blank)

Describer	Meaning
TDEAC	Time of deactivation of the coupling surface and the associated Eulerian mesh. (Real $\geq$ 0.0, 1.E20)
COUP1FL	The number of a COUP1FL entry, which defines the surrounding variables for the coupling surface when its segments fail. See Remark 9. (Integer $\geq$ 0; Default = blank)
HYDSTAT	The number of a HYDSTAT entry, which specifies a hydrostatic preset. The preset is applied to all Euler element specified by the SET1ID and MESHID. See Remark 11. (Integer $\geq$ 0, 0 (no hydrostatic preset))
SKFRIC	Skin friction value. See Remark 13. (Real $\geq$ 0.0, 0.0)
COHFRID	Number of a set of COUCOHF entries that define the cohesive friction on the subsurface(s) of the coupling surface. ( $I \geq 0$ Default = blank)

Remarks:

1. All coupling surfaces must form a multi-faceted closed volume. If necessary, additional segments must be defined to achieve the closed volume. The closed volume must intersect at least one Euler element initially.
2. All segments must be attached to the face of an element. Dummy elements can be used to define any additional segments that are required to create the closed volume.
3. The normals of all segments that form the coupling surface must point in the same general direction and result in a positive closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of the initial definition of the segments.
4. The “COVER” field determines how Eulerian elements that are inside and outside of the coupling surface are processed. The default setting of INSIDE is appropriate for most of the problems. In the majority of analyses, the Eulerian material flows around the outside of the coupling surface. Therefore, the Eulerian elements that fall within the coupling surface do not contain material. For some specific applications, such as airbag inflation, the Eulerian material (gas) is completely contained within the coupling surface. In these cases, the “COVER” definition should be set to OUTSIDE.
5. By default the fast coupling algorithm is used. The algorithm then used is substantially faster than the general coupling. The restriction is that you cannot use an arbitrarily shaped Euler mesh with the fast coupling algorithm. All element faces of the Euler mesh must have their normal pointing in any of the three basic coordinate directions. If you want to use the general coupling algorithm, you can define the parameter “DYPARAM,FASTCOUP,NO” in the input file.
6. The friction model implemented for the coupling algorithm is a simple Coulomb friction definition. The friction coefficient  $\mu$  is defined as:

$$\mu = \mu_k + (\mu_s - \mu_k) \cdot e^{-(\beta \cdot v)}$$

where  $\mu_s$  is the static friction coefficient,  $\mu_k$  is the kinetic friction coefficient,  $\beta$  the exponential decay coefficient and  $v$  the relative sliding at the point of contact.

7. An initial gas composition is for use with the single-material hydrodynamic Euler solver and an ideal-gas equation of state (EOSGAM) only.
8. Multiple coupling surfaces are available when you associate one Eulerian domain with a single coupling surface by either using the SETIID or the MESHID option. Note that only one of the two options may be set and will work only in combination with the fast coupling algorithm.
9. The COUP1FL option is available and valid only in combination with the fast coupling algorithm with the failure option (DYPARAM,FASTCOUP, ,FAIL). If no number is given, the default values of the surrounding variables will be used; the density (RHO) is set equal to the reference density as defined on the MATDEUL entry. By default, the other variables (SIE, XVEL, YVEL and ZVEL) are set equal to zero.
10. If an ACTIVE entry is present, its definition is ignored in case the TDEAC value is defined in combination with the fast coupling algorithm.
11. If there is only one coupling surface and no adaptive meshing is used, the HYDSTAT field can be left blank. To impose a boundary condition that matches the hydrostatic initialization, the PORHYDS entry can be used.
12. A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.
13. The skin friction is defined as:

$$C_f = \frac{\tau_w}{0.5 \cdot \rho u^2}$$

Here,  $\tau_w$  denotes the shear friction in an Euler element adjacent to a couple surface segment where  $\rho$  is the density and  $u$  is the tangential relative velocity in the Euler element that is adjacent to a couple surface segment.

SKFRIC will only be used when VISC has been set on either an EOSGAM or an EOSPOL entry. If VISC has been set and if SKFRIC has not been set then a no slip condition will be prescribed at the interface between fluid and structure.

14. If the coupling surface is a structural solid then Sol700 can automatically create a coupling surface for the solid. This coupling surface consists of the boundary faces of the solid. To activate this, a BCPROP has to be created for the solid and the BCPROP has to be used for the BSID of the COUPLE entry.
15. Option NONE requires the use of DYPARAM, AUTCOUP. For details refer to this DYPARAM.
16. When BSID refers to a ID of a BCPROP or BCMATL, only properties of shell elements can be referenced. All solid elements referenced with the same PID or MID will be ignored. Use BCSEG to reference faces of solid elements.

## CPENTA

### Five-Sided Solid Element Connection

Defines the connections of a five-sided solid element with six to fifteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPENTA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15								

Example:

CPENTA	112	2	3	15	14	4	103	115	
	5	16	8				120	125	
	130								

Describer	Meaning	Type	Default
EID	Element identification number.	$0 < \text{Integer} < 100,000,000$	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer $\geq 0$ or blank	Required

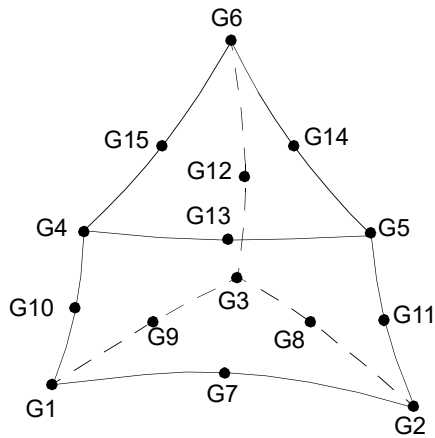


Figure 9-46 CPENTA Element Connection

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved; i.e., G1, G2, and G3 define a triangular face, G1, G10, and G4 are on the same edge, etc.
3. For Nastran conventional element, the edge grid points, G7 to G15, are optional. Any or all of them may be deleted. In the example shown, G10, G11, and G12 have been deleted. The continuations are not required if all edge grid points are deleted.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system except for hyperelastic elements, which are output in the basic coordinate system.
5. For nonhyperelastic elements the element coordinate system for the CPENTA element is derived accordingly. The origin of the coordinate system is located at the midpoint of the straight line connecting the points G1 and G4. The Z axis points toward the triangle G4-G5-G6 and is oriented somewhere between the line joining the centroids of the triangular faces and a line perpendicular to the midplane. The midplane contains the midpoints of the straight lines between the triangular faces. The X and Y axes are perpendicular to the Z axis and point in a direction toward, but not necessarily intersecting, the edges G2 through G5 and G3 through G6, respectively.

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

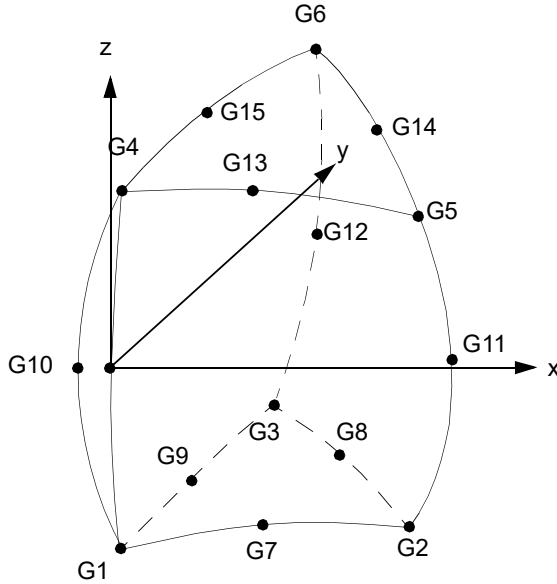


Figure 9-47 CPENTA Element Coordinate System

6. We recommend that the edge grid points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CPENTAFD element name in [Item Codes](#).
8. By default, all the nine edges of the element are considered straight unless, any of G7 through G15 are specified.
9. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-48](#).

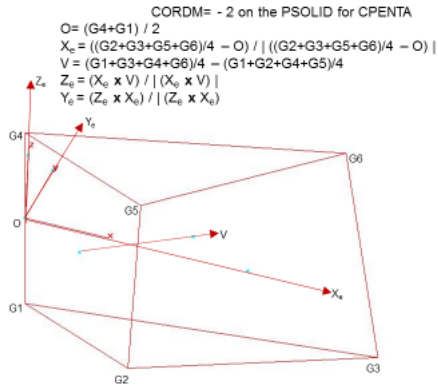


Figure 9-48 PSOLID for CPENTA



## CPYRAM

## Five-Sided Solid Element Connection

Defines the connections of the five-sided solid element with five or thirteen grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CPYRAM	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13		

Example:

CPYRAM	7	4	15	16	17	18	40	21	
	22	23	24	31	32	33	34		

Describer	Meaning	Type	Default
EID	Element identification number.	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

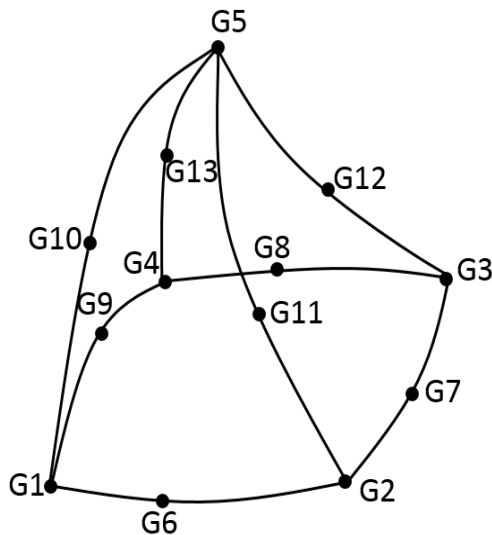


Figure 9-49

CPYRAM Element Connection

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, that is, G1, G2, G3, G4 define the quadrilateral base, G5 defines the apex, G1, G5, G10 are on the same edge, etc.
3. For a 5 noded CPYRAM element, only G1 - G5 should be provided. For a 13 noded CPYRAM element, all the corner and mid-side nodes (G1 - G13) should be provided. Note that partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. The pyramid elements use the basic coordinate system as the element coordinate system. They can use a material coordinate system defined by the CORDM field of the corresponding PSOLID entry. CORDM = 0, -1, -2 or blank is the same as the basic coordinate system.
5. For CORDM > 0, the material coordinate system is defined with respect to the initial geometry and is used to input orthotropic or anisotropic material properties and for stress output. For geometric nonlinear analysis, the material coordinate system rotates with the element. Component of stresses are output in the material coordinate system referenced by the CORDM field of PSOLID entry (CORDM > 0).

## CQUAD

## Fully Nonlinear Plane Strain Element

Defines a plane strain quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	THETA or MCID					

Example:

CQUAD	111	203	31	74	75	32			
-------	-----	-----	----	----	----	----	--	--	--

Describer	Meaning	
EID	Element identification number. (0 < Integer < 100,000,000)	
PID	Property identification number of a PLPLANE or PLCOMP entry. (Integer > 0)	
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)	
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer ≥ 0 or blank)	
G9	Identification number of center grid point. Optional. (Integer ≥ 0 or blank)	
THETA	Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=PSTRS or PLSTRN codes, THETA is measured relative to the line defined from G1-G2. For PSHLN2 or PLCOMP BEHi=COMPS code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0)	
MCID	Material coordinate system identification number MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:	
CORD1R,	x -axis of MCID the coordinate is projected onto shell surface and the	
CORD2R	material angle is measured from the G1-G2 line to the to the projected x-axis	

Describer	Meaning
CORD1C, CORD2C CORD1S, CORD2S	r-axis of MCID the coordinate is projected onto shell surface through the element center and the material angle is measured from the G1-G2 line to the to the projected r-axis
	For PSHLN2 (or PLCOMP) BEHi=COMPS code the MCID value on the element connection entry will be ignored. (Integer $\geq 0$ ; if blank, then THETA = 0.0 is assumed.)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G9 must be numbered as shown and must lie on a plane. G5 - G9 are Optional data for any or all four grid points for Nastran conventional element only.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
3. It is recommended that the edge points be located within the middle third of the edge.
4. The plot codes are specified under the CQUADFD element name in [Item Codes](#).
5. Stresses and strains are output in the coordinate system identified by the CID field of the PLPLANE entry.

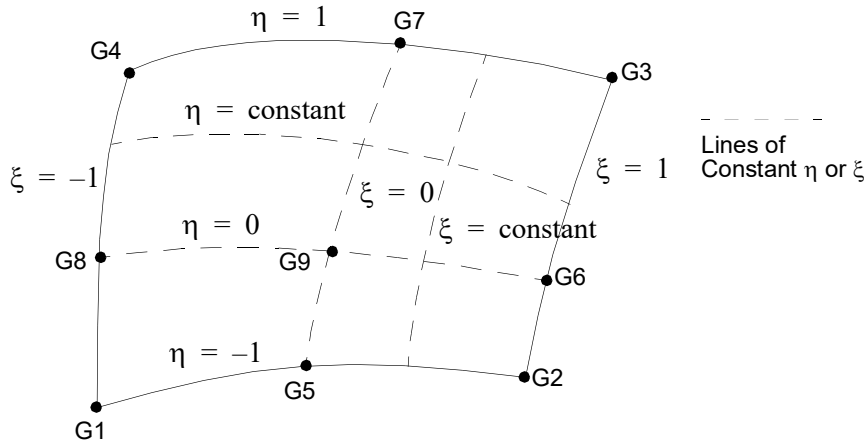


Figure 9-50 CQUAD Element Coordinate System

## CQUAD4 Quadrilateral Plate Element Connection

Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUAD4	111	203	31	74	75	32	2.6	0.3	
			1.77	2.04	2.09	1.80			

Describer	Meaning	
EID	Element identification number. (0 < Integer < 100,000,000)	
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0; Default = EID)	
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique.)	
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. See <a href="#">Figure 9-52</a> . See Remark 9. (Real; Default = 0.0)	
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:	
	CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the	
	CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis	
	CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the	
	CORD2C element center and the material angle is measured from the G1-G2 line	
	CORD1S, to the to the projected r-axis	
	CORD2S	
	Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 9. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)	

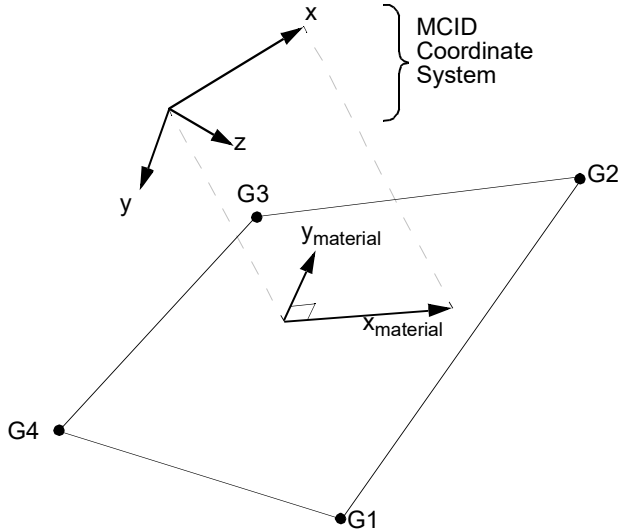


Figure 9-51 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. ZOFFS is ignored for hyperelastic elements. See Remark 6. (Real)
TFLAG	An integer flag, signifying the meaning of the $T_i$ values. (Integer 0, 1, or blank)
$T_i$	Membrane thickness of element at grid points G1 through G4. If “TFLAG” is zero or blank, then $T_i$ are actual user specified thicknesses. See Remark 4. for default. (Real $\geq 0.0$ or blank, not all zero.) If “TFLAG” is one, then the $T_i$ are fractions relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) $T_i$ are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than  $180^\circ$ .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.

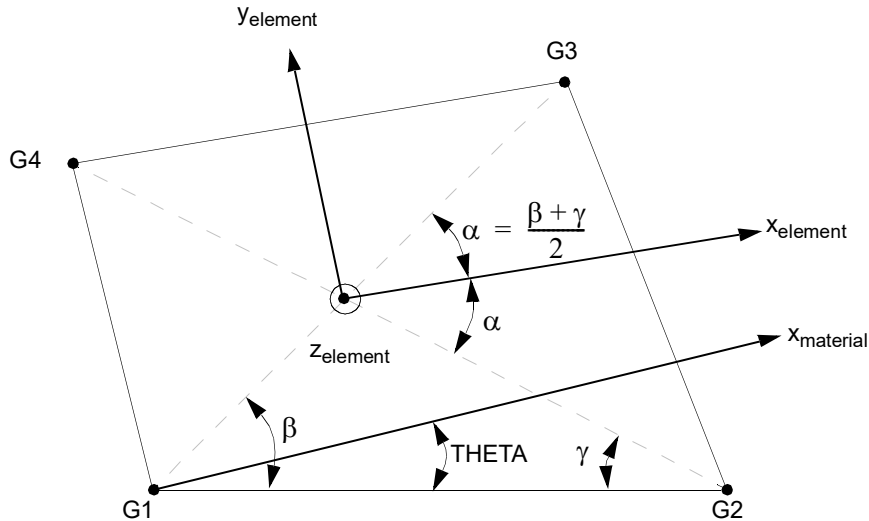


Figure 9-52 CQUAD4 Element Geometry and Coordinate Systems

5. The reference coordinate system for the output of stress, strain and element force depends on the element type.
  - For CQUAD4 elements which are not hyperelastic, the reference coordinate system for the output is the element coordinate system. See PARAM,OMID for output in the material system.
  - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
6. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

7. For finite deformation hyperelastic analysis, the plot codes are given by the CQUADFD element name in [Item Codes](#).
8. By default, all of the four edges of the element are considered straight.
9. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
10. For RC network solver in thermal analysis, the ZOFFS is ignored.
11. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.



**CQUAD8****Curved Quadrilateral Shell Element Connection**

Defines a curved quadrilateral shell or plane strain element with eight grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CQUAD8	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	T1	T2	T3	T4	THETA or MCID	ZOFFS	
	TFLAG								

Example:

CQUAD8	207	3	31	33	73	71	32	51	
	53	72	0.125	0.025	0.030	.025	30.	.03	

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE or PLCOMP entry. (Integer > 0)
G1, G2, G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0)
G5, G6, G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank). Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
Ti	Membrane thickness of element at grid points G1 through G4. If "TFLAG" zero or blank, then Ti are actual user specified thickness. See Remark 4. for default. (Real ≥ 0.0 or blank, not all zero.) If "TFLAG" one, then the Ti are fraction relative to the T value of the PSHELL. (Real > 0.0 or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.
THETA	Material property orientation angle in degrees. See Figure 9-53. THETA is ignored for hyperelastic elements. See Remark 10. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:  CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis

Describer	Meaning
	<p>CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line CORD1S, to the to the projected r-axis CORD2S</p> <p>(see Remark 3.) MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. See Remark 10. (Integer <math>\geq 0</math>; if blank, then THETA = 0.0 is assumed.)</p>
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 6. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

#### Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. Grid points G1 to G8 must be numbered as shown in [Figure 9-53](#).
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between  $x_{\text{material}}$  and the line of constant  $\eta$ .

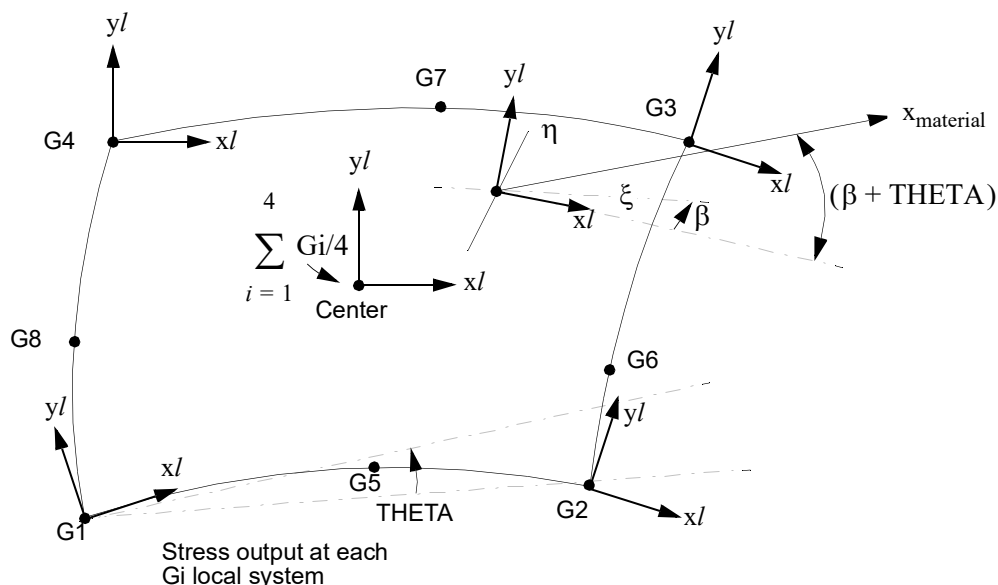
The definition of the material coordinate system by projection is used to calculate an angle THETA. Please note that since xi changes directions throughout the element based on element shape, the material coordinate system varies similarly. Because of this an orthotropic or anisotropic material will cause the CQUAD8's stiffness to be biased by both it's shape and grid ordering. Use the QUAD4 element if a constant material coordinate system direction is desired with orthotropic and anisotropic materials.

4. T1, T2, T3 and T4 are optional. If they are not supplied and no TFLAG, then T1 through T4 will be set to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge. If the edge point is located at the quarter point, the program may fail with a divide-by-zero error or the calculated stresses will be meaningless.
6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

- If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces incorrect. A User Warning Message is printed, and a CQUAD4 element is recommended instead. If the element is hyperelastic, then it is processed identically to the hyperelastic CQUAD4 element.
- For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by  $x_l$  and  $y_l$  in [Figure 9-53](#). However, for hyperelastic elements the stress and strain are output in the coordinate system identified by the CID field on the PLPLANE entry.
- For hyperelastic elements the plot codes are specified under the CQUADFD element name in [Item Codes](#).



where

- $\vec{e}_\eta$  is tangent to  $\eta$  at  $G_i$
- $\vec{e}_\xi$  is tangent to  $\xi$  at  $G_i$
- $\vec{A}$  is formed by bisection of  $\vec{e}_\eta$  and  $\vec{e}_\xi$
- $\vec{B}$  and  $\vec{A}$  are perpendicular

$y_l$  is formed by bisection of  $\vec{A}$  and  $\vec{B}$   
 $x_l$  is perpendicular to  $y_l$

**Figure 9-53 CQUAD8 Element Geometry and Coordinate Systems**

10. If element has an associated PSHLN2 or PLCOMP entry with BEHi=COMPS code, the THETA/MCID value on the element connection entry will be ignored.
11. For RC network solver in thermal analysis, the ZOFFS is ignored.
12. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

**CQUADR****Quadrilateral Plate Element Connection**

Defines an isoparametric membrane and bending quadrilateral plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CTRIAR element.

Format:

1	2	3	4	5	6	7	8	9	10
CQUADR	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	
		TFLAG	T1	T2	T3	T4			

Example:

CQUADR	82	203	31	74	75	32	2.6		
			1.77	2.04	2.09	1.80			

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP or PCOMPEntry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. See <a href="#">Figure 9-55</a> . (Real; Default = 0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:  CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis  CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line CORD1S, to the to the projected r-axis CORD2S  Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)
ZOFFS	Offset from the surface of grid point to the element plane. See Remark 8.

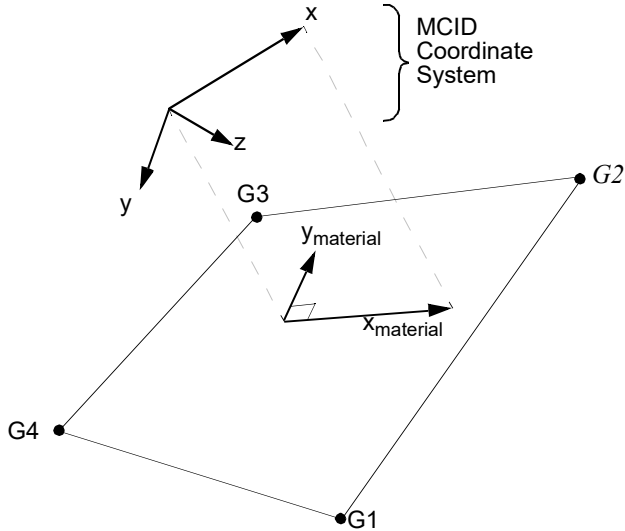


Figure 9-54 MCID Coordinate System Definition

TFLAG	An integer flag, signifying the meaning of the $T_i$ values. (Integer 0, 1, or blank)
$T_i$	Membrane thickness of element at grid points G1 through G4. If “TFLAG” zero or blank, then $T_i$ are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the $T_i$ are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) $T_i$ are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All the interior angles must be less than  $180^\circ$ .
4. The continuation is optional. If it is not supplied, then T1 through T4 will be set equal to the value of T on the PSHELL entry.
5. Stresses and strains are output in the element coordinate system at the centroid and grid points G1 through G4.
6. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4](#) entry (SORL option) and the [Consistent Surface and Edge Loads](#) in the *MSC Nastran Reference Guide* for additional information.

7. The CTRIAR element is the triangular companion to the CQUADR element and should be used instead of CTRIA3 or CTRIA6.

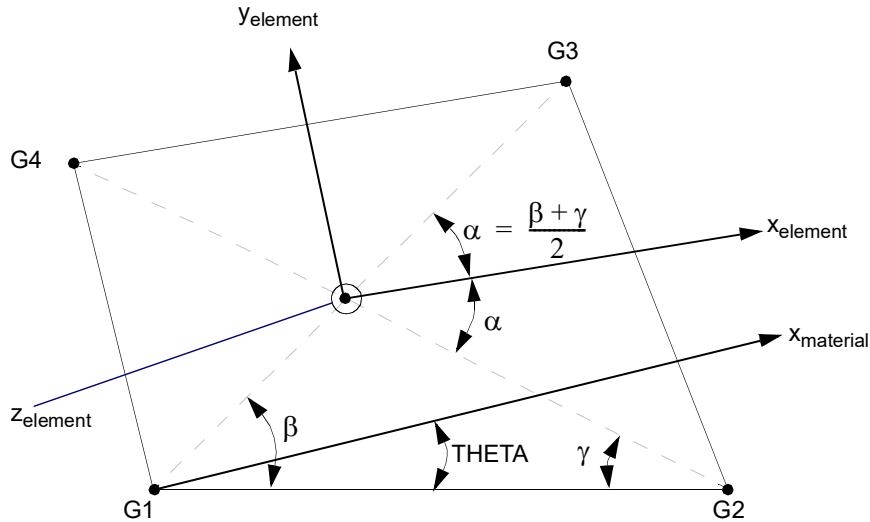


Figure 9-55 CQUADR Element Geometry and Coordinate Systems

8. Elements may be offset from the connection points by means of ZOFFS. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then the MID1 and MID2 fields must be specified on the PSHELL entry referenced both by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by MDLPRM, OFFDEF, NOMASS.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

9. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

## CQUADX Axisymmetric Quadrilateral Element (Fully Nonlinear or Linear Harmonic)

Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e., large strain and large rotations) analysis or a linear harmonic or rotordynamic analysis. The element has between four and nine grid points

Format:

1	2	3	4	5	6	7	8	9	10
CQUADX	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	THETA or MCID					

Example:

CQUADX	111	203	31	74	75	32			
--------	-----	-----	----	----	----	----	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000). See Remark 1.
PID	Property identification number of a PLPLANE or PAXSYMH or PLCOMP entry. (Integer > 0). See Remark 2.
G1, G2 G3, G4	Identification numbers of connected corner grid points. Required data for all four grid points. (Unique Integers > 0). See Remark 3., 6., 7.
G5, G6 G7, G8	Identification numbers of connected edge grid points. Optional data for any or all four grid points. (Integer ≥ 0 or blank). See Remark 3., 4., 6., 7.
G9	Identification number of center grid point. Optional. (Integer ≥ 0 or blank). Not used for linear harmonic elements. See Remark 3., 6., 7.
THETA	Material property orientation angle in degrees. THETA is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. For PSHLN2 or PLCOMP BEHi=AXCOMP code the THETA value on the element connection entry will be ignored. (Real; Default = 0.0). THETA is ignored if PID refers to a PAXSYMH entry.
MCID	Material coordinate system identification number, MCID is only applicable if PLCOMP referenced, or the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. For PSHLN2 BEHi=AXSOLID the resulting angle is measured relative to the R axis of the element. For PSHLN2 (or PLCOMP) BEHi=AXCOMP code the MCID value on the element connection entry will be ignored. (Integer ≥ 0; If blank, then THETA = 0.0 is assumed.)



## Remarks:

1. Element identification numbers should be unique with respect to all other element IDs of any kind.
2. If PID refers to a PLPLANE or PLCOMP entry, CQUADX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CQUADX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-56](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CQUADXFD element name in [Item Codes](#).
6. All Gi must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is multiplied by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50 \text{ N}$$

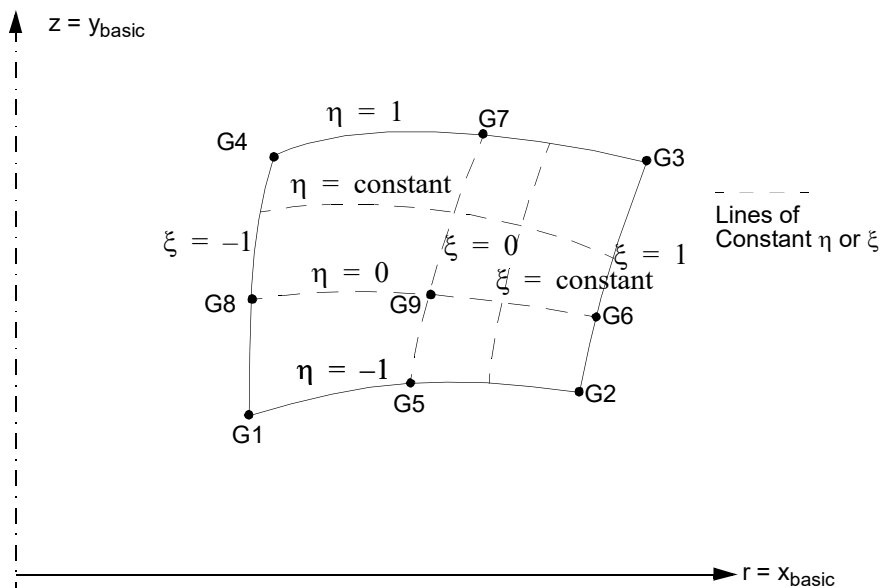


Figure 9-56 CQUADX Element Coordinate System

## CRAC2D Two-Dimensional Crack Tip Element

Defines a two-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC2D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18					

Example:

CRAC2D	114	108	2	5	6	8	7	11	
	12	14	16	17		20	22		

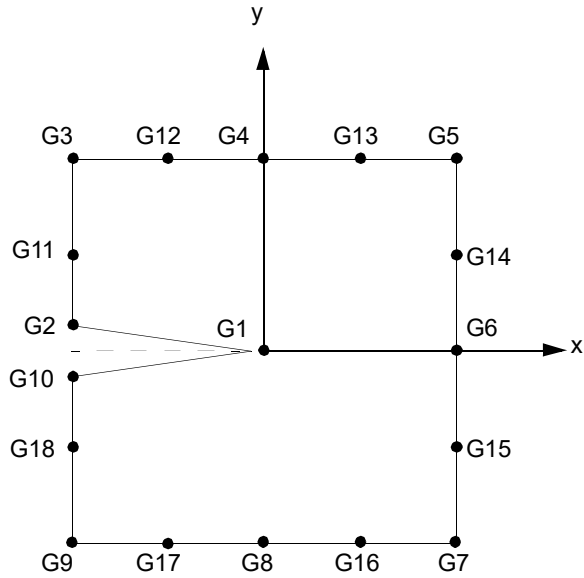
Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PRAC2D entry. (Integer > 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0; G11 through G18 may be blank.)

Remarks:

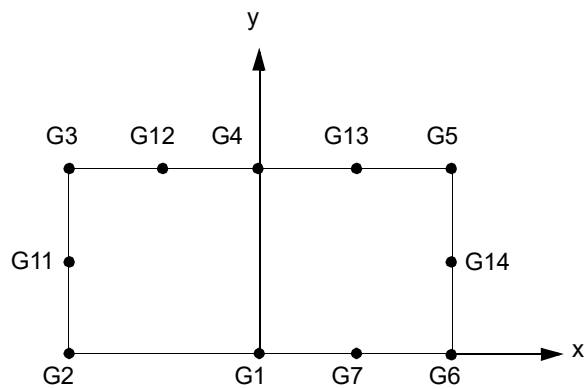
1. CRAC2D is a dummy element and requires the presence of this Bulk Data entry:

ADUM8	18	0	5	0	CRAC2D				
-------	----	---	---	---	--------	--	--	--	--

2. The element should be planar. Significant deviations will produce fatal errors.
3. Grid points G1 through G10 are required while grid points G11 through G18 are optional for the quadrilateral form of the element.
4. The stresses and stress intensity factors are calculated assuming that G2 and G10 are coincident. Deviations from this will produce erroneous results.
5. For the symmetric half-crack option, grid points G1 through G7 are required while grid points G11 through G14 are optional. Grid points G8 through G10 and G15 through G18 must not be present for this option.
6. The ordering conventions for the full-crack and half-crack options are shown in [Figure 9-57](#).
7. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-57](#) should be in the range 2.0 to 0.5.
8. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) in the *MSC Nastran Reference Guide*.



(a) Full Crack Option



(b) Symmetric Half-Crack Option

Figure 9-57 CRAC2D Element Connection for Full and Symmetric Options

## CRAC3D

## Three-Dimensional Crack Tip Element

Defines a three-dimensional crack tip element.

Format:

1	2	3	4	5	6	7	8	9	10
CRAC3D	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10	G11	G12	G13	G14	
	G15	G16	G17	G18	G19	G20	G21	G22	
	G23	G24	G25	G26	G27	G28	G29	G30	
	G31	G32	G33	G34	G35	G36	G37	G38	
	G39	G40	G41	G42	G43	G44	G45	G46	
	G47	G48	G49	G50	G51	G52	G53	G54	
	G55	G56	G57	G58	G59	G60	G61	G62	
	G63	G64							

Example:

CRAC3D	113	101	2	5	7	8	4	10	
	11	14	15	17		3	6	9	
	12		16		102	105	107	108	
	104	110	111	114	115	117		103	
	106	109	112		116		202	205	
	207	208	204	210	211	214	215	217	
	225	226							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PRAC3D entry. (Integer < 0)
Gi	Grid point identification numbers of connection points. (Integer ≥ 0)

Remarks:

1. CRAC3D is a dummy element and requires the presence of this Bulk Data entry:

ADUM9	64	0	6	0	CRAC3D				
-------	----	---	---	---	--------	--	--	--	--

2. Element identification numbers should be unique with respect to all other element identification numbers.

3. This element, including grid point numbering conventions, is shown in [Figure 9-58](#) and [Figure 9-59](#). Grid points G1 through G10, and G19 through G28 are required; midside and surface grid points G11 through G18, G29 through G36, and G37 through G64 are optional. Either all or none of grid points G37 through G46 should be present. A fatal error message will be issued for partial connectivity.
4. The ratio of the element dimensions in the y to x axis shown for the element coordinate system in [Figure 9-58](#) should be in the range 2.0 to 0.5.
5. For the symmetric half-crack option Grid Points G1 through G7, and G19 through G25 are required, whereas grid points G11 through G14, G29 through G32, and G37 through G42 are optional. Grid points G8 through G10, G15 through G18, G26 through G28, G33 through G36, G43 through G46, G51 through G55, and G60 through G64 should not be specified to invoke this option.
6. It is recommended that both the faces (formed by grid points G2 through G18 and grid points G20 through G36) and the midplane (formed by grid points G37 through G46 and grid points G37 through G46) be planar. It is also recommended that midside grid points G37 through G46 be located within the middle third of the edges.
7. The midside nodes on both the faces should be defined in pairs. For example, if grid point G11 is not defined, then grid point G29 should not be defined and vice versa.
8. The stresses and stress intensity factors are calculated with the assumptions that grid points G2 and G10, G20 and G28, and G38 and G46 are coincident. Deviation from this condition will produce erroneous results.
9. The stress output is interpreted as shown in [Crack Tip Elements \(CRAC2D, CRAC3D\)](#) in the *MSC Nastran Reference Guide*.
10. As depicted in [Figure 9-58](#) and [Figure 9-59](#), the element is a right-handed element. Thus define the vectors  $\overline{G1G9}$  and  $\overline{G1G3}$ , then the cross-product  $\overline{G1G9} \times \overline{G1G3}$  points to the face defined by G19, G20, ...

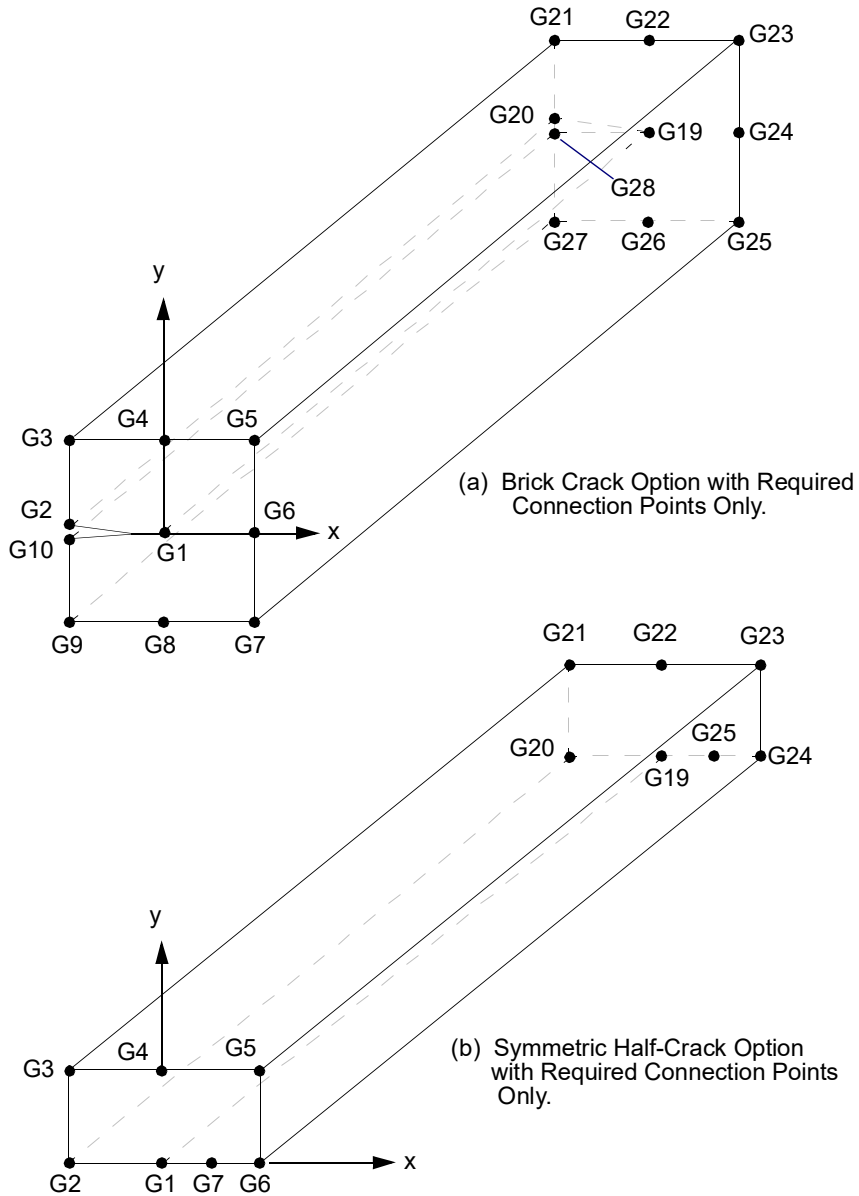


Figure 9-58 CRAC3D Solid Crack Tip Element with Required Connection Points Only

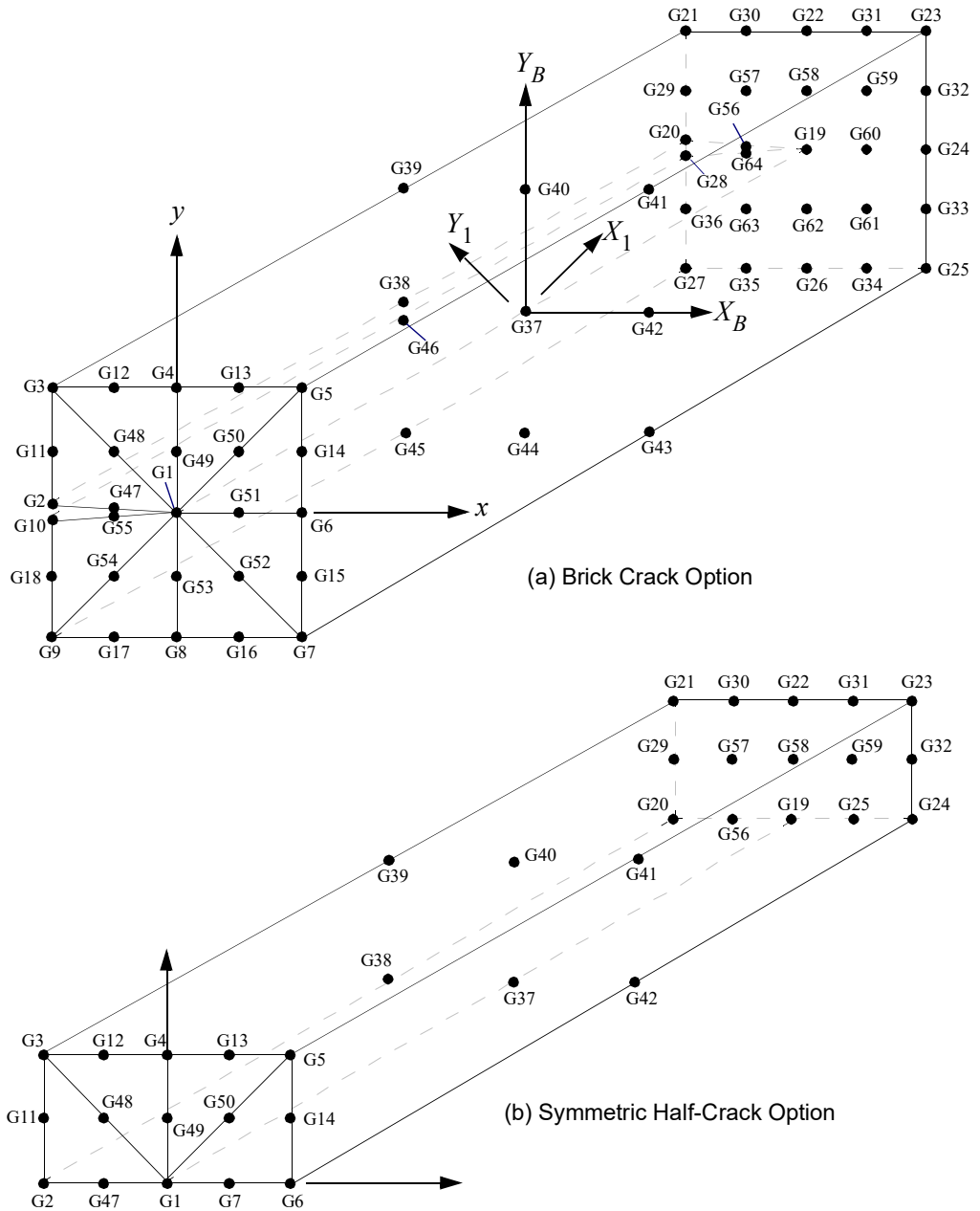


Figure 9-59 CRAC3D Solid Crack Tip Element with All Connection Points

## CREEP

## Creep Characteristics

Defines creep characteristics based on experimental data or known empirical creep law. This entry will be activated if a MAT1, MAT2, or MAT9 entry with the same MID is used and the NLPARM entry is prepared for creep analysis. The creep formulation is principally suited for isotropic materials and, in general, when used with anisotropic materials may produce incorrect results. However, slightly anisotropic materials may produce acceptable results.

Format:

1	2	3	4	5	6	7	8	9	10
CREEP	MID	T0	EXP	FORM	TIDKP	TIDCP	TIDCS	THRESH	
	TYPE	a	b	c	d	e	f	g	

Example:

CREEP	8	1100.		CRLAW					
	121	6.985-6	2.444	7.032-4	0.1072	6.73-9	0.1479	3.0	

Describer	Meaning
MID	Material identification number of a MAT1, MAT2, or MAT9 entry. (Integer > 0)
T0	Reference temperature at which creep characteristics are defined. See Remark 2. (Real; Default = 0.0)
EXP	Temperature-dependent term, $e^{(-\Delta H/(R \cdot T0))}$ , in the creep rate expression. See Remark 2. (0.0 < Real ≤ 1.0; Default = 1.0E-9)
FORM	Form of the input data defining creep characteristics. (Character: “CRLAW” for empirical creep law, or “TABLE” for tabular input data of creep model parameters.)
TIDKP, TIDCP, TIDCS	Identification number of a TABLES1 entry, which defines the creep model parameters $K_p(\sigma)$ , $C_p(\sigma)$ , and $C_s(\sigma)$ , respectively. See Remarks 3. through 4. (Integer > 0)
THRESH	Threshold limit for creep process. Threshold stress under which creep does not occur is computed as THRESH multiplied by Young’s modulus. (0.0 < Real < 1.0E-3; Default = 1.0E-5)
TYPE	Identification number of the empirical creep law type. See Remark 1. (Integer: 111, 112, 121, 122, 211, 212, 221, 222, or 300)
a through g	Coefficients of the empirical creep law specified in TYPE. Continuation should not be specified if FORM = “TABLE”. See Remark 1. (Real)

Remarks:

- Two classes of empirical creep law are available.

Creep Law Class 1



The first creep law class is expressed as:

$$\varepsilon^c(\sigma, t) = A(\sigma)[1 - e^{-R(\sigma)t}] + K(\sigma)t \tag{9-3}$$

Parameters  $A(\sigma)$ ,  $R(\sigma)$ , and  $K(\sigma)$  are specified in the following form, as recommended by Oak Ridge National Laboratory:

Parameter	Function 1	Digit	Function 2	Digit
$A(\sigma)$	$a\sigma^b$	i=1	$ae^{b\sigma}$	i=2
$R(\sigma)$	$ce^{d\sigma}$	j=1	$c\sigma^d$	j=2
$K(\sigma)$	$e \cdot [\sinh(f\sigma)]^g$	k=1	$ee^{f\sigma}$	k=2

TYPE=ijk where i, j, and k are digits equal to 1 or 2, according to the desired function in the table above. For example, TYPE=122 defines  $A(\sigma) = a\sigma^b$ ,  $R(\sigma) = c\sigma^d$ , and  $K(\sigma) = ee^{f\sigma}$ .

### Creep Law Class 2

The second creep law class (TYPE=300) is expressed as:

$$\varepsilon^c(\sigma, t) = a\sigma^b t^d \tag{9-4}$$

where the values of b and d must be defined as follows:

The above expression is determined by curve fitting using a Newton-Raphson procedure based on the expressions:

$$\lambda = C_p / K_{pi} = \left( \frac{\frac{-c}{\varepsilon}}{a\sigma^b} \right)^{1/d} \tag{9-5}$$

where  $\bar{\sigma}$  is the effective stress and the nonlinear equation:

$$\lambda^2(e^{t_0/\lambda} - 1) - \lambda t_0 - t_0^2 / d = 0$$

Then the creep model parameters are determined by:

$$\frac{1}{C_s} = \frac{\bar{\varepsilon}^c d}{t_0 \bar{\sigma}} \left[ 1 - \frac{\lambda(1-d)}{t_0} \right]$$

$$\frac{1}{K_p} = \left( \frac{\lambda}{t_0} \right)^2 \frac{\bar{\varepsilon}^c}{\bar{\sigma}} (1-d)d - \frac{t_0}{C_s} + \frac{\bar{\varepsilon}^c}{\bar{\sigma}}$$

$$C_p = \lambda K_p$$

To keep the solution of these expressions stable, the values of b and d are recommended to be defined as follows:

$$1.0 < b < 8.0$$

and

$$0.2 < d < 2.0$$

The coefficient g should be blank if TYPE = 112, 122, 222, or 212 and c, e, f, and g should be blank if TYPE = 300. The coefficients a through g are dependent on the structural units; caution must be exercised to make these units consistent with the rest of the input data.

- Creep law coefficients a through g are usually determined by least squares fit of experimental data, obtained under a constant temperature. This reference temperature at which creep behavior is characterized must be specified in the T0 field if the temperature of the structure is different from this reference temperature. The conversion of the temperature input (°F or °C) to °K (degrees Kelvin) must be specified in the PARAM,TABS entry as follows:

PARAM,TABS,273.16 (If Celsius is used.)

PARAM,TABS,459.69 (If Fahrenheit is used.)

When the correction for the temperature effect is required, the temperature distribution must be defined in the Bulk Data entries (TEMP, TEMPP1 and/or TEMPRB), which are selected by the Case Control command TEMP(LOAD) = SID within the subcase.

From the thermodynamic consideration, the creep rate is expressed as:

$$\dot{\varepsilon}^C = \dot{\varepsilon}_A (e^{-\Delta H/RT}) \quad (9-6)$$

where:

$\Delta H$  = energy of activation

$R$  = gas constant (= 1.98 cal/mole °K)

$T$  = absolute temperature (°K)

$\dot{\varepsilon}_A$  = strain/sec per activation

If the creep characteristics are defined at temperature T0, the creep rate at temperature T is corrected by a factor

$$\frac{\dot{\varepsilon}^c}{\dot{\varepsilon}_o^c} = \text{EXP} \left( \frac{T_0}{T} - 1 \right) \quad (9-7)$$

where:

$$\dot{\varepsilon}^c = \text{corrected creep rate}$$

$$\dot{\varepsilon}_o^c = \text{creep rate at } T_0$$

$$\text{EXP} \left( \frac{T_0}{T} - 1 \right) = \text{correction factor}$$

3. Creep model parameters  $K_p$ ,  $C_p$ , and  $C_s$  represent parameters of the uniaxial rheological model as shown in Figure 9-60.

Tabular values ( $X_i$ ,  $Y_i$ ) in the TABLES1 entry correspond to  $(\sigma_i, K_{pi})$ ,  $(\sigma_i, C_{pi})$ , and  $(\sigma_i, C_{si})$  for the input of  $K_p$ ,  $C_p$ , and  $C_s$ , respectively. For linear viscoelastic materials, parameters  $K_p$ ,  $C_p$ , and  $C_s$  are constant and two values of  $\sigma_i$  must be specified for the same value of  $K_{pi}$ ,  $C_{pi}$ , and  $C_{si}$ .

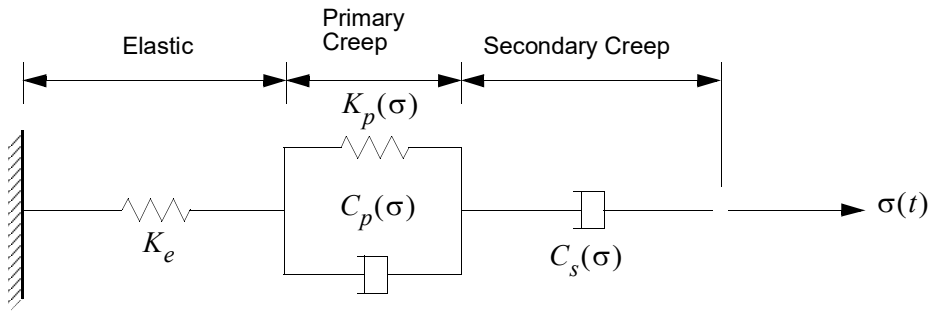


Figure 9-60 CREEP Parameter Idealization

Creep model parameters, as shown in Figure 9-61 through Figure 9-63, must have positive values. If the table look-up results in a negative value, the value will be reset to zero and a warning message (TABLE LOOK-UP RESULTS IN NEGATIVE VALUE OF CREEP MODEL PARAMETER IN ELEMENT ID=\*\*\*\*) will be issued.

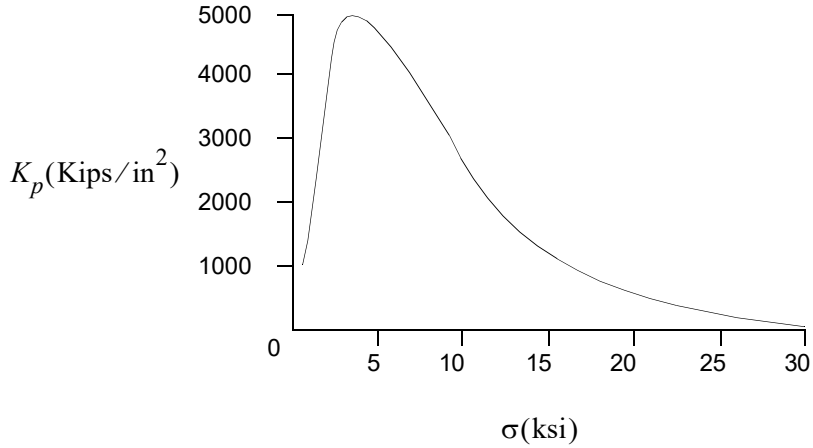


Figure 9-61  $K_p$  Versus  $\sigma$  Example for CREEP

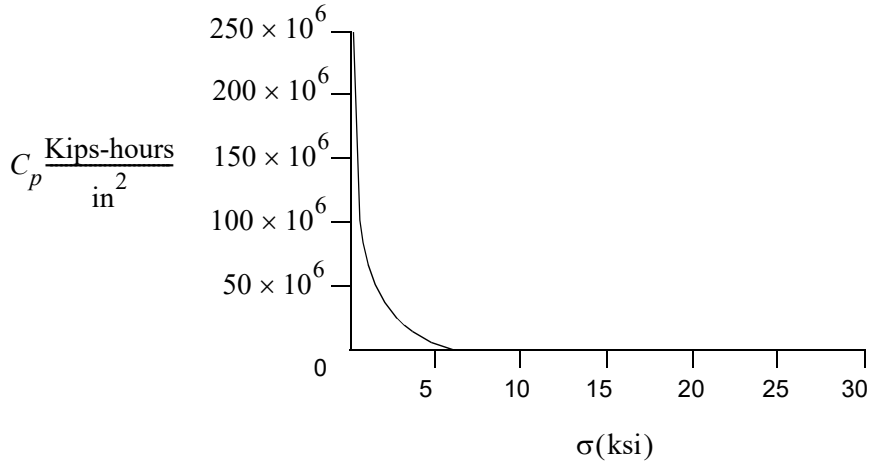
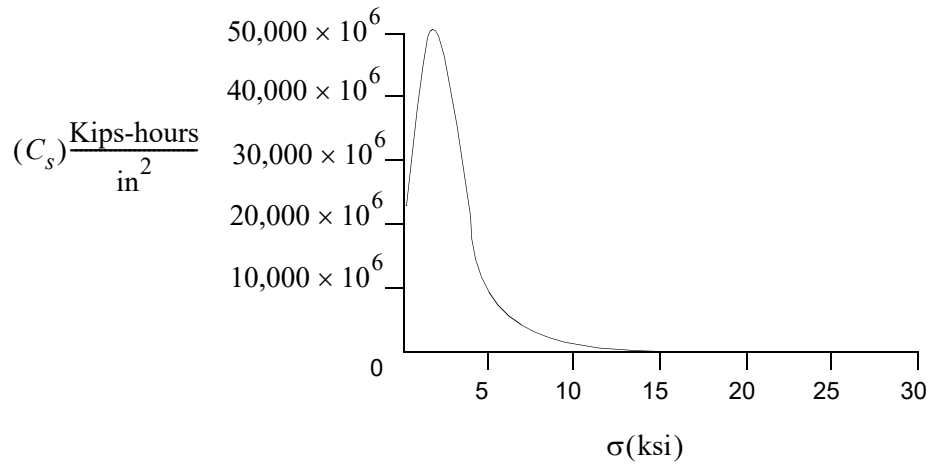


Figure 9-62  $C_p$  Versus  $\sigma$  Example for CREEP

Figure 9-63  $C_s$  Versus  $\sigma$  Example for CREEP

- Creep analysis requires an initial static solution at  $t = 0$ , which can be obtained by specifying a subcase that requests an NLPARM entry with  $DT = 0.0$ .



## CROD Rod Element Connection

Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	12	13	21	23					
------	----	----	----	----	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PROD entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See [CONROD](#) for alternative method of rod definition.
3. Only one element may be defined on a single entry.

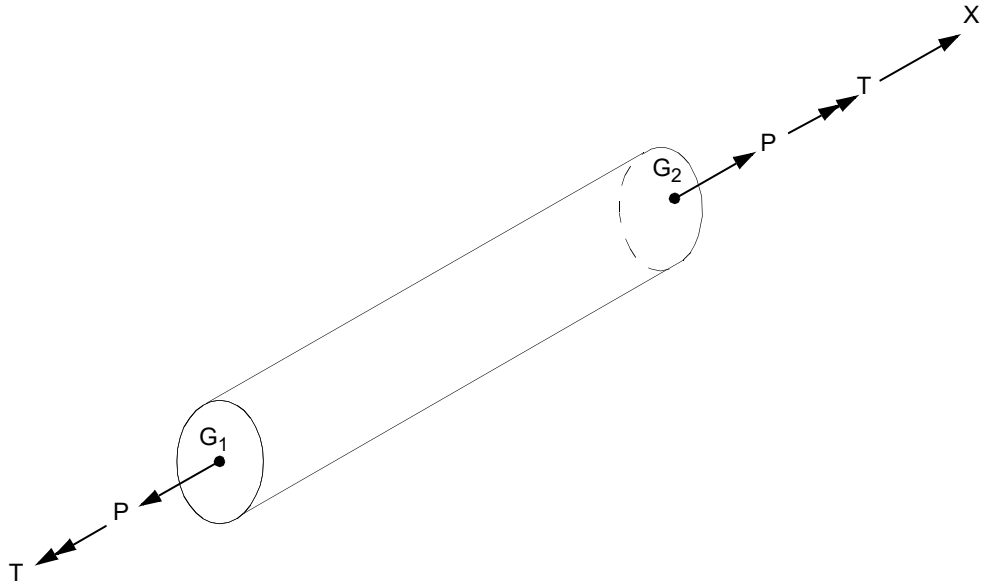


Figure 9-64 CROD Element Internal Forces and Moments

## CSEAM A Shell Patch SEAM Connection

Defines a SEAM connecting two surface patches.

Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE							

Alternate Format:

1	2	3	4	5	6	7	8	9	10
CSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE			

Example:

CSEAM	552	297			43	48			
	30422	77987							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSEAM entry. (Integer > 0)
SMLN	SEAM line identification. See Remark 2. (CHAR or blank)
CTYPE	Connectivity search type. (Character) If CTYPE = "PSHELL", IDAS and IDBS are property identification numbers of PSHELL's. (Default) If CTYPE = "ELEM", IDAS and IDBS are element identification numbers.
IDAS, IDBS	Used to define patch A and B or the start of patch A or B for a tailored blank. See Remark 4. (Integer > 0) If CTYPE = "PSHELL", required property id defining patches A and B. If CTYPE = "PSHELL" and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom. See Remark 6. If CTYPE = "ELEM", required element id defining patches A and B. IDAS ≠ IDBS.
IDAE, IDBE	Used to define the end of patch A and the end of patch B for a tailored blank. See Remark 4. (Integer ≥ 0 or blank) If CTYPE = "PSHELL", property id defining patches A and B. If CTYPE = 'PSHELL' and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom. If CTYPE = "ELEM", element id defining patches A and B. IDAE ≠ IDBE.



Describer	Meaning
GS, GE	Grid ids of piercing points on patches A and B of the Start and End of the SEAM. (Integer > 0)
XS,YS,ZS	Location of the SEAM Start. (Real or blank)
XE,YE,ZE	Location of the SEAM End. (Real or blank)

**Remarks:**

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. With no embedded blanks any combination of up to eight of the acceptable characters in MSC Nastran may be employed for the SMLN entry.

The seam line will be considered continuous between each connected element and where any two elements have a common face, the faces of the resulting internal CHEXA's will be adjusted to a single common face. If a CSEAM's GS or GE is not common to the GE or GS of any other CSEAM the faces will not be adjusted.

A SMLN cannot have a branch.

3. GS and GE define the start and end points of the SEAM element. At these points and using the value *W* specified on the PSEAM entry, surface patches A and B are determined. Points are projected onto the surface patches A and B with the four points at end GS and the four points at end GE then used to form faces of a CHEXA element.

The auxiliary points forming the faces of the CHEXA element are then connected to the physical grids of the patches. The number of unique physical grids per patch ranges from a possibility of 6 to 64 grids.

The auxiliary points must have a projection on patches A and B, but they do not have to lie on patch A or B.

A maximum of three shell elements of patch A and three shell elements of patch B can be connected with one CSEAM element, see [Figure 9-65](#).

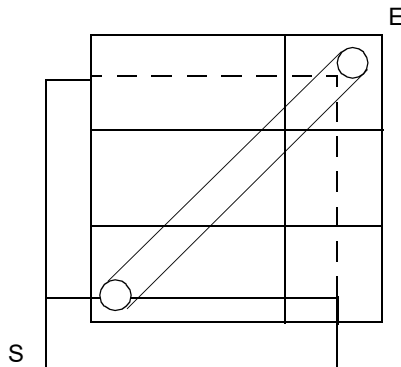


Figure 9-65 Connected Shell Elements for a CSEAM Element

## 4. For CTYPE = 'PSHELL'

If patch A is uniform in thickness, then only its IDAS is needed to define it.

If patch B is uniform in thickness, then only its IDBS is needed to define it.

If patch A has stepped tapering, then IDAS and IDAE are used to define it.

If patch B has stepped tapering, then IDBS and IDBE are used to define it.

## 5. Projection Algorithms for the CSEAM Elements

Because of complex geometry, the user supplied start point GS may not have a projection SA and SB, and the end point GE may not have a projection EA and EB. Even though these four projection points are found, the program still has to find projections for the eight auxiliary points SA1, SA2, ..., EB2, and EB1 of the HEXA. The default projection strategy can be changed by overwriting the default values of the flags and parameters in the SWLDPRM Bulk Data entry.

## a. Find Projections for SA, SB, EA, and EB

For CTYPE = "PSHELL", the program finds the closest shell grids to GS and GE. The shell elements that are connected to these closest grids are defined as the candidate shell elements. While looping through each candidate shell element to compute the projection of GS and GE onto that element, the program always tries to get the most accurate projection. Even though a projection is found with PROJ TOL > 0.0, the program still continues the projection calculations using PROJ TOL=0.0. If a projection is found with PROJ TOL=0.0, that shell element will be selected as the connecting element. Otherwise, the shell element that gets projection with PROJ TOL>0.0 is selected as EIDSA, EIDSB, EIDEA, or EIDEB. For CTYPE = "ELEM", the above processes are skipped, because EIDSA, EIDSB, EIDEA, and EIDEB have already been specified by the user.

If GSTOL > 0.0 and the distance GS-SA, GS-SB, GE-EA, or GE-EB is greater than GSTOL, a UFM 7549 is issued and the CSEAM element is rejected.

If the projection of GS or GE lies outside the shell sheet, or the connected shell elements fail the geometry check with GMCHK > 0, the program will issue a UFM and the CSEAM element will be rejected.

If GMCHK > 0, the program checks errors of CSEAM across a cutout or over a corner with elements in plane. The program also computes the angle between the shell normal vectors of EIDSA and EIDEA and the angle between the shell normal vectors of EIDSB and EIDEB to check a corner with elements out of plane.

For CTYPE = "PSHELL", if there is an error detected, the program loops back to compute the other possibility of projection until a correct connection is found or all candidate shell elements are processed. In the latter case, either UFM 7638 (the seam spans a cutout) or UFM 7667 (the seam spans a corner) will be issued. If GMCHK=2, the program also lists all candidate shell elements with their projection status for each connecting type after issuing a UFM. This will help the user to select the correct shell elements for EIDSA, EIDSB, EIDEA, and EIDEB.

For CTYPE="ELEM", the program only checks errors and issues UFM 7638 or 7667 for the kind of error encountered. No looping back will be performed.

If  $GMCHK > 0$  and  $GSPROJ \geq 0.0$ , the program also computes the angle between the shell normal vectors of EIDSA and EIDSB and the angle between the shell normal vectors of EIDEA and EIDEB. A UFM 7595 is issued if the angle between the shell normal vectors is greater than  $GSPROJ$ . By default,  $GSPROJ = 20^\circ$ , that means the shell patches A and B can be tilted relative to each other by not more than  $20^\circ$ .

b. Find Projections of the Eight Auxiliary Points SA1, SA2, ..., EB2, and EB1

After the projections for SA, SB, EA, and EB have been found, eight auxiliary points for an internal hexagonal polygon are formed. If the GS or GE of a CSEAM element is connected to the GE or GS of another CSEAM element, then the internal HEXA elements are adjusted to a common face.

If  $GSPROJ \geq 0.0$  and the angle between the face vectors parallel to the thickness direction of the internal HEXA and the normal vector of the shell element that gets projection exceeds  $GSPROJ$ , the program will skip picking this shell element and will proceed to process next candidate shell element.

The most common error condition occurs when the seam lies on the edge of the shell patches. Under this situation, half of the seam hangs outside the shell sheets (Figure 9-66). It is required that each of the eight points has a projection. If at least one point does not have a projection and  $GSMOVE > 0$ , GS will be moved by  $W/2$ . Same algorithms apply to end E. The move will be repeated until either all projections are found or the number of moves reaches  $GSMOVE$ .

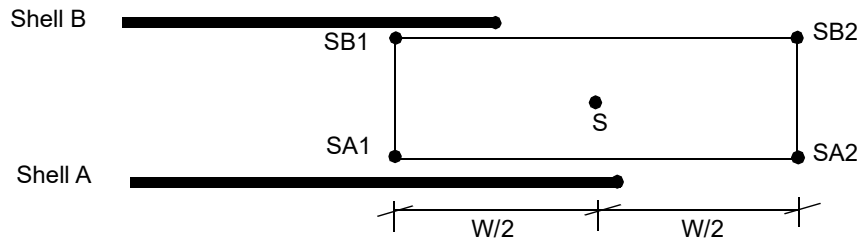


Figure 9-66 Seam Weld at an Edge

c. Error Checks by GMCHK Parameter

The GMCHK parameter specified in the SWLDPRM Bulk Data entry checks the errors of CSEAM elements across cutouts or over corners. There are three allowable values of GMCHK.

- GMCHK = 0 (Default) Do not check errors
- GMCHK = 1 Check errors
- GMCHK = 2 Check errors and output all candidate shell elements if there is an error encountered

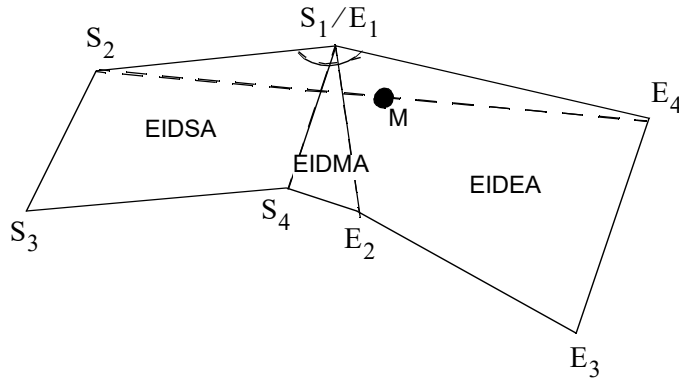
If GMCHK is turned on, Nastran will perform the following checking while searching for the projected shell elements. Note that EIDSA is the shell element that gets projection from GS on shell A; EIDEA is the shell element that gets projection from GE on shell A. Same algorithms are applied to EIDSB and EIDEB for shell B.

d. Check the CSEAM Across a Cutout or Over a Corner with Elements in Plane

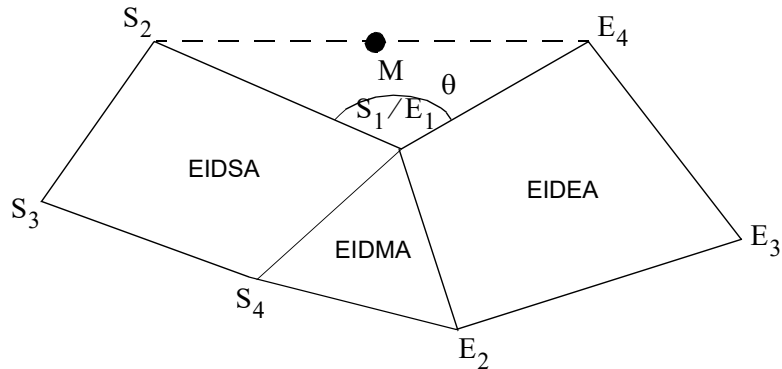
- If EIDSA is equal to EIDEA, the seam lies within one element. No checks are required.

- If EIDSA and EIDEA share two corner grids, these elements are adjacent. No checks are required.
- If EIDSA and EIDEA share only one corner grid, the seam is over a corner. There are two exceptions:

There exists a shell element (EIDMA) that shares two corner grids with EIDSA and EIDEA. Also, either the angle  $\theta$  between vector  $S_1S_2$  and vector  $E_1E_4$  is greater than CNRAGLI degrees or the middle point (M) of line segment  $S_2E_4$  projects to EIDSA, EIDMA, or EIDEA.

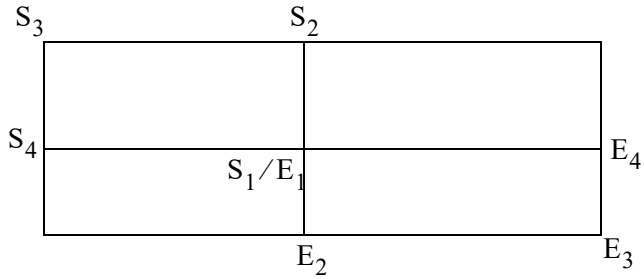


This model is acceptable - CONVEX ( $\theta > \text{CNRAGLI}$ ).



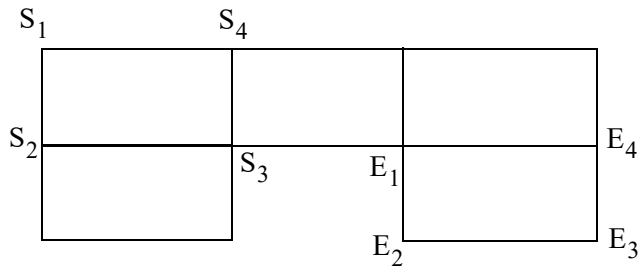
This model fails - not CONVEX ( $\theta < \text{CNRAGLI}$  and point M does not project to EIDSA, EIDMA or EIDEA).

This shared grid is a shell grid of another two different shell elements.

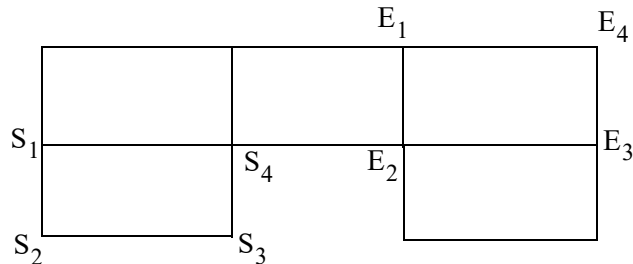


- If EIDSA and EIDEA do not share any corner grid, Nastran will check if there is an element (EIDMA) lying between EIDSA and EIDEA. EIDMA must share two corner grids with EIDSA and another different one corner grid with EIDEA, or vice versa. The following five examples demonstrate the acceptable and failed cases.

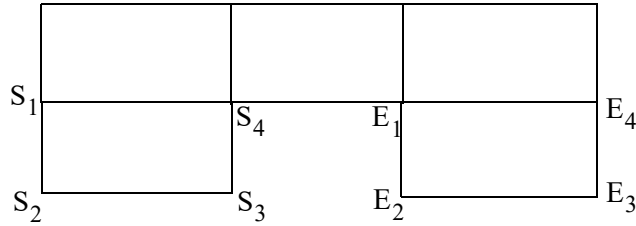
EIDMA shares one edge with EIDSA and shares one corner grid with EIDEA. This case is acceptable.



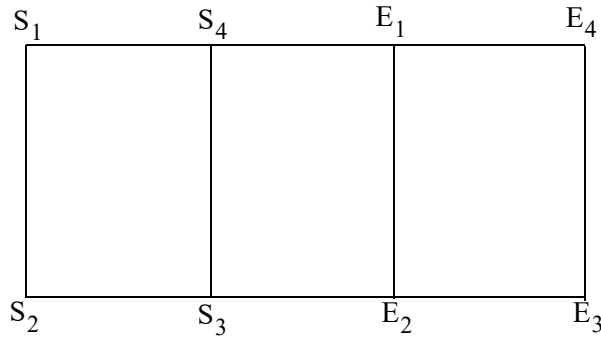
EIDMA shares one edge with IEDEA and shares one corner grid with EIDSA. This case is acceptable.



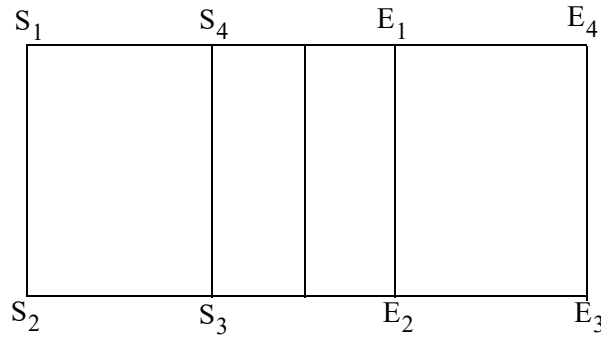
EIDMA shares one corner grid with EIDSA and shares another corner grid with EIDEA. An error is detected because the seam spans a cutout.



EIDMA shares one edge with EIDSA and shares another edge with EIDEA. This case is acceptable.

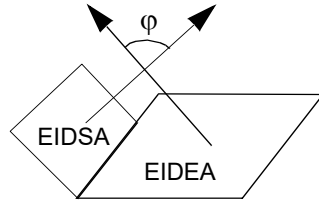


There does not exist a single element that shares an edge or corner grid with EIDSA or EIDEA. An error is detected because the length of the seam spans more than three elements.



- e. Check the CSEAM Over a Corner with Elements Out of Plane

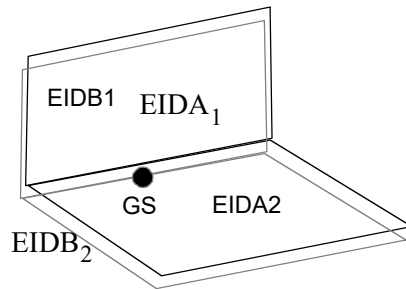
The CNRAGLO parameter is used to check the error of a seam over a corner with EIDSA and EIDEA not lying on a same plane. An error is detected if the angle  $\varphi$  between the shell normal vectors of EIDSA and EIDEA is greater than CNRAGLO. The default value of CNRAGLO is  $20^\circ$ . No angles will be checked if CNRAGLO = -1.



This model fails. ( $\varphi > \text{CNRAGLO}$ )

f. Modeling Guidelines

When there exist multiple pairs of connections, it is recommended that either the GMCHK and GSPROJ flags be turned on to filter out tilted connections or the ELEM option be used to specify the IDs of the connected shell elements directly. For example, if EIDA1 is connected to EIDB2 or EIDA2 is connected to EIDB1, the element tangent vectors will be computed wrong and the auxiliary points will not be able to find connected shell elements.



- The projection algorithm for the two-sided option will be the same as in Remark 5. above once the two patches A and B have been established. The program will find the closest shell grids to GS and GE as usual for candidate shell elements for patch A.

It will compute the normal for the candidate patch A (similar for GE) and for the candidate patch B. If the normals are approximately aligned (within a tolerance) the algorithm will proceed as in Remark 5.

If the normals of the candidate patch's A and B do not align within a specified tolerance, the algorithm will use another set of pairs of grids for candidate patches to find a new patch A and B. If their normals align within a specified tolerance it will proceed as in Remark 5. If after processing all reasonable pairs of patches, no alignment of normals are found or the patches A and B at GE have different normal alignment from the patches A and B at GB, a user fatal will be issued.

**Note:** For the two-sided option, GS and GB must lie between patches A and B. Also, the shell elements that get projections from GS/GE cannot share a common shell grid. This option always selects the patch with the shell grids closest to GS/GE as patch A. Avoid having GS/GE exactly midway between the two patches.

7. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
8. The CSEAM contributes to MASS if its PSEAM entry has an associated MATi entry with a non-zero density. PARAM,COUPMASS effects the mass calculation. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
9. If partitioned superelements are present, then CSEAM is supported in the main Bulk Data section only.



## CSET Free Boundary Degrees-of-Freedom

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

CSET	124	1	5	23	6	16			
------	-----	---	---	----	---	----	--	--	--

Describer	Meaning
IDi	Grid or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points. No embedded blanks.)

Remarks:

1. CSET and BNDFREE entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSETi/BNDFIXi or CSETi/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSETi/BNDFIXi entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSETi/BNDFREE/BNDFRE1 entries present or both BSETi/BNDFIXi and CSETi/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSETi/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSETi/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.
5. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:

- a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
- b. Linear perturbation with:
  - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
  - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
- c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## CSET1 Free Boundary Degrees-of-Freedom, Alternate Form of CSET Entry

Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

Format:

1	2	3	4	5	6	7	8	9	10
CSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	ID10	-etc.-					

Example:

CSET1	124	1	5	7	6	9	12	122	
	127								

Alternate Formats and Examples:

CSET1	C	ID1	“THRU”	ID2					
CSET1	3	6	THRU	32					

CSET1		“ALL”							
CSET1		ALL							

Describer	Meaning
C	Component numbers. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.)
ID <sub>i</sub>	Grid or scalar point identification numbers. (Integer > 0; For “THRU” option, ID <sub>1</sub> < ID <sub>2</sub> )

Remarks:

1. CSET1 and BDNFRE1 entries are equivalent to each other. Either one of them or any combination of them may be employed.
2. If there are no BSET<sub>i</sub>/BNDFIX<sub>i</sub> or CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, all a-set points are considered fixed during component mode analysis. If there are only BSET<sub>i</sub>/BNDFIX<sub>i</sub> entries present, any a-set degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are only CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present or both BSET<sub>i</sub>/BNDFIX<sub>i</sub> and CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries present, the c-set degrees-of-freedom are defined by the CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries, and any remaining a-set points are placed in the b-set.
3. Degrees-of-freedom specified on CSET<sub>i</sub>/BNDFREE/BNDFRE1 entries form members of the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.
5. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-6. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:
    - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## CSHEAR Shear Panel Element Connection

Defines a shear panel element and effective extensional stiffener rods.

Format:

1	2	3	4	5	6	7	8	9	10
CSHEAR	EID	PID	G1	G2	G3	G4			

Example:

CSHEAR	3	6	1	5	3	7			
--------	---	---	---	---	---	---	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHEAR entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2 ≠ G3 ≠ G4)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Grid points G1 through G4 must be ordered consecutively around the perimeter of the element.
3. All interior angles must be less than 180°.

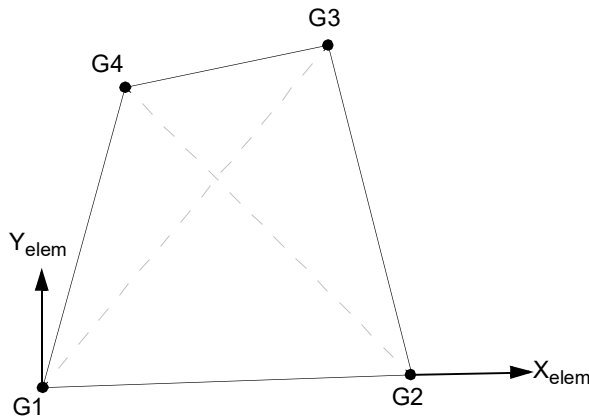


Figure 9-67 CSHEAR Element Connection and Coordinate System

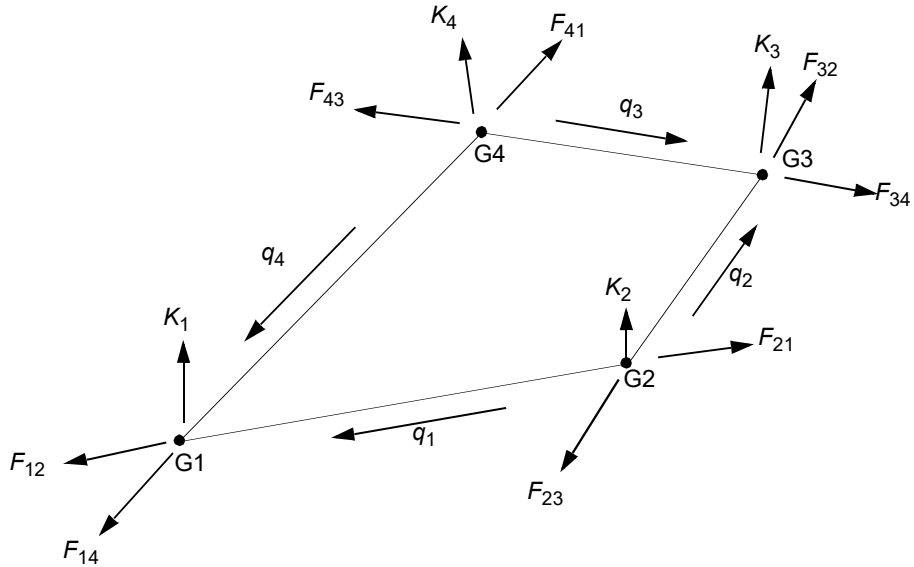
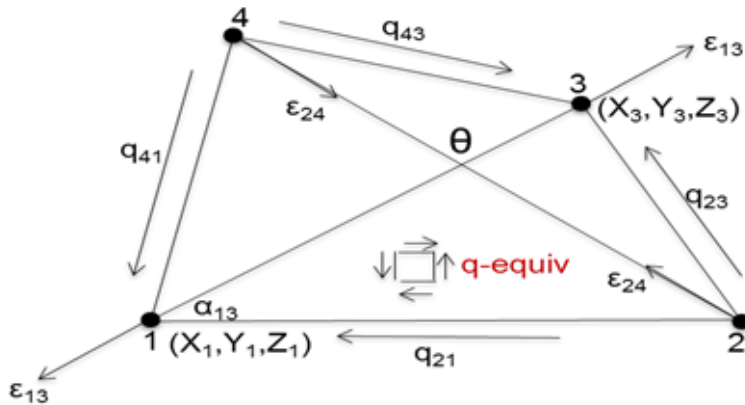


Figure 9-68 CSHEAR Element Corner Forces and Shear Flows

4. The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: The Harder panel is based on the following:  
Consider the following quadrilateral shear panel.



The shear stress  $\tau$  is related to the shear strain by the relationship  $\tau = G\gamma$  where G is the shear modulus.

Determining the value for shear for the general quadrilateral shown is a heuristic process. For the Harder element it is determined in the following manner. The strain along a diagonal such as L13 is found from the expression (or Mohr's circle):

$$\varepsilon_{13} = \varepsilon_x \cos^2 \alpha_{13} + \varepsilon_y \sin^2 \alpha_{13} + \gamma_{xy} \cos \alpha_{13} \sin \alpha_{13}$$

With a similar expression for  $\varepsilon_{24}$ , where  $\alpha_{24} = 180 - (\alpha_{13} + \theta)$ . For shear panels, the direct strains are negligible compared to the shear strain. For a rectangular shear panel  $\alpha_{24} = \alpha_{13}$  and we have

$$\Delta\varepsilon = \varepsilon_{13} - \varepsilon_{24} = \gamma \sin \theta \text{ or } \gamma = (\varepsilon_{13} - \varepsilon_{24}) / (\sin \theta).$$

In the above, the subscripts have been dropped from the term for shear strain. The Harder shear panel uses this definition for the measure of shear strain in the general quadrilateral. For a rectangular panel it is exact. For a reasonable panel it is within an error of < 1 degree. To insure reasonableness, all the geometry checks required by the Garvey element are still performed.

### Stress Recovery:

The output for either the Garvey or the Harder panel is the same with the following exception:

MAX SHEAR	AVG SHEAR	SHEARP
MAX( $\tau_1, \tau_2, \tau_3, \tau_4$ )	$(\tau_1 + \tau_2 + \tau_3 + \tau_4) / 4$	GARVEY
MAX( $\tau_1, \tau_2, \tau_3, \tau_4$ )	(q-Equiv) / $\tau$	HARDER

For both Garvey and Harder panels, q-Equiv is the shear flow on which all stress calculations are based.

## CSLOT3

## Three Point Slot Element Connection

Defines an element connecting three points that solve the wave equation in two dimensions. Used in the acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT3	EID	IDS1	IDS2	IDS3		RHO	B	M	

Example:

CSLOT3	100	1	3	2		3.0-3		6	
--------	-----	---	---	---	--	-------	--	---	--

Descriptor	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
IDS <sub>i</sub>	Identification number of connected GRIDS points. ( $\text{Integer} > 0$ )
RHO	Fluid density in mass units. ( $\text{Real} > 0.0$ ; Default is the value of RHOD on the AXSLOT entry)
B	Fluid bulk modulus. ( $\text{Real} \geq 0.0$ ; Default is the value of BD on the AXSLOT entry)
M	Number of slots in circumferential direction. ( $\text{Integer} \geq 0$ ; Default is the value of MD on the AXSLOT entry)

Remarks:

1. CSLOT3 is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates three plot elements, connecting points IDS1 to IDS2, IDS2 to IDS3, and IDS3 to IDS1.
5. If B=0.0, then the slot is considered to be an incompressible fluid.
6. If M=0, then no matrices for CSLOT3 elements are generated.



## CSLOT4

## Four Point Slot Element Connection

Defines an element connecting four points that solve the wave equation in two dimensions. Used in acoustic cavity analysis for the definition of evenly spaced radial slots.

Format:

1	2	3	4	5	6	7	8	9	10
CSLOT4	EID	IDS1	IDS2	IDS3	IDS4	RHO	B	M	

Example:

CSLOT4	101	1	3	2	4		6.2+4	3	
--------	-----	---	---	---	---	--	-------	---	--

Descriptor	Meaning
EID	Element identification number. ( $0 < \text{Integer} < 100,000,000$ )
IDS <sub>i</sub>	Identification number of connected GRIDS points. ( $\text{Integer} > 0$ )
RHO	Fluid density in mass units. ( $\text{Real} > 0.0$ ; Default is the value of RHOD on the AXSLOT entry.)
B	Fluid bulk modulus. ( $\text{Real} \geq 0.0$ ; Default is the value of BD on the AXSLOT entry.)
M	Number of slots in circumferential direction. ( $\text{Integer} \geq 0$ ; Default is the value of MD on the AXSLOT entry.)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. This element identification number (EID) must be unique with respect to all other fluid or structural elements.
3. If RHO, B, or M are blank, then the RHOD, BD, or MD fields on the AXSLOT entry must be specified.
4. This element generates four plot elements connecting points IDS1 to IDS2, IDS2 to IDS3, IDS3 to IDS4, and IDS4 to IDS1.
5. If  $B = 0.0$ , then the slot is considered to be an incompressible fluid.
6. If  $M = 0$ , then no matrices for CSLOT4 elements are generated.

## CSPR

## Springs With Offsets

Springs for use in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSPR	EID	PID	G1	G2					

Example:

CSPR	1	22	456	457					
------	---	----	-----	-----	--	--	--	--	--

Describer	Meaning	Type	Default
EID	Element ID. A unique number has to be used.	I > 0	Required
PID	Property ID of PSPRMAT entry	I > 0	Required
G1	Gridpoint 1	I > 0	Required
G2	Gridpoint 2.	I > 0	Required

## CSSCHD

## Aerodynamic Control Surface Schedule Input

Defines a scheduled control surface deflection as a function of Mach number and angle of attack.

Format:

1	2	3	4	5	6	7	8	9	10
CSSCHD	SID	AESID	LALPHA	LMACH	LSCHD				

Example:

CSSCHD	5	50	12	15	25				
--------	---	----	----	----	----	--	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
AESID	ID of an AESURF Bulk Data entry to which the schedule is being attached.
LALPHA	ID of an AEFACCT Bulk Data entry containing a list of angles of attack (in radians) at which schedule information is provided. (Integer > 0; Default = no angle information provided).
LMACH	ID of an AEFACCT Bulk Data entry containing a list of Mach numbers at which schedule information is provided. (Integer > 0; Default = no Mach information provided)
LSCHD	ID of an AEFACCT Bulk Data entry which contains the scheduling information. See Remarks 4. and 5. (Integer > 0; no Default)

Remarks:

- Control system schedules must be selected with the Case Control command CSSCHD = SID.
- The AESID cannot appear on an AELINK or TRIM Bulk Data entry for the same subcase.
- The control surface deflection is computed using a linear interpolation for the Mach number provided on the associated TRIM entry and the angle of attack derived as part of the trim calculation.
- The LSCHD data are provided as a list of deflections (in radians) as a function of Mach numbers and angles of attack. If there are NMACH Mach numbers and NALPHA angles of attack, the first NALPHA deflections are for the first Mach number, the next NALPHA are for the second Mach number, and so on, until the last NALPHA deflections are for the final Mach number.
- If LALPHA is blank, LSCHD contains NMACH deflections to define the Mach schedule. If LMACH is blank, LSCHD contains NALPHA deflections to define the angle of attack schedule.
- LALPHA and LMACH cannot be simultaneously blank. If LALPHA or LMACH are not blank, at least two values of angle of attack or Mach number must be defined in order to perform interpolation.
- If the Mach number or angle of attack is outside the range specified by the tabulated values, the value at the table end is used. That is, data are not extrapolated.

## CSSHL

## Solid Shell Element Connection

Defines a connection for a Solid Shell with 6 or 8 grid points in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHL	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8							

Example:

CSSHL	44	11	1	2	3	4	5	6	quad
	7	8							

CSSHL	51	22	11	12	13		21	22	tria
	23								

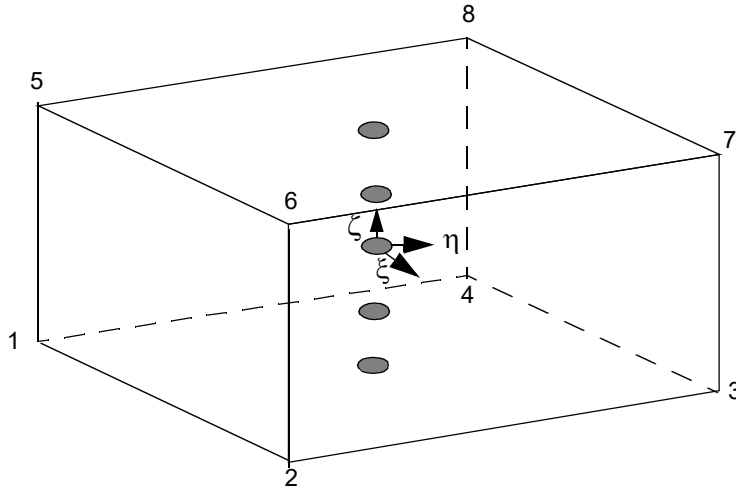
CSSHL	51	22	11	12	13	13	21	22	tria
	23	23							

(Note: the 2nd and 3rd examples are equivalent to each other.)

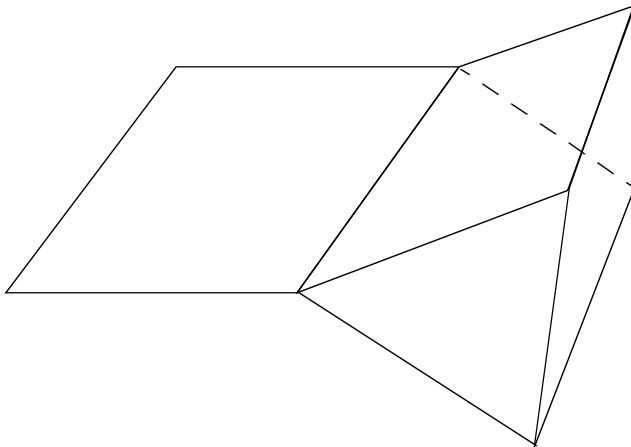
Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000; Required)
PID	Property identification of a PSSHL entry. (Integer > 0; Required)
Gi	Grid point identification number of connection points. (Integer U or blank, for quad shapes all 8 values are required, for triangle shapes only G4 and G8 may be left blank in which case G4=G3 and G8=G7.)

Remarks:

1. This element can degenerate to a triangle either by leaving G4 and G8 blank or by entering G4=G3 and G8=G7 (see 2<sup>nd</sup> and 3<sup>rd</sup> examples).
2. This element is usually only used when contact on each of the shell is anticipated.
3. Mid-side nodes are not available for this element.
4. Grid point ordering is shown in the following figure.



5. The stiffness of this element is formed using one integration point in the element plane and a user defined number through the element thickness. In this way the element can capture accurate material plasticity under bending load. An additional variationally consistent stiffness term is included to eliminate the hourglass modes that are normally associated with reduced integration.
6. The number of integration points through the thickness is given by PARAM,MARCSLHT
7. This element may be collapsed to a triangular solid shell to attach to a standard shell such as CQUAD4 as follows:



8. This element is not currently available with Total Lagrange, finite strain plasticity or hyperelastic materials.

## CSSLH

## CHEXA to Solid Shell Element Connection

Defines conversion of CHEXA elements to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSLH	EID1	PID	EID2	EID3	EID4	EID5	EID6		

Example:

CSSLH	44	11	54	200	250				
-------	----	----	----	-----	-----	--	--	--	--

Describer	Meaning
EID <sub>i</sub>	CHEXA element identification numbers. See Remark 1. (0 < Integer < 100,000,000; no Default; EID1 is required)
PID	Property identification of a PSSHL entry to be used with all the elements with EID <sub>i</sub> . (Integer > 0; Required)

1. All CHEXA elements in the range EID1 to EID2 will be converted to solid shells. The original CHEXA elements will be deleted and the solid shell elements will have the same element ID's as the original CHEXA elements.
2. All CHEXA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSHL for additional items.
6. The PSSHL entries associated with CSSLH are not generated automatically. They must be input using a GUI or with a text editor.

## CSSHLM

## CHEXA or CPENTA to Solid Shell Material Conversion

Defines conversion of CHEXA or CPENTA elements described by material ID to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHLM	MID1		MID2	MID3	MID4	MID5	MID6	MID7	
	MID8	MID9							

Example:

CSSHLM	100		200	700	90250				
--------	-----	--	-----	-----	-------	--	--	--	--

Describer	Meaning
MIDi	Material identification numbers. See Remark 1. (Integer > 0; no Default; MID1 is required)

Remarks:

1. All CHEXA and CPENTA elements with the MIDi specified will be converted to solid shells. The original CHEXA and CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original elements.
2. All PSOLID entries which reference all MIDi values will automatically be converted to PSSHL entries retaining the MID and CORDM values. The PSSHL property ID's will be the same as the PSOLID ID's plus IPOFF. The value of IPOFF should not be zero since the original PSOLID entries are not deleted and all property ID's should normally be unique.
3. Mid-side nodes are not allowed.
4. Elements for this entry are mapped to Marc element type 185.
5. Please see the remarks for CSSHL for additional items.
6. If this entry is used, CSSHLH and CSSHLP entries should not be used.
7. See parameters MCSSHLCK and MCSSHORR for additional options for this entry.

## CSSHLP

## CPENTA to Solid Shell Element Connection

Defines conversion of CPENTA elements to Solid Shell elements in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
CSSHLP	EID1	PID	EID2	EID3	EID4	EID5	EID6		

Example:

CSSHLP	44	11	54	200	250				
--------	----	----	----	-----	-----	--	--	--	--

Describer	Meaning
EID <sub>i</sub>	CPENTA element identification numbers. See Remark 1. (0 < Integer < 100,000,000; no Default; EID1 is required)
PID	Property identification of a PSSHL entry to be used with all the elements with EID <sub>i</sub> . (Integer > 0; Required)

Remarks:

- All CPENTA elements in the range EID1 to EID2 will be converted to solid shells. The original CPENTA elements will be deleted and the solid shell elements will have the same element ID's as the original CPENTA elements.
- All CPENTA elements in the range EID3 to EID4 as well as EID5 to EID6 will be converted to solid shells if these fields are entered.
- The solid shell elements will be collapsed Marc element 185 type elements with the 3<sup>rd</sup> and 4<sup>th</sup> grids set to G3 of the CPENTA and the 7<sup>th</sup> and 8<sup>th</sup> grids set to the 6<sup>th</sup> grid of the CPENTA.
- Mid-side nodes are not allowed.
- Please see the remarks for CSSHL for additional items.
- The PSSHL entries associated with CSSHLH are not generated automatically. They must be input using a GUI or with a text editor.



**CSUPER** Secondary Superelement Connection

Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source. These are all known as secondary superelements.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
CSUPER	SSID	PSID	GP1	GP2	GP3	GP4	GP5	GP6	
	GP7	GP8	-etc.-						

Example:

CSUPER	120003	21	3	6	4	10			
--------	--------	----	---	---	---	----	--	--	--

Descriptor	Meaning
SSID	Coded identification number for secondary superelement. See Remark 1. (Integer > 0)
PSID	Identification number for referenced primary superelement. See Remark 2. (Integer > 0 or blank)
GPi	Grid or scalar point identification numbers of the exterior points of the secondary superelement. See Remark 3. (Integer > 0)

Remarks:

- The value of SSID is written in the form XXX0000 + n, where n is the referenced secondary superelement identification number and n must be less than 10000 and XXX is a displacement component sign reversal code as follows:

The sign reversal code specifies the displacement component(s) normal to the plane of the mirror through which the reflection is to be made

Blank or 0	no reversal for identical superelement. If PSID is preceded by a minus sign and there is no xxx code on SSID, then a z-reversal mirror is generated.	
1	x-reversal	} Mirror Images
2	y-reversal	
3	z-reversal	
12	x and y-reversal	} Identical Images
23	y and z-reversal	
31	z and x-reversal	
123	x, y and z-reversal	} Mirror Images



2. If  $PSID = 0$  or blank, the superelement boundary matrices are obtained from an external source (such as a database or external file). See also PARAM, [EXTDRUNT](#).

If  $PSID \neq 0$ , the secondary superelement is identical to, or is a mirror image of, a primary superelement.

3. For identical or mirror image superelements, the grid point IDs,  $GPI$ , may appear in any order. However, if they are not in the same order as the external GRIDs of the primary superelement, then the SEQSEP entry is also required. In case of external superelements, the GRID IDs must be in the order that the terms in the associated matrices occur in.
4. Image superelements and their primaries must be congruent. The identical or mirror image superelement must have the same number of exterior grid points as its primary superelement. The exterior grid points of the image superelement must have the same relative location to each other as do the corresponding points of the primary superelement. The global coordinate directions of each exterior grid point of the image superelement must have the same relative alignment as those of the corresponding grid points of the primary superelement. If congruency is not satisfied because of round-off, then the tolerance may be adjusted with PARAM,CONFAC or DIAG 37.
5. For superelements from an external source, please refer to PARAMS [EXTDR](#), [EXTDRUNT](#) and [EXTUNIT](#).

## CSUPEXT      Superelement Exterior Point Definition

Assigns exterior points to a superelement.

Format:

1	2	3	4	5	6	7	8	9	10
CSUPEXT	SEID	GP1	GP2	GP3	GP4	GP5	GP6	GP7	

Example:

CSUPEXT	2	147	562	937					
---------	---	-----	-----	-----	--	--	--	--	--

Alternate Format and Example:

CSUPEXT	SEID	GP1	“THRU”	GP2					
---------	------	-----	--------	-----	--	--	--	--	--

CSUPEXT	5	12006	THRU	12050					
---------	---	-------	------	-------	--	--	--	--	--

Describer	Meaning
SEID	Identification number of a primary superelement. (Integer > 0)
GPi	Grid or scalar point identification number in the downstream superelement or residual structure. (Integer > 0 or “THRU”; for “THRU” option, GP1 < GP2)

Remarks:

1. Grid or scalar points are connected (that is, are exterior) to a superelement only if they are connected by structural, rigid, or plot elements. MPC entries are not regarded as elements. This entry is a means of providing connectivity for this purpose.
2. Open sets are allowed with the “THRU” option.
3. Scalar points may be interior to the residual structure (SEID = 0) only.
4. This entry may be applied only to the primary superelements. The CSUPER entry is used for secondary superelements (identical image, mirror image, and external superelements).

## CTETRA Four-Sided Solid Element Connection

Defines the connections of the four-sided solid element with four to ten grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTETRA	EID	PID	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	G10					

Example:

CTETRA	112	2	3	15	14	4	103	115	
	5	16	8	27					

Describer	Meaning	Type	Default
EID	Element identification number.	0 < Integer < 100,000,000	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Identification numbers of connected grid points.	Integer ≥ 0 or blank	Required

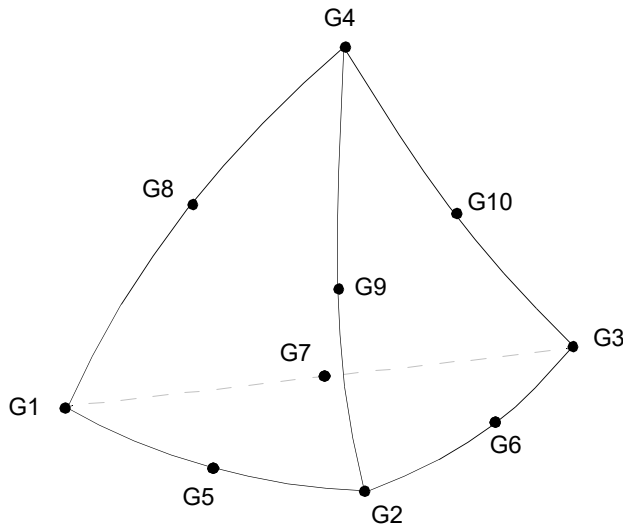


Figure 9-69 CTETRA Element Connection

Remarks:

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. The topology of the diagram must be preserved, i.e., G1, G2, G3 define a triangular face; G1, G8, and G4 are on the same edge, etc.
3. The edge points, G5 to G10, are optional. For Nastran conventional elements, any or all of them may be deleted. If the ID of any edge connection point is left blank or set to zero, the equations of the element are adjusted to give correct results for the reduced number of connections. Corner grid points cannot be deleted. The element is an isoparametric element in all cases.  
Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
4. Components of stress are output in the material coordinate system, except hyperelastic elements which outputs stress in the basic coordinate system.
5. For nonhyperelastic elements, the element coordinate system is derived from the three vectors R, S, and T, which join the midpoints of opposite edges.

R vector joins midpoints of edges G1-G2 and G3-G4.

S vector joins midpoints of edges G1-G3 and G2-G4.

T vector joins midpoints of edges G1-G4 and G2-G3.

The origin of the coordinate system is located at G1. The element coordinate system is chosen as close as possible to the R, S, and T vectors and points in the same general direction. (Mathematically speaking, the coordinate system is computed in such a way that, if the R, S, and T vectors are described in the element coordinate system, a 3x3 positive definite symmetric matrix would be produced.)

Solid elements have both a material and an element coordinate system. Both systems are defined for the initial geometry, and for geometric nonlinear analysis they will rotate with the element. The material coordinate system is used to input anisotropic material properties and for stress output. The material coordinate system is defined by the CORDM field of the PSOLID entry. The element coordinate system is used for element stiffness integration (reduced shear for example) and optionally to define the material coordinate system (only if PSOLID,CORDM=-1).

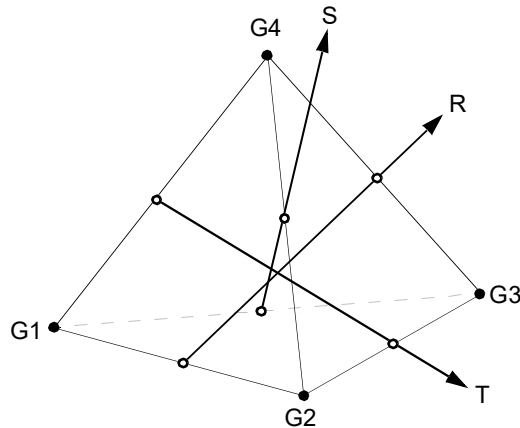


Figure 9-70 CTETRA Element R, S, and T Vectors

6. It is recommended that the edge points be located within the middle third of the edge.
7. For hyperelastic elements, the plot codes are specified under the CTETRAFD element name in [Item Codes](#).
8. By default, all of the six edges of the element are considered straight unless, any of G5 through G10 are specified.
9. The internal coordinate system of the element is used internally and is based on eigenvalue techniques to insure non bias in the element formulation. For stress/strain output this internal coordinate system (CORDM=-1 on PSOLID entry) is hard to visualize. Thus a CORDM=-2 on the PSOLID is available as shown in [Figure 9-71](#).

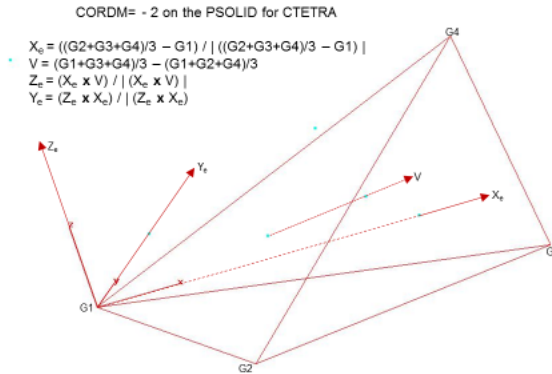


Figure 9-71 PSOLID on CTETRA

## CTRIA3      Triangular Plate Element Connection

Defines an isoparametric membrane-bending or plane strain triangular plate element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA3	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIA3	111	203	31	74	75	3.0	0.98		
			1.77	2.04	2.09				

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0; Default = EID)
Gi	Grid point identification numbers of connection points. (Integers > 0, all unique)
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:  CORD1R,    x -axis of MCID the coordinate is projected onto shell surface and the CORD2R    material angle is measured from the G1-G2 line to the to the projected x-axis  CORD1C,    r-axis of MCID the coordinate is projected onto shell surface through CORD2C    the element center and the material angle is measured from the G1-G2 CORD1S,    line to the to the projected r-axis CORD2S
	Use DIAG 38 to print the computed THETA values. MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.)

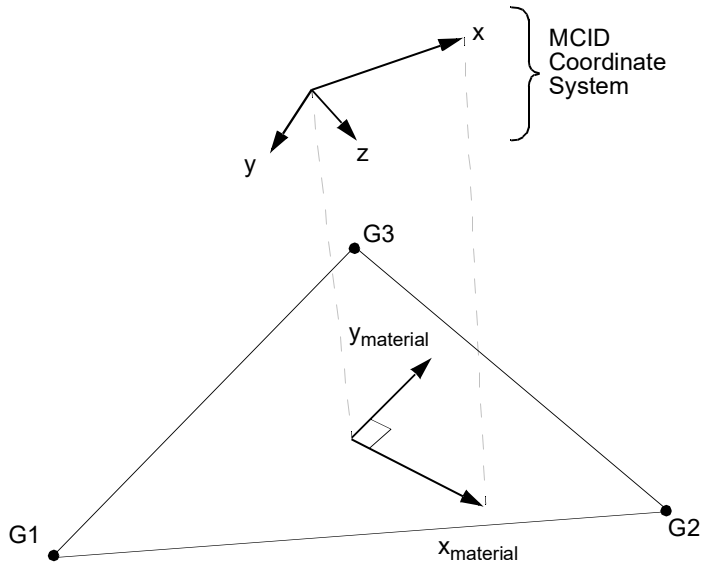


Figure 9-72 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 3. ZOFFS is ignored for hyperelastic elements. (Real)
TFLAG	An integer flag, signifying the meaning of the $T_i$ values. (Integer 0, 1, or blank)
$T_i$	Membrane thickness of element at grid points G1 through G3. If “TFLAG” zero or blank, then $T_i$ are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the $T_i$ are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank; not all zero. Default = 1.0) $T_i$ are ignored for hyperelastic elements.

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.



Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

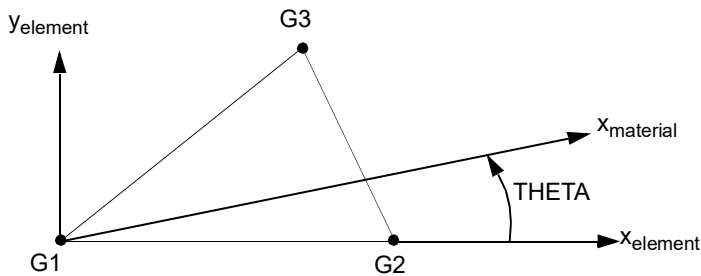


Figure 9-73 CTRIA3 Element Geometry and Coordinate Systems

4. The reference coordinate system for the output of stress, strain and element force depends on the element type.
  - For CTRIA3 elements, which are not hyperelastic, the reference coordinate system for output is the element coordinate system.
  - For hyperelastic elements the stress and strain are output according to CID on the PLPLANE entry.
5. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes](#).
6. SYSTEM(218), alias T3SKEW, allows the user to control the minimum vertex angle for TRIA3 elements at which USER WARNING MESSAGE 5491 is issued. The default value is 10. degrees.
7. By default, all of the three edges of the element are considered straight.
8. For RC network solver in thermal analysis, the ZOFFS is ignored.
9. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

## CTRIA6 Curved Triangular Shell Element Connection

Defines a curved triangular shell element or plane strain with six grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIA6	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID	ZOFFS	T1	T2	T3	TFLAG			

Example:

CTRIA6	302	3	31	33	71	32	51	52	
	45	.03	.020	.025	.025				

Describer	Meaning
EID	Element Identification number. (0 < Integer < 100,000,000)
PID	Property identification number of PSHELL, PCOMP, PCOMPG or PLPLANE entry. (Integer > 0)
G1, G2, G3	Identification numbers of connected corner grid points. (Unique Integers > 0)
G4, G5, G6	Identification number of connected edge grid points. Optional data for any or all three points for Nastran conventional element only. (Integer > 0 or blank) Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
THETA	Material property orientation angle in degrees. THETA is ignored for hyperelastic elements. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:  CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis  CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line CORD1S, to the to the projected r-axis CORD2S  MCID is ignored for hyperelastic elements. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed)

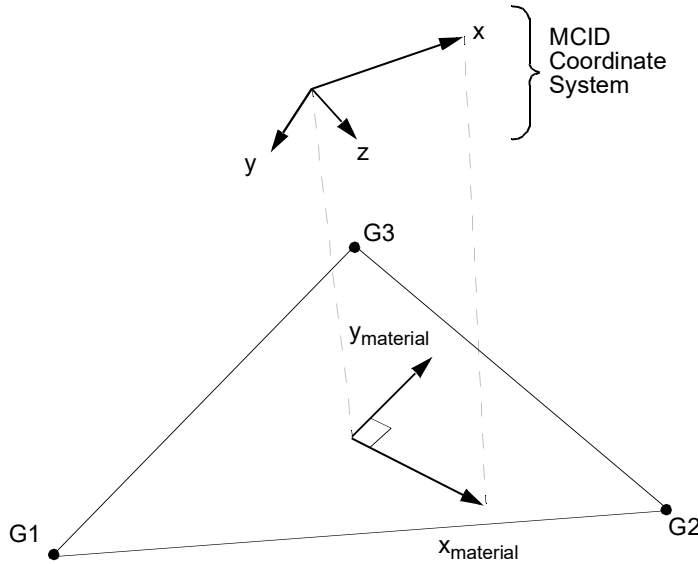


Figure 9-74 MCID Coordinate System Definition

ZOFFS	Offset from the surface of grid points to the element reference plane; see Remark 6. ZOFFS is ignored for hyperelastic elements. (Real)
Ti	Membrane thickness of element at grid points G1 through G4. If “TFLAG” zero or blank, then Ti are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the Ti are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) Ti are ignored for hyperelastic elements.
TFLAG	An integer flag, signifying the meaning of the Ti values. (Integer 0, 1, or blank)

Remarks:

1. Element identification numbers should be unique with respect to all other element IDs.
2. Grid points G1 through G6 must be numbered as shown in [Figure 9-75](#).
3. The orientation of the material property coordinate system is defined locally at each interior integration point by THETA, which is the angle between  $x_{material}$  and the line of constant  $\eta$ .
4. T1, T2, and T3 are optional. If they are not supplied and no TFLAG, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
5. It is recommended that the midside grid points be located within the middle third of the edge.

6. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, it is **highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

7. If all midside grid points are deleted, then the element will be excessively stiff and the transverse shear forces will be incorrect. A User Warning Message is printed. A CTRIA3 element entry is recommended instead. If the element is hyperelastic, then the element is processed identically to the hyperelastic CTRIA3 element.
8. For a description of the element coordinate system, see [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) in the *MSC Nastran Reference Guide*. Stresses and strains are output in the local coordinate system identified by  $x_l$  and  $y_l$  in [Figure 9-75](#). For hyperelastic elements, stresses and strains are output in the coordinate system defined by the CID field on the PLPLANE entry.

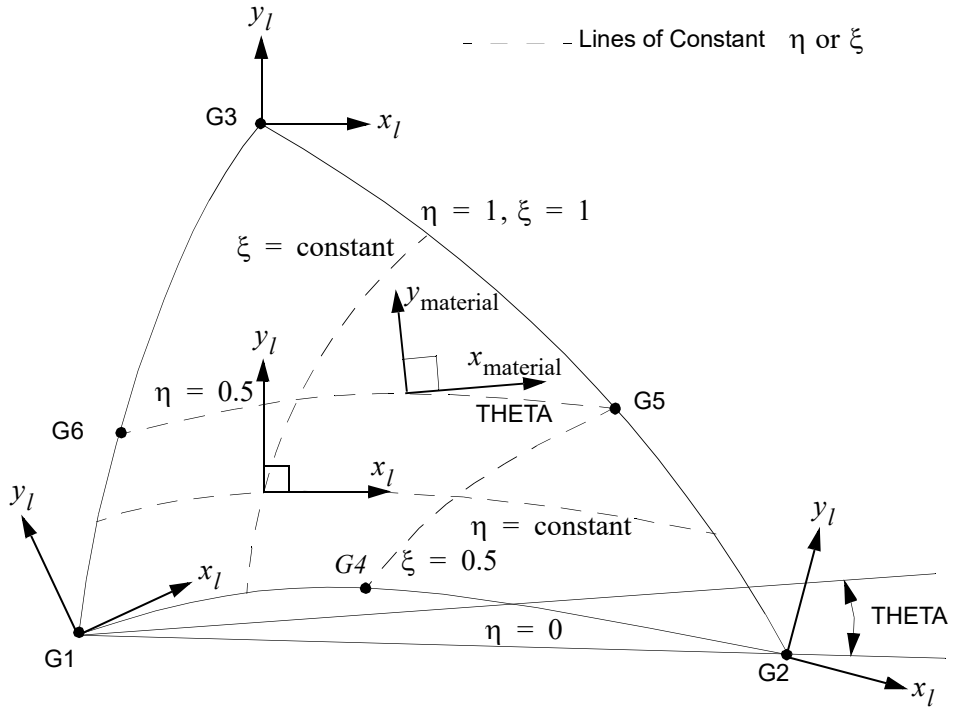


Figure 9-75 CTRIA6 Element Geometry and Coordinate Systems

9. For hyperelastic elements, the plot codes are specified under the CTRIAFD element name in [Item Codes, 1045](#).
10. For RC network solver in thermal analysis, the ZOFFS is ignored.
11. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.

## CTRIAR Triangular Plate Element Connection

Defines an isoparametric membrane-bending triangular plate element. This element has a normal rotational (drilling) degrees-of-freedom. It is a companion to the CQUADR element.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAR	EID	PID	G1	G2	G3	THETA or MCID	ZOFFS		
		TFLAG	T1	T2	T3				

Example:

CTRIAR	111	203	31	74	75	3.0			
			1.77	2.04	2.09				

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSHELL, PCOMP or PCOMPG entry. (Integer > 0; Default = EID)
G1, G2, G3	Grid point identification numbers of connection points. (Integers > 0; all unique)
THETA	Material property orientation angle in degrees. (Real; Default = 0.0)
MCID	Material coordinate system identification number. The x-axis of the material coordinate system is determined by projecting the T1-axis of the MCID coordinate system onto the surface of the shell element as follows:  CORD1R, x -axis of MCID the coordinate is projected onto shell surface and the CORD2R material angle is measured from the G1-G2 line to the to the projected x-axis  CORD1C, r-axis of MCID the coordinate is projected onto shell surface through the CORD2C element center and the material angle is measured from the G1-G2 line to CORD1S, the to the projected r-axis CORD2S
	Use DIAG 38 to print the computed THETA values. For SOL 600, only CORD2R is allowed. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed)
ZOFFS	Offset from the surface of grid points to the element reference plane. See Remark 5.

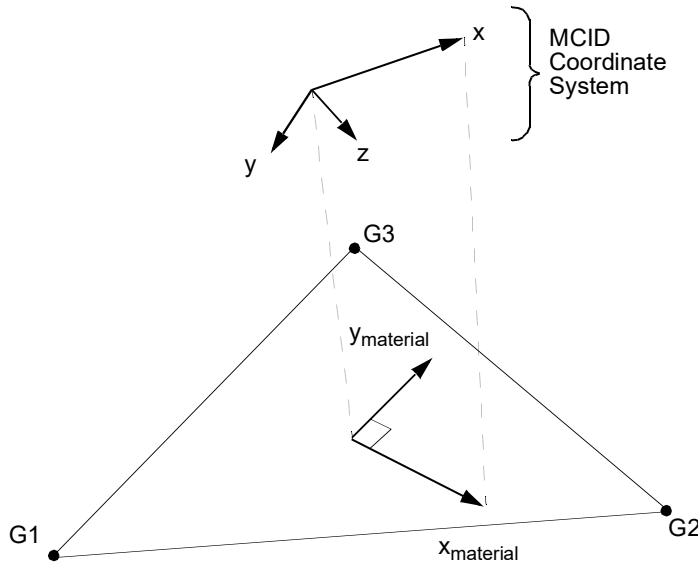


Figure 9-76 MCID Coordinate System Definition

TFLAG	An integer flag, signifying the meaning of the $T_i$ values. (Integer 0, 1, or blank)
$T_i$	Membrane thickness of element at grid points G1 through G4. If “TFLAG” zero or blank, then $T_i$ are actual user specified thickness. (Real $\geq 0.0$ or blank, not all zero. See Remark 4. for default.) If “TFLAG” one, then the $T_i$ are fraction relative to the T value of the PSHELL. (Real $> 0.0$ or blank, not all zero. Default = 1.0) $T_i$ are ignored for hyperelastic elements.

#### Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. The continuation is optional. If it is not supplied, then T1 through T3 will be set equal to the value of T on the PSHELL entry.
3. Stresses are output in the element coordinate system at the centroid and grid points G1 through G3.
4. Inaccurate results will be obtained if interior grids have the rotation normal (drilling) to the element constrained. At the boundary of a model, the drilling degrees-of-freedom must be constrained if the user wants a fixed boundary. Also, for this element it is critical that consistent membrane (in plane) edge loads be applied. Reference the [PLOAD4](#) entry (SORL option) and the [Consistent Surface and Edge Loads](#) in the *MSC Nastran Reference Guide* for additional information.

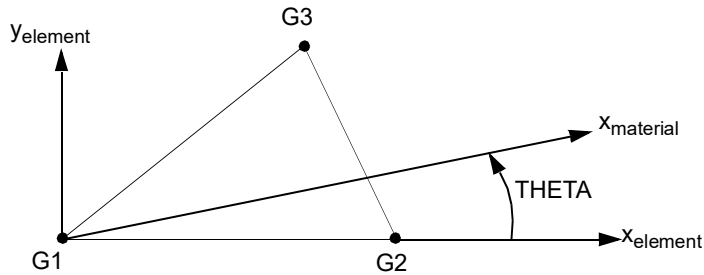


Figure 9-77 CTRIAR Element Geometry and Coordinate Systems

5. Elements may be offset from the connection points by means of the ZOFFS field. Other data, such as material matrices and stress fiber locations, are given relative to the reference plane. A positive value of ZOFFS implies that the element reference plane is offset a distance of ZOFFS along the positive Z-axis of the element coordinate system. If the ZOFFS field is used, then both the MID1 and MID2 fields must be specified on the PSHELL entry referenced by PID.

Two methods are available for the computation of offsets: original and enhanced. The default method is the original method. The enhanced method is requested by the Bulk Data entry MDLPRM,OFFDEF,LROFF. For options of offsets, please refer to the Bulk Data entry MDLPRM,OFFDEF,option.

For solution sequences that require differential stiffness such as SOL103 (with preloading), SOL105, and SOL400, **it is highly recommended** to use MDLPRM, OFFDEF, LROFF. In SOL103, if the case control ADAMSMNF FLEXBODY=YES or AVLEXB EXBBODY=YES is present, then MDLPRM, OFFDEF, LROFF must be replaced by **MDLPRM, OFFDEF, NOMASS**.

For SOLs 106, 129, 153, and 159 the differential stiffness for offset vectors will give incorrect results with PARAM, LGDISP, 1. In addition in SOLs 106 and 129 offset vectors will produce incorrect results with thermal loading.

6. In SOL 600, when PCOMP is used, one must define the material property with orientation, MCID>0.



## CTRIAX Axisymmetric Triangular Element (Fully Nonlinear or Linear Harmonic)

Defines an axisymmetric triangular element with up to six grid points for use in fully nonlinear (i.e., large strain and large rotations) hyperelastic analysis or a linear harmonic or rotordynamic analysis. The element has between three and six grid points.

### Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX	EID	PID	G1	G2	G3	G4	G5	G6	
	THETA or MCID								

### Example:

CTRIAX	111	203	31	74	75				
--------	-----	-----	----	----	----	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000) See Remark 1.
PID	Property identification number of a PLPLANE or PAXSYMH entry. (Integer > 0). See Remark 2.
G1, G2, G3	Identification numbers of connected corner grid points. Required data for all three grid points. (Unique Integers > 0). See Remark 3., 6., 7.
G4, G5, G6	Identification numbers of connected edge grid points. Optional data for any or all four grid points for Nastran conventional element only. (Integer > 0 or blank). See Remark 3., 4., 6., 7. Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified
THETA	Material property orientation angle in degrees. THETA is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Real; Default = 0.0)
MCID	Material coordinate system identification number, MCID is only applicable if the PLPLANE entry has an associated PSHLN2 entry which is honored only in SOL 400. The x-axis of the material coordinate system is determined by projecting the x-axis of the MCID coordinate system onto the surface of the element. The resulting angle is measured relative to the R axis of the element. For PSHLN2 BEHi=AXSOLID code, THETA is measured relative to the R axis of the element. (Integer ≥ 0; if blank, then THETA = 0.0 is assumed.) THETA is ignored if PID refers to a PAXSYMH entry.

### Remarks:

1. Element identification numbers must be unique with respect to all other element IDs of any kind.

2. If PID refers to a PLPLANE entry, CTRIAX defines an element for use in fully nonlinear analysis. If PID refers to a PAXSYMH entry, CTRIAX defines a linear harmonic element for use in rotordynamic or harmonic analysis.
3. Gi must be numbered as shown in [Figure 9-78](#).
4. It is recommended that the edge points be located within the middle third of the edge.
5. The plot codes are specified under the CTRIAXFD element name in [Item Codes](#).
6. The grid points of the axisymmetric element must lie on the x-y plane of the basic coordinate system. Stress and strain are output in the basic coordinate system.
7. A concentrated load (e.g., FORCE entry) at Gi is divided by the radius to Gi and then applied as a force per unit circumferential length. For example, in order to apply a load of 100 N/m on the circumference at G1, which is located at a radius of 0.5 m, then the magnitude specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot (0.5 \text{ m}) = 50 \text{ N}$$

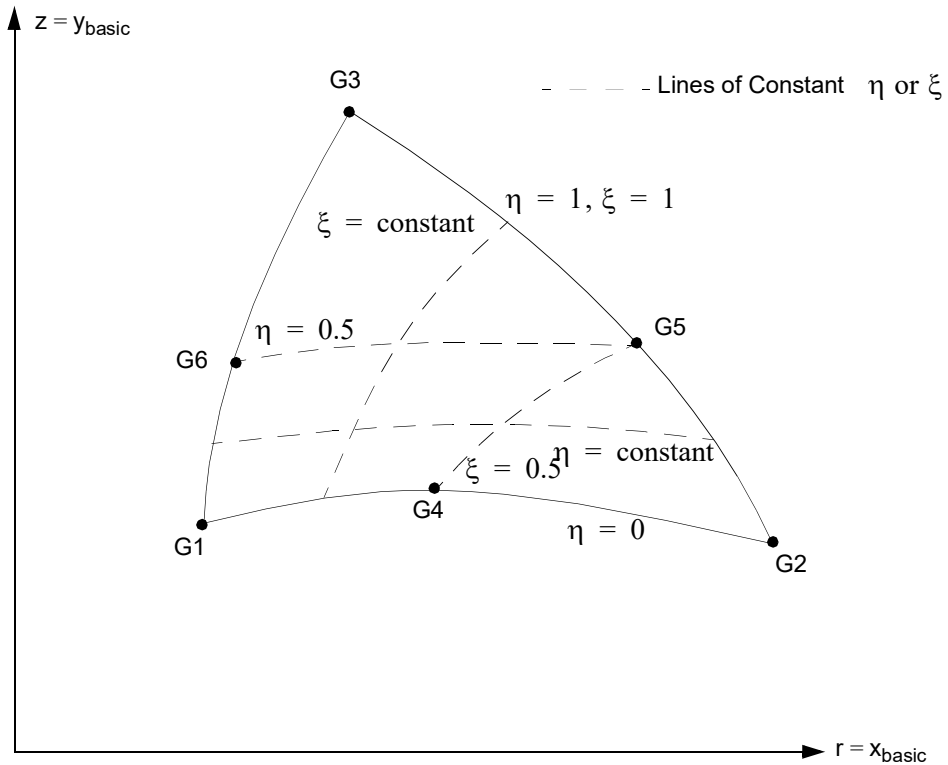


Figure 9-78 CTRIAX Element Coordinate System

## CTRIAX6

## Axisymmetric Triangular Element Connection

Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.

Format:

1	2	3	4	5	6	7	8	9	10
CTRIAX6	EID	MID	G1	G2	G3	G4	G5	G6	
	TH								

Example:

CTRIAX6	22	999	10	11	12	21	22	32	
	9.0								

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
MID	Material identification number. (Integer > 0)
Gi	Grid point identification numbers of connected points (unique Integers > 0, or blank for deleted nodes.)
TH	Material property orientation angle in degrees. (Real; Default = 0.0)

Remarks:

1. The grid points must lie in the x-z plane of the basic coordinate system, with  $x = r \geq 0$ . The grid points must be listed consecutively beginning at a vertex and proceeding around the perimeter in either direction. Corner grid points G1, G3, and G5 must be present. Any or all edge grid points G2, G4, or G6 may be deleted for Nastran conventional element. Note that the alternate corner-edge grid point pattern is different from the convention used on the CTRIA6 element. Note that for advanced nonlinear elements, partial specification of mid-points is not allowed. i.e. either none of the mid-points should be specified or all of the mid points should be specified.
2. For structural problems, the MID may refer to a MAT1 or MAT3 entry.
3. The continuation is optional.
4. Material properties (if defined on a MAT3 entry) and stresses are given in the  $(r_m, z_m)$  coordinate system shown in [Figure 9-80](#).
5. A concentrated load (e.g., FORCE entry) at Gi is multiplied by  $2\pi$  times the radius to Gi and then applied as a force per unit circumferential length. (which is located at a radius of 0.5 m), the magnitude of the load specified on the static load entry must result in:

$$(100 \text{ N/m}) \cdot 2\pi \cdot (0.5 \text{ m}) = 314.159 \text{ N}$$

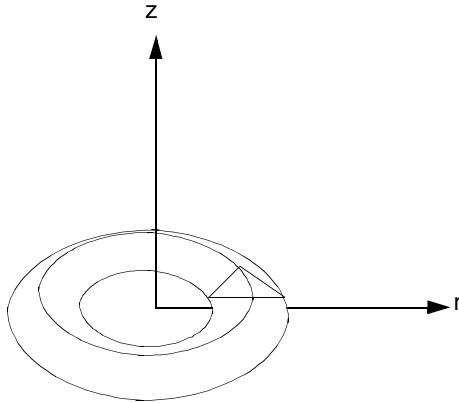


Figure 9-79 CTRIAX6 Element Idealization

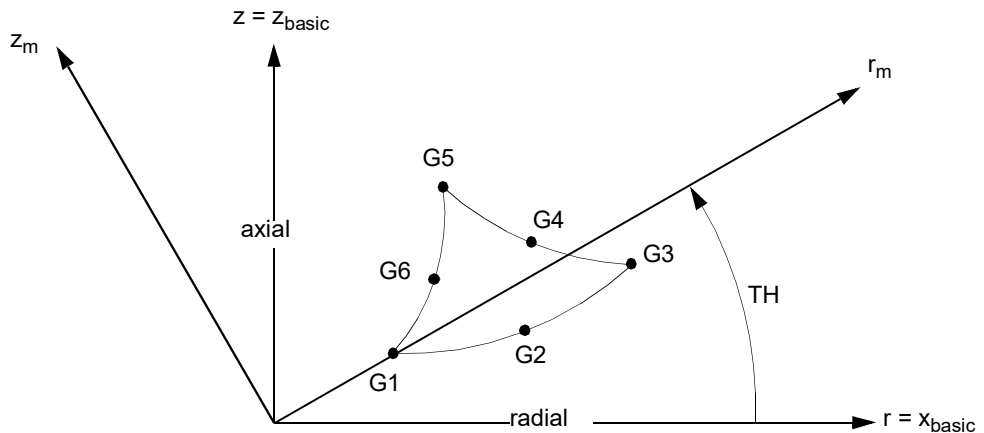


Figure 9-80 CTRIAX6 Element Geometry and Coordinate Systems

6. For thermal problems, the MID may refer to a MAT4 or MAT5 entry. In order to model the convection or radiation along the edges the CHBDYG with TYPE=REV must be used.
7. To model axi-symmetric view factor computations using the CTRIAX6 elements, the normal direction for the CTRIAX6 element must point in the negative Y direction. The reason for this is to set up PATRAN for the correct nodal specification on the CHBDYG with REV option

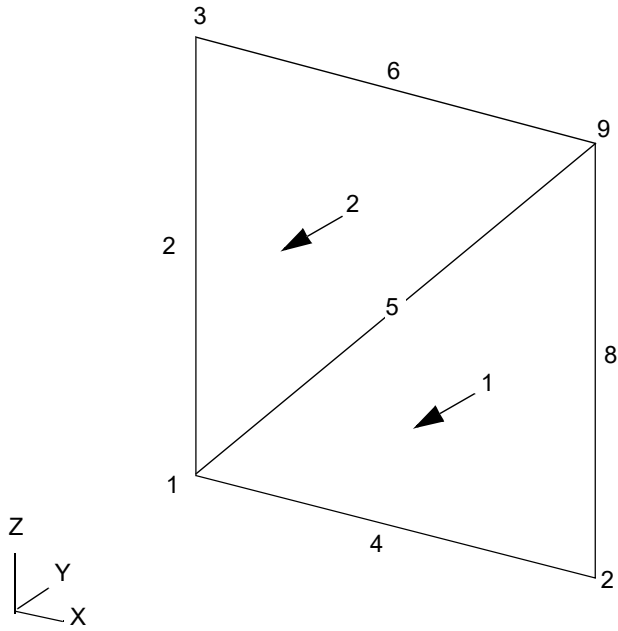


Figure 9-81 Normal Vectors That Point in a Negative Y Direction.

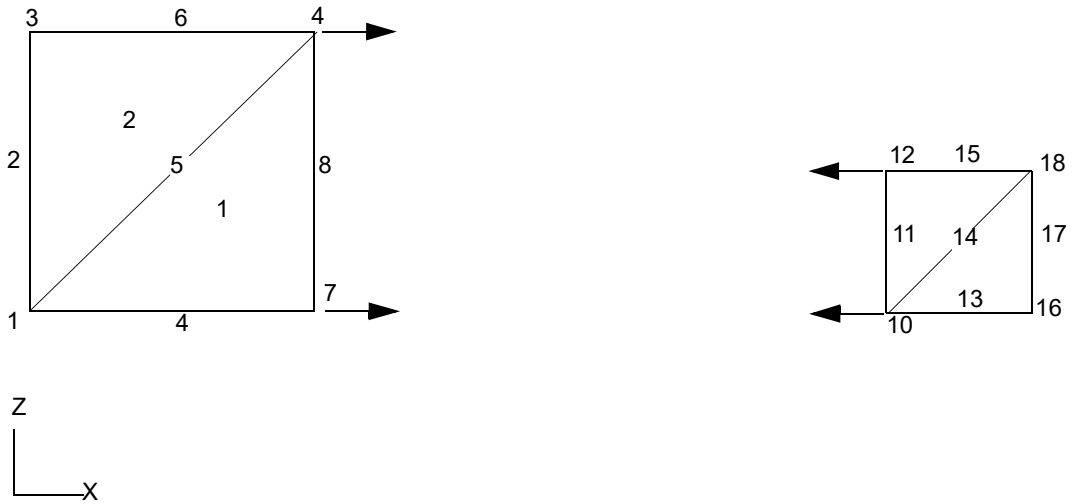


Figure 9-82 View Factor Boundary Conditions.

```
$ CHBDYG Surface Elements
CHBDYG 100001          REV    1          1
        7            9          8
CHBDYG 100002          REV    2          2
        12           10         11
```

Please note that on the CHBDYG, REV option, the nodal connectivity specification is in the clockwise fashion. In this case the CHBDYG with element ID of 100001 is defined by grids 7,9,8 which means the normal vector of the edge is pointing towards right. On the other hand the edges define by grid 12,10,11 means the normal vector of the edge is pointing towards left. If you have not reverse the element direction to a negative Y direction on the CTRIAX6 element, the view factor computation will result a net view factor of zero.

## CTUBE Tube Element Connection

Defines a tension-compression-torsion tube element.

Format:

1	2	3	4	5	6	7	8	9	10
CTUBE	EID	PID	G1	G2					

Example:

CTUBE	12	13	21	23					
-------	----	----	----	----	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PTUBE entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one tube element may be defined on a single entry.

## CVISC Viscous Damper Connection

Defines a viscous damper element.

Format:

1	2	3	4	5	6	7	8	9	10
CVISC	EID	PID	G1	G2					

Example:

CVISC	21	6327	29	31					
-------	----	------	----	----	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PVISC entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. Only one viscous damper element may be defined on a single entry.
3. Grids G1 and G2 must not be coincident. If coincident grids are required, use either the CDAMP or CBUSH entry.



## CWELD

### Weld or Fastener Element Connection

Defines a weld or fastener connecting two surface patches or points. Large displacement and large rotational effects are supported when using SOL 600 and SOL 400 only.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
CWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB							
	XS	YS	ZS						

Example:

CWELD	101	8	203	PARTPAT					
	21	33							

Alternate formats and examples:

Format ELPAT:

CWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB							
	XS	YS	ZS						

Example:

CWELD	103	5	403	ELPAT					
	309	511							

Format ELEMID:

CWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB							

Example:

CWELD	103	5	403	ELEMID					
	309	511							

Format GRIDID:

CWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

CWELD	7	29	233	GRIDID			QT		
	15	28	31	35	46	51	55	60	
	3	5	8						

Format ALIGN:

CWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	
-------	------	------	--	---------	----	----	--	------	--

Example:

CWELD	7	29		ALIGN	103	259			
-------	---	----	--	-------	-----	-----	--	--	--

Describer	Meaning	Type	Default
EWID	CWELD element identification number. See Remark 1.	0 < Integer < 100,000,000	Required
PWID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. See Remarks 2. and 3.	Integer > 0 or blank	

Describer	Meaning	Type	Default
“PARTPAT”	Character string indicating the type of connection. The format of the subsequent entries depends on the type. “PARTPAT”, for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA and PIDB, respectively. The “PARTPAT” format connects up to 3x3 elements per patch. See Remark 4.	Character	Required
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. See Remark 5.	Integer > 0 or blank	blank
MCID	Specifies the element stiffness coordinate system. See Remark 16.	Integer $\geq$ -1 or blank	Default = -1
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A and B respectively.	Integer > 0	Required for “PARTPAT”
XS, YS, ZS	Coordinates of spot weld location in basic. See Remark 2.	Real	Required if GS and GA are not defined.

For the alternate formats, the describer meaning are described below:

Describer	Meaning	Type	Default
“ELPAT”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The “ELPAT” format connects up to 3x3 elements per patch. See Remark 6.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for “ELPAT”
“ELEMID”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA and SHIDB, respectively. The “ELEMID” format connects one shell element per patch. See Remark 7.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A and B, respectively.	Integer > 0	Required for “ELEMID”

Describer	Meaning	Type	Default
“GRIDID”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two sequences of grid point identification numbers, $GA_i$ and $GB_i$ , respectively. The “GRIDID” format connects the surface of any element. See Remark 8.	Character	Required
SPTYP	Character string indicating types of surface patches A and B. SPTYP = “QQ”, “TT”, “QT”, “TQ”, “Q” or “T”. See Remark 9.	Character	Required for “GRIDID”
$GA_i$	Grid identification numbers of surface patch A. $GA_1$ to $GA_3$ are required. See Remark 10.	Integer > 0	Required for “GRIDID”
$GB_i$	Grid identification numbers of surface patch B. See Remark 10.	Integer > 0	
“ALIGN”	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points $GA$ and $GB$ , respectively. See Remark 11.	Character	Required
$GA$ , $GB$	Vertex grid identification number of shell A and B, respectively.	Integer > 0	Required for “ALIGN”

#### Remarks:

1. CWELD defines a flexible connection between two surface patches, between a point and a surface patch, or between two shell vertex grid points. See Figure 9-83 through Figure 9-87.
2. Grid point  $GS$  defines the approximate location of the connector in space.  $GS$  is projected on surface patch A and on surface patch B. The resulting piercing points  $GA$  and  $GB$  define the axis of the connector.  $GS$  must have a normal projection on surface patch A and B.  $GS$  does not have to lie on the surface patches.  $GS$  is ignored for format “ALIGN”.  $GA$  is used instead of  $GS$  if  $GS$  is not specified. For the formats “ELPAT” and “PARTPAT,” if  $GS$  and  $GA$  are not specified, then  $XS$ ,  $YS$ , and  $ZS$  will be assumed.
3. The connectivity between grid points on surface patch A and grid points on surface patch B is generated depending on the location of  $GS$  and the cross sectional area of the connector. Diagnostic print outs, checkout runs and non default settings of search and projection parameters are requested on the SWLDPRM Bulk Data entry. It is recommended to start with the default settings.
4. The format “PARTPAT” defines a connection of two shell element patches A and B with PSHELL property identification numbers  $PIDA$  and  $PIDB$ , respectively. The two property identification numbers must be different, see Figure 9-83. The number of connected elements depends on the location of the piercing points  $GA$ ,  $GB$  and the size of the diameter  $D$ . The diameter  $D$  is defined on the PWELD property entry. For this option, shell element patches A and B are allowed to share a common grid.

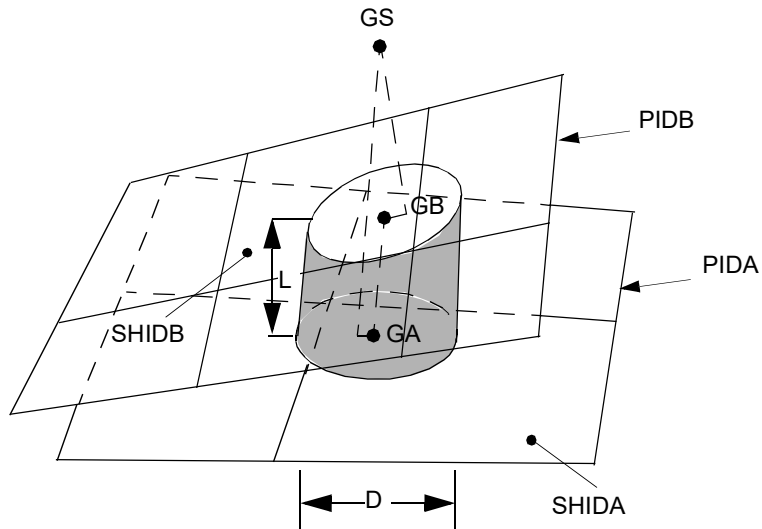


Figure 9-83 Patch to Patch Connection Defined with the Formats PARTPAT or ELPAT

5. The definition of the piercing grid points GA and GB is optional for all formats with the exception of the format “ALIGN”. If GA and GB are given, GS is ignored. If GA and GB are not specified, they are generated from the normal projection of GS on surface patches A and B and internal identification numbers are generated for GA and GB starting with 101e+6 by default. The offset number can be changed with PARAM,OSWPPT. If GA and GB are specified, they must lie on or at least have a projection on surface patches A and B, respectively. By default, the locations of user specified GA and GB will not be changed. If the user specifies “SWLDPRM, MOVGAB, 1,” then the locations will be corrected so that they lie on surface patches A and B within machine precision accuracy. The length of the connector is the distance of grid point GA to GB, subject to being adjusted to the effective length defined in PWELD entry.
6. The format “ELPAT” defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-83](#). The connectivity is similar to the format “PARTPAT”. The number of connected elements depends on the location of the piercing points GA, GB and the size of the diameter D. For this option, shell element patches A and B are allowed to share a common grid.
7. The format “ELEMID” defines a connection of two shell element patches A and B with shell element identification numbers SHIDA and SHIDB, see [Figure 9-84](#). The connectivity is restricted to a single element per patch regardless of the location of GA, GB and regardless of the size of the diameter of the connector. In addition, the format “ELEMID” can define a point to patch connection if SHIDB is left blank, see [Figure 9-85](#). Then grid GS is connected to shell SHIDA.

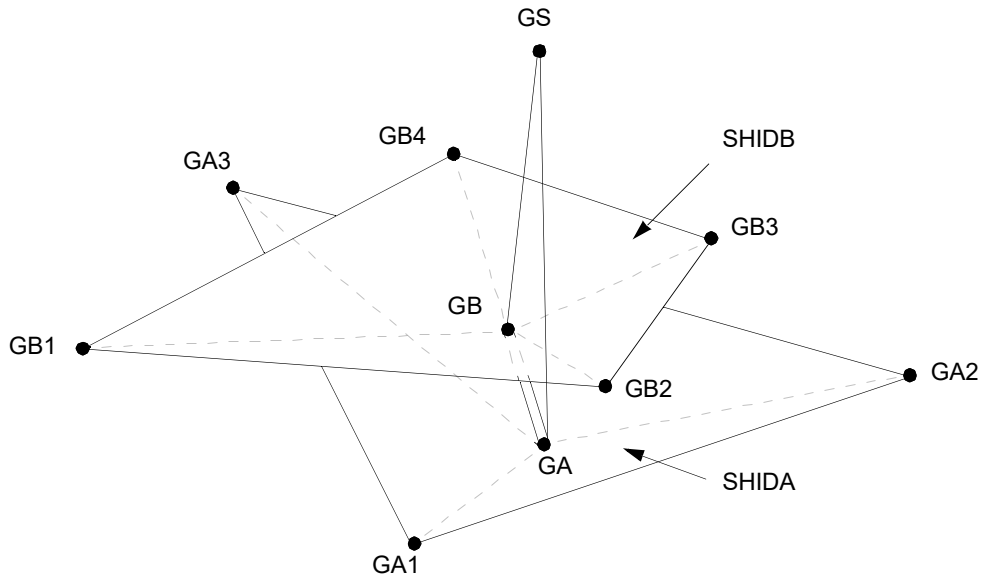


Figure 9-84 Patch to Patch Connection Defined with Format ELEMID or GRIDID

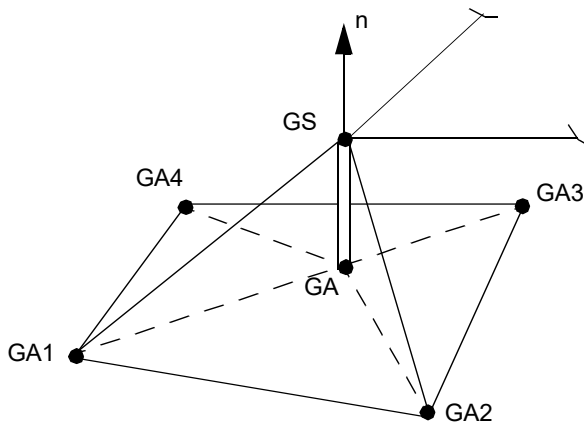


Figure 9-85 Point to Patch Connection Defined with Format ELEMID or GRID.

8. The format “GRIDID” defines a connection of two surface patches A and B with a sequence of grid points  $GA_i$  and  $GB_i$ , see [Figure 9-84](#). In addition, the format “GRIDID” can define a point to patch connection if all  $GB_i$  fields are left blank, see [Figure 9-85](#). Then grid GS is connected to grids  $GA_i$ . The grids  $GA_i$  and  $GB_i$  do not have to belong to shell elements.
9. SPTYP defines the type of surface patches to be connected. SPTYP is required for the format “GRIDID” to identify quadrilateral or triangular patches. The combinations are:

SPTYP	Description
QQ	Connects a quadrilateral surface patch A (Q4 to Q8) with a quadrilateral surface patch B (Q4 to Q8).
QT	Connects a quadrilateral surface patch A (Q4 to Q8) with a triangular surface patch B (T3 to T6).
TT	Connects a triangular surface patch A (T3 to T6) with a triangular surface patch B (T3 to T6).
TQ	Connects a triangular surface patch A (T3 to T6) with a quadrilateral surface patch B (Q4 to Q8).
Q	Connects the shell vertex grid GS with a quadrilateral surface patch A (Q4 to Q8) if surface patch B is not specified.
T	Connects the shell vertex grid GS with a triangular surface patch A (T3 to T6) if surface patch B is not specified.

10. GAI are required for the format “GRIDID”. At least 3 and at most 8 grid IDs may be specified for GAI and GBi, respectively. The rules of the triangular and quadrilateral elements apply for the order of GAI and GBi, see [Figure 9-86](#). Missing midside nodes are allowed.

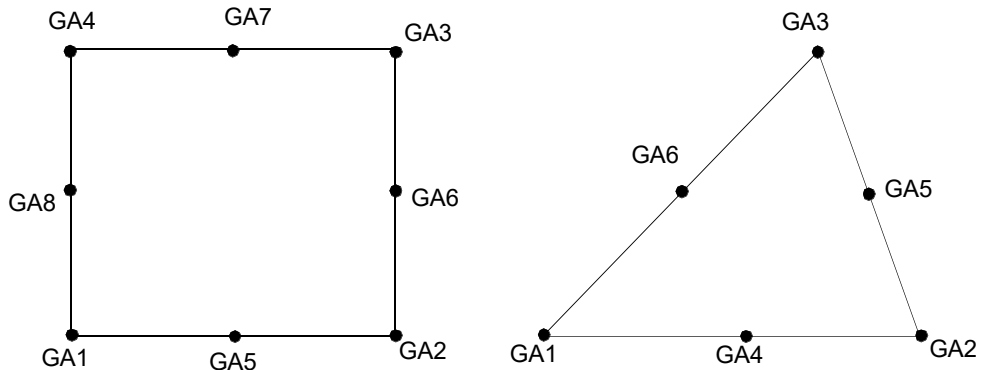


Figure 9-86 Quadrilateral and Triangular Surface Patches defined with Format GRIDID

11. The format “ALIGN” defines a point to point connection, see [Figure 9-87](#). GA and GB are required, they must be existing vertex nodes of shell elements. For the other formats, GA and GB are not required. Two shell normals in the direction GA-GB are generated at GA and GB, respectively.

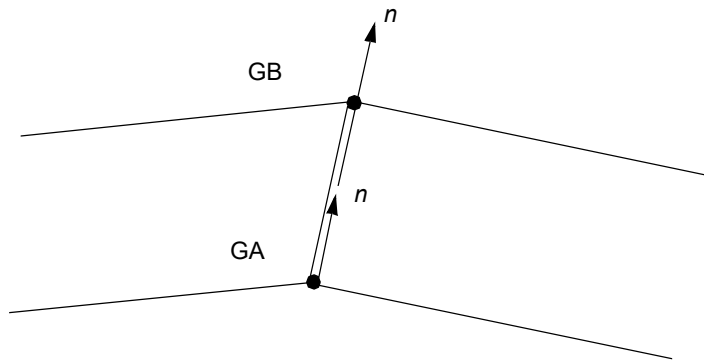


Figure 9-87 Point to Point Connection Defined with Format ALIGN

12. Forces and moments are output in the element coordinate system, see [Figure 9-88](#). The element coordinate system is constructed using the following rules:

The element x-axis points from GA to GB.

$$e_1 = \frac{x_B - x_A}{\|x_B - x_A\|} \quad \text{element x-axis}$$

In case of zero length, the normal of shell A is taken. All vector components are in basic if not noted otherwise.

Find the smallest component  $j$  of  $e_1$

$$ce_1^i = |e_1^i|$$

$$ce_1^j = \min_{i = 1, 2, 3} \left\{ ce_1^i \right\}.$$

Note that  $ce_1^i$  will be set to  $10^{-6}$  if  $ce_1^i < 10^{-6}$ .

In case of two equal components we take the one with the smaller  $i$ . The corresponding basic vector

$$b_j, \text{ e.g., for } j = 3, b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$



provides a good directional choice for  $e_2$ . In addition, the vector  $e_2$  must be orthogonal to  $e_1$ .

$$\tilde{e}_2 = b_j - \frac{e_1^T b_j}{e_1^T e_1} e_1 \quad e_2 = \frac{\tilde{e}_2}{\|\tilde{e}_2\|} \text{ element y-axis}$$

and  $e_3$  is just the cross product

$$e_3 = e_1 \times e_2 \quad \text{element z-axis}$$

The final transformation matrix is

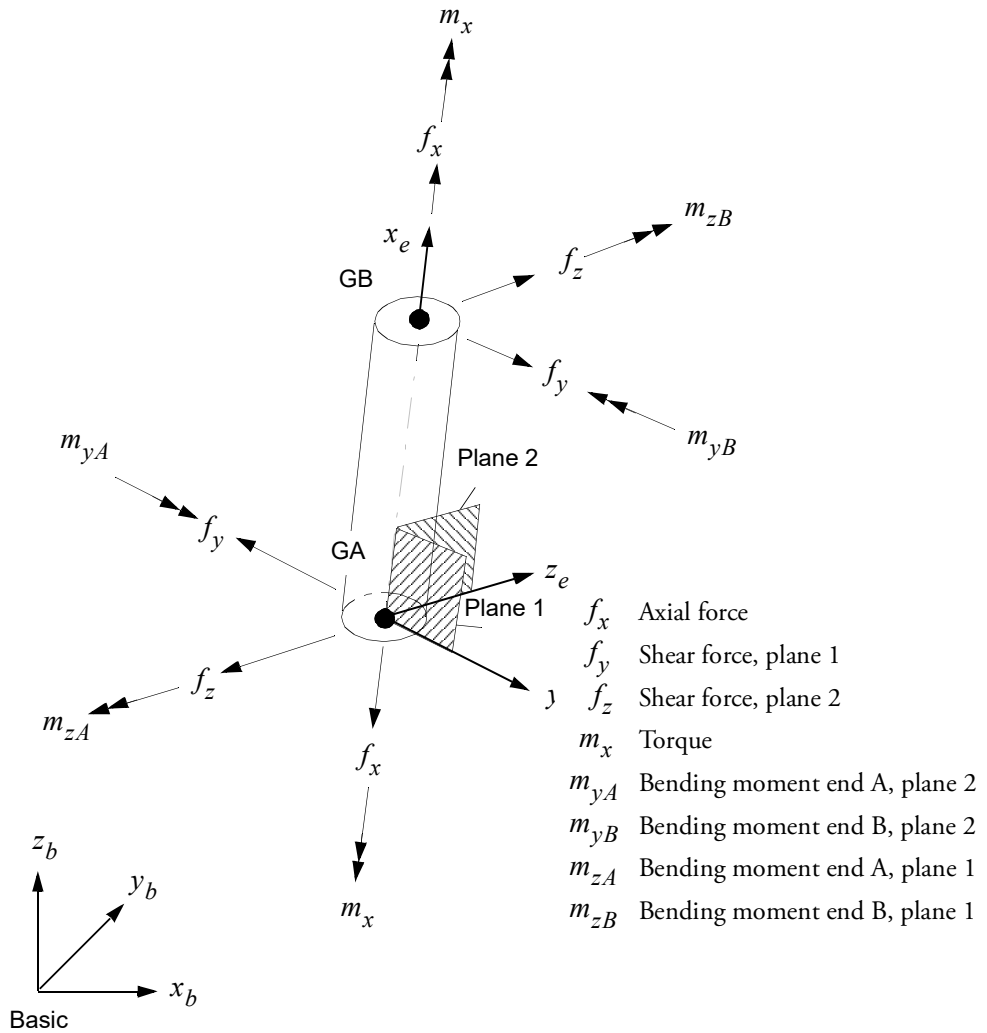
$$T_{be} = \begin{bmatrix} e_1 & e_2 & e_3 \end{bmatrix}$$

Figure 9-88 Element Coordinate System and Sign Convention of Element Forces

13. If "PARAM, OLDWELD, YES", in a SOL 400 analysis the CWELD element using method ELEMID, GRIDID, ELPAT or PARTPAT internally gets decomposed in a CBEAM element and a number of RBE3 elements. For method ALIGN the CWELD element internally gets replaced by a CBEAM element. The CBEAM element obtains a circular cross section with its diameter and material properties taken from the PWELD input and its length is determined by the final locations of the GA and GB grids of the CWELD. The RBE3 elements connect the GA and GB grids to the plate structures. For methods ELEMID and GRIDID one RBE3 element is used on each side to establish the connection. For methods ELPAT and PARTPAT five RBE3 elements are used on each side to establish the connection.
14. The output format of the forces and moments including the sign convention is identical to the CBAR element, see [Element Force Item Codes](#).
15. This entry is not supported in SOL 700.
16. MCID = -1 or blank (Default), then the coordinate system is as defined in Remark 12.

If MCID ≥ 0, then a “beam” like coordinate system is defined. The  $x_{elem}$  axis direction of the connector defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\|\vec{x}_B - \vec{x}_A\|}$$



The T2 direction defined by MCID will be used to define the orientation vector  $\vec{V}$  of the connector. Then the  $z_{elem}$  element axis will be defined as

$$\vec{e}_3 = \frac{\vec{e}_1 \times \vec{v}}{\|\vec{e}_1 \times \vec{v}\|}$$

The element  $y_{elem}$  axis is defined as

$$\vec{e}_2 = \vec{e}_3 \times \vec{e}_1$$

17. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
18. In SOL400, the behavior of this element in regard to large rotation is affected by the Case Control Command Rigid.
19. If partitioned superelements are present, then CWELD is supported in the main Bulk Data section only.

**CYAX**      Grid Points on Axis of Symmetry

Lists grid points that lie on the axis of symmetry in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYAX	G1	G2	G3	G4	G5	G6	G7	G8	
	G9	G10	-etc.-						

Example:

CYAX	27	152	THRU	160	192	11			
------	----	-----	------	-----	-----	----	--	--	--

Describer	Meaning
Gi	A list of grid points on the axis of symmetry. (Integer > 0 or Character "THRU")

Remarks:

1. The displacement coordinate system (see CD field on GRID entry) for a grid point lying on the axis of symmetry must be a rectangular system with the z-component of motion aligned with the axis of symmetry. The positive axis of symmetry is defined so that the azimuthal angle from positive side 1 to side 2 of a segment is in the same direction as the angle from T1 to T2 for the axis point. This is consistent with left- or right-hand rule.
2. If the dihedral symmetry option (STYPE = "DIH" on the CYSYM entry) is selected, the y-axis must be perpendicular to side 1.
3. Grid points lying on the axis of symmetry may be constrained by SPCs but not by MPCs. If the number of segments is three or more, SPCs must be applied to both components 1 and 2 or to neither, and SPCs must be applied to both components 4 and 5 or to neither in order to satisfy symmetry. In addition, the degrees-of-freedom (not constrained by SPCs, if any) at these grid points must be in the analysis set (a-set). If all degrees-of-freedom of grid points on the axis of symmetry are constrained by SPCs (including heat transfer, where there is only one degree-of-freedom), the grid point should not be listed on the CYAX entry.
4. Grid points lying on the axis of symmetry must not be defined on side 1 or on side 2 by means of a CYJOIN entry.
5. The word "THRU" must not appear in fields 2 or 9.

## CYJOIN

## Cyclic Symmetry Boundary Points

Defines the boundary points of a segment in cyclic symmetry problems.

Format:

1	2	3	4	5	6	7	8	9	10
CYJOIN	SIDE	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

CYJOIN	1	T2	7	9	16	THRU	33	64	
	72	THRU	89						

Describer	Meaning
SIDE	Side identification. (Integer 1 or 2)
C	Type of coordinate system used on boundaries of dihedral or axisymmetry problems. See Remark 3. (Character: "T1", "T2", "T3", "R", "C", "S")
Gi	Grid or scalar point identification numbers. (Integer > 0 or Character "THRU")

Remarks:

1. CYJOIN entries are used only for cyclic symmetry problems. The CYSYM entry must be used to specify rotational, dihedral, or axisymmetry.
2. For rotational or axisymmetry problems, there must be one logical entry for SIDE = 1 and one for SIDE = 2. The two lists specify grid points to be connected; therefore, both lists must have the same length.
3. For dihedral problems, side 1 refers to the boundary between segments and side 2 refers to the middle of a segment. For dihedral and/or AXI type of symmetry, the grid point degree-of-freedom that is normal to the boundary must be specified in field 3 as "T1", "T2", or "T3" ("R", rectangular, and "C", cylindrical, are the same as "T2" while "S", spherical, is the same as "T3"). For scalar and extra points with one degree-of-freedom, these should be specified as blank, "T2", or "T3" if they are to have the same sign, and "T1", if the two connected points are to be opposite in sign.
4. All components of displacement at boundary points are connected to adjacent segments except those constrained by SPCi, MPC, or OMITi entries.
5. The points on the axis of symmetry of the model, defined in the CYAX entry must not be defined as a side 1 or side 2 point by means of this entry.
6. The word "THRU" may not appear in fields 4 or 9 of the parent entry and fields 2 or 9 on the continuation entries.
7. All grid points that are implicitly or explicitly referenced must be defined.

8. For rotational and axisymmetry problems, the displacement coordinate systems must be consistent between sides 1 and 2. This is best satisfied by the use of a spherical or cylindrical coordinate system.

**CYLINDR**

## Defines the Shape of a Cylinder

Cylindrical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
CYLINDR	VID		XC1	YC1	ZC1	XC2	YC2	ZC2	
	RAD								

Example:

CYLINDR	4		0.	0.	0.	1.	1.	1.	
	.5								

Describer	Meaning
VID	Unique cylinder number. (Integer > 0; Required)
XC1, YC1, ZC1	Coordinates of point 1, See Remark 1. (Real; Required)
XC2, YC2, ZC2	Coordinates of point 2. See Remark 1. (Real; Required)
RAD	Radius of the cylinder. (Real; Required)

Remarks:

1. A cylinder is defined by the two end points of the cylinder axis and a radius.
2. Initial conditions are defined for the elements that are fully or partially inside the cylinder.
3. See also the TICEUL1 Bulk Data entry.

## CYSUP

## Fictitious Supports for Cyclic Symmetry

Defines fictitious supports for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSUP	GID	C							

Example:

CYSUP	16	1245							
-------	----	------	--	--	--	--	--	--	--

Describer	Meaning
GID	Grid point identification number. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

- Components of motion defined on this entry may not appear on SPC, SPC1, OMIT, OMIT1 entries, or in rigid elements or multipoint constraints as dependent degrees-of-freedom.
- Supports are applied at the grid point identified in field 2 to prevent rigid body motions in static analysis, or to define rigid body modes in eigenvalue analysis. All degrees-of-freedom should be at a single grid point. In other words, there can only be one such supported grid point in the model. The supports are applied only to the cyclic components of order  $k=0$  or  $k=1$ . In order to satisfy conditions of symmetry, certain restrictions are placed on the location of the grid point and the orientation of its displacement coordinate system, as shown in the following table:

Symmetry Option (STYPE on the CYSYM entry)	ROT	ROT	DIH	DIH	DIH
Number of Segments, N	2	$\geq 3$	1	2	$\geq 3$
Location of Grid Point	See Note <a href="#">c</a> .	See Note <a href="#">d</a> .	Side 1	Side 1	Side 1
Special Restrictions on Displacement Coordinate System	See Notes <a href="#">a</a> . and <a href="#">e</a> .	See Note <a href="#">b</a> .	None	See Note <a href="#">a</a> .	See Note <a href="#">b</a> .

Notes:

- T3 axis must be parallel to axis of symmetry.
- Displacement coordinate system at the referenced grid point must be cylindrical with z-axis along the axis of symmetry.



- c. Any location except on side 2.
  - d. Any location except on the axis of symmetry or on side 2.
  - e. If the grid point is on the axis of symmetry, the displacement coordinate system must be rectangular.
3. If the number of segments,  $N$ , is 1 (in the case of DIH symmetry) or 2 (in the case of ROT or AXI symmetry), it is important that the rotational components referenced in field 3 be elastically connected to the structure. If  $N \geq 2$  (in the case of DIH symmetry) or  $N \geq 3$  (in the case of ROT or AXI symmetry), it is not important, because in this case the supports for rigid body rotation are actually applied to translational motions.
  4. If  $N \geq 3$ , supports will be applied to both the 1 and 2 (inplane-translational) components, if either is referenced, and to both the 4 and 5 (out-of-plane rotational) components, if either is referenced. If component 6 is supported, component 2 should not appear on OMIT or OMIT1 entries.
  5. The restrictions noted in Remarks 2. and 4. are related to symmetry requirements. For  $N \geq 3$ , symmetry requires that the supports be symmetrical (or antisymmetrical), with respect to any plane passing through the axis of symmetry. For the DIH options,  $N = 1$  and  $N = 2$ , symmetry requires that the supports be symmetrical (or antisymmetrical) with respect to the plane(s) of symmetry. For the ROT option,  $N = 2$ , symmetry requires that a support be either parallel or perpendicular to the axis of symmetry.
  6. GID must be a grid point and not a scalar point.

**CYSYM****Cyclic Symmetry Parameters**

Defines parameters for cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
CYSYM	NSEG	STYPE							

Example:

CYSYM	6	ROT							
-------	---	-----	--	--	--	--	--	--	--

Describer	Meaning
NSEG	Number of segments. (Integer > 0)
STYPE	Symmetry type. (Character: "ROT", "DIH", or "AXI")

Remarks:

1. STYPE = "AXI" is a special case of STYPE = "ROT" used to model axisymmetric structures.
2. If STYPE = "AXI", then all grid points must lie on side 1, side 2, or the axis. Also, plate elements with midside grid points may not be defined.

## Entries D - E

### DAMPING

### Parameter and Hybrid Damping Specification

Specifies the values for parameter damping and/or selects optional HYBRID damping.

#### Format:

1	2	3	4	5	6	7	8	9	10
DAMPING	ID	G	ALPHA1	ALPHA2	HYBRID	GEFACT	ROTSEP		
	W3	W4	WH						

#### Example:

DAMPING	100	.02					NO		
---------	-----	-----	--	--	--	--	----	--	--

Describer	Meaning
ID	Damping entry identification number. (Integer > 0; no Default)
G	Structural damping coefficient, see Remark 1. (Real; Default = 0.0)
ALPHA1	Scale factor for mass portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)
ALPHA2	Scale factor for stiffness portion of Rayleigh damping, see Remark 4. (Real; Default = 0.0)
HYBRID	Identification number of HYBDAMP entry for hybrid damping, see Remarks 5. and 6. (Integer $\geq$ 0; Default = 0)
GEFACT	Scale factor for material damping. See Remark 1. and Remark 3. (Real; Default = 1.0)
ROTSEP	Remove rotor stiffness, mass, and structural damping from the hybrid damping calculation (Character: YES or NO; Default=YES)
W3	Average frequency for calculation of structural damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0)
W4	Average frequency for calculation of material damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0)
WH	Average frequency for calculation of hybrid ‘structural’ damping in transient response, see Remark 7. (Real $\geq$ 0.0; Default = 0.0)

#### Remarks:

1. The DAMPING entry is referenced by the RSDAMP or SEDAMP Case Control commands. If a DAMPING entry is selected in the Case Control, the DAMPING values, including defaults, will override parameter inputs.
2. All damping selections are cumulative.
3. Structural damping specified by the G and GEFACT entries will replace any structural damping by:

$$[K4]_{NEW} = G[K] + GFACT[K4]$$

The GFACT value used is  $GFACT = GE$  (default=0.0) \*  $GEFACT$  (default=1.0) where  $GE$  is the structural damping value on the material entry.

4. Rayleigh viscous damping is calculated as:

$$[B]_{Rayleigh} = \alpha_1[M] + \alpha_2[K]$$

5. Hybrid damping on the residual structure is only active for direct solution sequences. For modal solution sequences, the SDAMP Case Control request should be used.
6. Hybrid damping for superelements uses modes that are calculated using the superelement mass and stiffness matrices before upstream superelements are added and before SPC or MPC constraints are imposed. These matrices are known as the JJ-type matrices.
7. The W3 and W4 values are used in transient response only. A zero value (default) will result in no damping. The equivalent viscous damping is calculated as:

$$[B]_{equiv} = \left(\frac{G}{W3}\right)[K] + \left(\frac{GFACT}{W4}\right)[K4] + \left(\frac{1}{WH}\right)[KH]$$

8. For rotordynamic analyses (RGYRO in the Case Control Section), the DAMPING calculations for the residual structure (RSDAMP) are performed without any rotor contributions (support structure only).
9. Rayleigh damping is designed to be applied only at Superelement, Part Superelement, or External Superelement residual assembly time. It is not applied in individual Parts or External superelements.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

### Examples:

- Specify a structural damping coefficient of 0.03 for the residual structure for frequency response.  
Case Control:  
RSDAMP=100  
Bulk Data:  
DAMPING, 100, 0.03
- Specify a structural damping coefficient of 0.03 for the residual structure for transient response (use a “natural” frequency of 100 Hz)  
Case Control:  
RSDAMP=100  
Bulk Data:  
DAMPING, 100, 0.03,  
, 628.3
- Specify hybrid damping for superelement 1. Use modal damping of two percent critical for the first 6 modes.

Case Control:

```
SUBCASE 1  
  SUPER=1  
  SEDAMP=100
```

Bulk Data:

```
DAMPING, 100, , , , 101  
HYBDAMP, 101, 102, 1001  
EIGRL, 102, , , 6  
TABDMP1, 1001, CRIT,  
, 0.0, 0.02, 1000.0, 0.02, ENDT
```

**DAREA**

## Load Scale Factor

Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with ACSRCE, RLOADi and TLOADi entries.

**Format:**

1	2	3	4	5	6	7	8	9	10
DAREA	SID	P1	C1	A1	P2	C2	A2		

**Example:**

DAREA	3	6	2	8.2	15	1	10.1		
-------	---	---	---	-----	----	---	------	--	--

Describer	Meaning
SID	Identification number. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point; blank or 0 for extra or scalar point.)
Ai	Scale (area) factor. (Real)

**Remarks:**

1. One or two scale factors may be defined on a single entry.
2. Refer to RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entries for the formulas that define the scale factor Ai in dynamic analysis.
3. Component numbers refer to the displacement coordinate system.
4. In dynamic analysis, DAREA entries may be used with LSEQ Bulk Data entries if LOADSET is specified in Case Control. The LSEQ and static load entries will be used to internally generate DAREA entries.
5. If DAREA is referenced by a GUST entry, Pi must be defined. However, it is only used if selected through a DLOAD Case Control command. WG from the GUST entry is used instead of Ai when requested via a GUST entry.
6. All DAREA entries corresponding to all grid and scalar points are automatically converted internally by the program to equivalent FORCE/MOMENT/SLOAD entries (as appropriate) *if there are no LSEQ Bulk Data entries.*
7. In superelement analysis, DAREA may be used to specify loads not only on the interior points of the residual, but also on the interior points of upstream superelements *if there are no LSEQ Bulk Data entries.*
8. In static analysis, DAREA entries may be used *only if there are no LSEQ Bulk Data entries.* They are ignored if there are any LSEQ Bulk Data entries.

## DCONADD

## Design Constraint Set Combination

Defines the design constraints for a subcase as a union of DCONSTR entries.

### Format:

1	2	3	4	5	6	7	8	9	10
DCONADD	DCID	DC1	DC2	DC3	DC4	DC5	DC6	DC7	
	DC8	-etc.-							

### Example:

DCONADD	10	4	12						
---------	----	---	----	--	--	--	--	--	--

Describer	Meaning
DCID	Design constraint set identification number. (Integer > 0)
DCi	DCONSTR entry identification number. (Integer > 0)

### Remarks:

1. The DCONADD entry is selected by a DESSUB or DESGLB Case Control command.
2. All DCi must be unique from other DCi.
3. For PART SE, DCi from different PART SEs can be referenced on a single DCONADD and only DCONADD in the main Bulk Data Section, starts with 'BEGIN BULK' or 'BEGIN SUPER=0', will be considered as part of design constraints. Note that DCONADD entries in 'BEGIN SUPER=seid' where seid>0 will be ignored.

## DCONSTR

## Design Constraints

Defines design constraints.

### Format:

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW/LID	UALLOW/UID	LOWFQ	HIGHFQ			

### Example:

DCONSTR	10	4	1.25						
---------	----	---	------	--	--	--	--	--	--

Describer	Meaning
DCID	Design constraint set identification number. (Integer > 0)
RID	DRESPi entry identification number. (Integer > 0)
LALLOW/LID	Lower bound on the response quantity or the set identification ID of a number of a TABLEDi entry that supplies the lower bound as a function of frequency. (Real; Default = -1.0E20)
UALLOW/UID	Upper bound on the response quantity or the set identification ID of a number of a TABLEDi entry that supplies the upper bound as a function of frequency. (Real; Default = 1.0E20)
LOWFQ	Low end of frequency range in Hertz (Real ≥ 0.0; Default = 0.0). See Remark 8.
HIGHFQ	High end of frequency range in Hertz (Real ≥ LOWFQ; Default = 1.0E+20). See Remark 8.

### Remarks:

1. The DCONSTR entry may be selected in the Case Control Section by the DESSUB or DESGLB command.
2. DCID may be referenced by the DCONADD Bulk Data entry.
3. For a given DCID, the associated RID can be referenced only once.
4. The units of LALLOW and UALLOW must be consistent with the referenced response defined on the DRESPi entry. If RID refers to an "EIGN" response, then the imposed bounds must be expressed in units of eigenvalue, (radian/time)<sup>2</sup>. If RID refers to a "FREQ" response, then the imposed bounds must be expressed in cycles/time.
5. LALLOW and UALLOW are unrelated to the stress limits specified on the MATi entry.
6. Constraints are computed as follows:

$$g = \frac{LALLOW - r}{GNORM} \text{ for lower bound constraints}$$



$$g = \frac{r - UALLOW}{GNORM} \text{ for upper bound constraints}$$

$$GNORM = \begin{cases} |LALLOW| & \text{for lower bounds if } |LALLOW| > GSCAL \\ |UALLOW| & \text{for upper bounds if } |UALLOW| > GSCAL \\ GSCAL & \text{otherwise} \end{cases}$$

GSCAL is specified on the DOPTPRM entry (Default = 0.001)

7. As Remark 6. indicates, small values of UALLOW and LALLOW require special processing and should be avoided. Bounds of exactly zero are particularly troublesome. This can be avoided by using a DRESP2 entry that offsets the constrained response from zero.
8. LOWFQ and HIGHFQ fields are functional only for frequency response RTYPE's, including DYSTIFF or those with a 'AC', 'AF', 'FR' or 'PS' prefix. (FREQ is not a frequency response). The LOWFQ and HIGHFQ fields are also functional when they are applied to a DRESP2 or DRESP3 that inherit the frequency values from these RTYPES. The bounds provided in LALLOW and UALLOW are applicable to a response only when the value of the forcing frequency of the response falls between LOWFQ and HIGHFQ. The behavior varies depending on the value of ATTB on the DRESP1 entry.
  - a. If the field is blank, the LOWFQ/HIGHFQ fields are honored. If the DCONSTR with LOWFQ/HIGHFQ refers to a DRESP2 that invokes a DRESP1 with ATTB blank, the fields are not honored.
  - b. If the field is a character, the LOWFQ/HIGHFQ fields are honored.
  - c. If the field is a real number, the LOWFQ/HIGHFQ fields are honored even when the DCONSTR refers to a DRESP2 that references the DRESP1.
9. LID and UID are optional inputs that identify tabular input to specify the lower and upper bounds as a function of frequency. They are applicable to the 'FR' and 'PSD' responses of Remark 8. and to DRESP2 and DRESP3 responses that inherit the frequency value from these RTYPES.
10. For PART SE, DCONSTR entries can reside in each individual PART SE Bulk Data Section starting with 'BEGIN SUPER=seid'. If DCID is different from a PART SE to the next, DCONADD in the residual (or main Bulk Data Section) can be utilized to group DCONSTR entries together for a single subcase.

**DDVAL** Discrete Design Variable Values

Defines real, discrete design variable values for use in discrete variable optimization, topology optimization or rotordynamics.

**Format:**

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DVAL1	DVAL2	DVAL3	DVAL4	DVAL5	DVAL6	DVAL7	

**Alternate Format:**

DDVAL	ID	DVAL1	THRU	DVAL2	BY	INC			
-------	----	-------	------	-------	----	-----	--	--	--

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

**Continuation Entry Format 1:**

	DVAL8	DVAL9	DVAL10	DVAL11	-etc.-				
--	-------	-------	--------	--------	--------	--	--	--	--

**Continuation Entry Format 2:**

	DVAL8	THRU	DVAL9	BY	INC				
--	-------	------	-------	----	-----	--	--	--	--

**Example:**

1	2	3	4	5	6	7	8	9	10
DDVAL	110	0.1	0.2	0.3	0.5	0.6	0.4		
	.7	THRU	1.0	BY	0.05				
	1.5	2.0							

Descriptor	Meaning
ID	Unique set identification number (Integer > 0) See Remark 1.
DVALi	Discrete values (Real, or THRU or BY) See Remarks 2. through 5.
INC	Discrete value increment (Real) See Remarks 2., 4., 5.

**Remarks:**

- DDAVAL entries may be referenced by a DESVAR entry in its DDVAL field (field 8), a TOMVAR entry in its DSVID field, a entry in its DDVALID field (field 4) or a RSPINR entry in its SPTID field (field 6).

2. Trailing fields on a DDVAL record can be left blank if the next record is of type DVAL<sub>i</sub> THRU DVAL<sub>j</sub> BY INC. Also fields 7 - 9 must be blank when the type DVAL<sub>i</sub> THRU DVAL<sub>j</sub> BY INC is used in fields 2 - 6 and fields 8 - 9 must be blank when the type DVAL<sub>i</sub> "THRU" DVAL<sub>j</sub> "BY" INC is used in fields 3 - 7 for the first record. Embedded blanks are not permitted in other cases.
3. The DVAL<sub>i</sub> values in a sequence need not be in an ascending or descending order. They can be in any order.
4. If the format DVAL<sub>i</sub> THRU DVAL<sub>j</sub> BY INC is employed, INC must be a positive real value if DVAL<sub>i</sub> < DVAL<sub>j</sub> and INC must be a negative real value if DVAL<sub>i</sub> > DVAL<sub>j</sub>. The program terminates the execution with an appropriate fatal error if this requirement is not satisfied.
5. The format DVAL<sub>i</sub> THRU DVAL<sub>j</sub> BY INC defines a list of discrete values given by DVAL<sub>i</sub>, DVAL<sub>i</sub>+INC, DVAL<sub>i</sub>+2.0\*INC, ..., DVAL<sub>j</sub>. The last discrete value of DVAL<sub>j</sub> is always included, even if the range is not evenly divisible by INC.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## DEACTEL

## Elements To Be Deactivated for SOL 400 Analysis

This entry identifies the elements that do not participate in the analysis or do not participate in a particular physics pass of a coupled analysis. SOL 400 for NLSTATIC, and NLTRAN, as well Perturbation analyses only.

### Format:

1	2	3	4	5	6	7	8	9	10
DEACTEL	ID				ISET				

### Example:

DEACTEL	2				200				
---------	---	--	--	--	-----	--	--	--	--

Describer	Meaning
ID	ID of the DEACTEL entry. This is referenced by a DEACTEL=ID Case Control command. (Integer; no Default)
ISET	SID of a list of elements defined by a SET3 entry. (Integer; no Default)

### Remarks:

1. DEACTEL can only be referenced before the first subcase and/or inside the first subcase and/or inside the first step of the first subcase and/or inside each substep of the first step of the first subcase. References that are made anywhere else are currently ignored. It is referenced by the Case Control command DEACTEL=ID.
2. The description field (DES) of the SET3 entry referenced by ISET can only be of type ELEM.
3. Element IDs in the list originating from some set that do not exist are ignored.

## DEACTEL Define Elements that Should be Deactivated for a Particular Subcase in SOL 600

This entry allows the user to deactivate elements that have failed or are no longer necessary in a particular subcase. Some or all of these elements can be re-activated in a subsequent subcase using the ACTIVAT entry.

### Format:

1	2	3	4	5	6	7	8	9	10
DEACTEL	ID	STRESS	STRAIN	IGEOM	ISET				

### Example:

DEACTEL	2	1	1	0	200				
---------	---	---	---	---	-----	--	--	--	--

Describer	Meaning
ID	Identification number of a matching DEACTEL Case Control command defining the subcase to which these elements should be deactivated. (Integer; no Default)
STRESS	Flag to control output stresses after the elements have been deactivated. (Integer; Default = 0) 0 Stresses have the same value as just prior to deactivation 1 Stresses are set to zero
STRAIN	Flag to control output strains after the elements have been deactivated. (Integer; Default = 0) 0 Strains have the same value as just prior to deactivation 1 Strains are set to zero
IGEOM	Flag to control whether to update the geometry on the post file. (Integer; Default = 0) 0 Update the geometry. 1 Do not update the geometry.
ISET	ID of a list of elements described by SET3 is ID=ISET to be deactivated. (Integer; no Default)

### Remarks:

1. This entry maps to Marc's DEACTIVATE (option A) History definition (option B is not supported in SOL 600).

**DEFORM**

## Static Element Deformation

Defines enforced axial deformation for one-dimensional elements for use in statics problems.

**Format:**

1	2	3	4	5	6	7	8	9	10
DEFORM	SID	EID1	D1	EID2	D2	EID3	D3		

**Example:**

DEFORM	1	535	.05	536	-.10				
--------	---	-----	-----	-----	------	--	--	--	--

Describer	Meaning
SID	Deformation set identification number. (Integer > 0)
EIDi	Element number. (Integer > 0)
Di	Deformation. (Real; positive value represents elongation.)

**Remarks:**

1. The referenced element must be one-dimensional (CROD, CONROD, CTUBE, CBAR, CBEAM).
2. Deformation sets must be selected in the Case Control Section with DEFORM = SID.
3. One to three enforced element deformations may be defined on a single entry.
4. The DEFORM entry, when called by the DEFORM Case Control command, is applicable to linear static, inertia relief, differential stiffness, and buckling (Solutions 101, 105, 114, and 200) and will produce fatal messages in other solution sequences. Use SPCD to apply enforced displacements in solution sequences for which DEFORM does not apply.

## DEFUSET

## Degree-of-Freedom Set Name Definition

Defines new names for degree-of-freedom sets.

### Format:

1	2	3	4	5	6	7	8	9	10
DEFUSET	OLD1	NEW1	OLD2	NEW2	OLD3	NEW3	OLD4	NEW4	

### Example:

DEFUSET	U2	X	U4	Y	U3	Z			
---------	----	---	----	---	----	---	--	--	--

Describer	Meaning
OLDi	Default set name. (One to four characters)
NEWi	New set name. (One to four characters)

### Remarks:

1. From one to four set names may be specified on a single entry.
2. OLDi must refer to any of the set names given in [Degree-of-Freedom Sets](#). It is recommended that OLDi refer only to the set names U1 through U6. If sets PA or PS are referenced, a user fatal message is issued.
3. All NEWi names must be unique with respect to all other set names.
4. The DEFUSET entry is optional since default set names exist for all displacement sets.
5. The DEFUSET entry must be present in the Bulk Data Section in all restarts.

## DELAY Dynamic Load Time Delay

Defines the time delay term  $\tau$  in the equations of the dynamic loading function.

### Format:

1	2	3	4	5	6	7	8	9	10
DELAY	SID	P1	C1	T1	P2	C2	T2		

### Example:

DELAY	5	21	6	4.25	7	6	8.1		
-------	---	----	---	------	---	---	-----	--	--

Describer	Meaning
SID	Identification number of the DELAY entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integer 1 through 6 for grid point, blank or 0 for extra point or scalar point.)
Ti	Time delay $\tau$ for designated point Pi and component Ci. (Real)

### Remarks:

1. One or two dynamic load time delays may be defined on a single entry.
2. SID must also be specified on a RLOAD1, RLOAD2, TLOAD1, TLOAD2, or ACSRCE entry. See those entry descriptions for the formulas that define the manner in which the time delay  $\tau$  is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DELAY entries may only be applied to loads on points in the residual structure.



## DEQATN Equation Definition

Defines one or more equations for use in analysis.

### Format:

1	2	3	4	5	6	7	8	9	10	
DEQATN	EQID	EQUATION								
		EQUATION (Cont.)								

### Example:

DEQATN	14	F1(A, B, C, D, R) = A + B · C – (D ** 3 + 10.0) + sin(PI(1) · R) + A**2/(B – C) ; F = A + B – F1 · D							
--------	----	---	--	--	--	--	--	--	--

Describer	Meaning
EQID	Unique equation identification number. (Integer > 0)
EQUATION	Equation(s). See Remarks. (Character)

### Remarks:

1. EQUATION is a single equation or a set of nested equations and is specified in fields 3 through 9 on the first entry and may be continued on fields 2 through 9 on the continuation entries. On the continuation entries, no commas can appear in columns 1 through 8. All data in fields 2 through 9 must be specified in columns 9 through 72. The large-field format is not allowed.

A single equation has the following format:

$$\text{variable-1 (x1, x2, ..., xn)=expression-1}$$

A set of nested equations is separated by semicolons and has the format:

$$\begin{aligned} &\text{variable-1 (x1, x2, ..., xn)=expression-1;} \\ &\text{variable-2=expression-2;variable-3=expression-3;} \end{aligned}$$

etc.

$$\text{variable-m=expression-m}$$

Expression-i is a collection of constants, real variables, and real functions, separated by operators, and must produce a single real value. (x1, x2, ..., xn) is the list of all the variable names (except variable-i) that appear in all expressions. Variable-i may be used in subsequent expressions. The last equation, variable-m=expression-m, provides the value that is returned to the Bulk Data entry that references EQID; e.g., DRESP2. The example above represents the following mathematical equations:

$$F1 = A + B \cdot C - (D^3 + 10) + \sin(\pi \cdot R) + \frac{A^2}{B - C}$$

$$F = A + B + F1 \cdot D$$

where SIN and PI are intrinsic functions. See Remark 4.

2. EQUATION may contain embedded blanks. EQUATION must contain less than 32,000 nonblank characters. If more characters are required for use with a DRESP2 entry, the DRESP2 can be divided into two or more DRESP2 entries with a master DRESP2 referencing subsequent DRESP2s.
3. The syntax of the expressions follows FORTRAN language standards. The allowable arithmetic operations are shown in Table 8 in the order of execution precedence. Parenthesis are used to change the order of precedence. Operations within parentheses are performed first with the usual order of precedence being maintained within the parentheses.

Table 8 DEQATN Entry Operators

Operator	Operation	Sample Expressions	Interpreted As
-, +	Negative or Positive immediately preceded by exponentiation	$X^{**}(-Y)$	$X^{**}(-Y)$
**	Exponentiation	$-X^{**}Y$	$(-X^{**}Y)$
-, +	Negative or Positive	$-X-Y$	$(-X)-Y$
*, /	Multiplication or Division	$X*Y-Z$	$(X*Y)-Z$
+, -	Addition or Subtraction	$X+Y$	$X+Y$

4. The expressions may contain intrinsic functions. Table 9 contains the format and descriptions of functions that may appear in the expressions. The use of functions that may be discontinuous must be used with caution because they can cause discontinuous derivatives. These are ABS, DIM, MAX, MIN, and MOD. For examples and further details see the [MSC Nastran DMAP Programmer's Guide](#).

Table 9 DEQATN Entry Functions

Format	Description	Mathematical Expressions
ABS(x)	absolute value	$ x $
ACOS(x)	arccosine	$\cos^{-1} x$
ACOSH(x)	hyperbolic arccosine	$\cosh^{-1} x$
ASIN(x)	arcsine	$\sin^{-1} x$
ASINH(x)	hyperbolic arcsine	$\sinh^{-1} x$
ATAN(x)	arctangent	$\tan^{-1} x$
ATAN2(x,y)	arctangent of quotient	$\tan^{-1} (x/y)$
ATANH(x)	hyperbolic arctangent	$\tanh^{-1} x$
ATANH2(x,y)	hyperbolic arctangent of quotient	$\tanh^{-1} (x/y)$

Table 9 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
AVG( $X_1, X_2, \dots, X_n$ )	average	$\frac{1}{n} \sum_{i=1}^n X_i$
COS(x)	cosine	cos x
COSH(x)	hyperbolic cosine	cosh x
DB(P, PREF)	sound pressure in decibel	$20.0 \cdot \log\left(\frac{P}{PREF}\right)$
DBA(P, PREF, F)	sound pressure in decibel (perceived)	$20.0 \cdot \log\left(\frac{P}{PREF}\right) + 10.0 \cdot \log(Ta1) + 10.0 \cdot \log(Ta2)$
DIM(x,y)	positive difference	x-MIN(x,y)
EXP(x)	exponential	$e^x$
INVDB(DB, PREF)	inverse Db	$10^{\left(\frac{DB}{20.0} + \log PREF\right)}$
INVDBA(DBA, PREF, F)	inverse Dba	$10^{\left(\frac{DBA - 10.0 \cdot \log(Ta1) - 10.0 \cdot \log(Ta2)}{20.0}\right)}$
LOG(x)	natural logarithm	$\log_e x$
LOG10(x)	common logarithm	$\log_{10} x$
LOGX(x,y)	base x logarithm	$\log_x y$
MAX( $x_1, x_2, \dots$ )	maximum	maximum of $x_1$ , etc.
MIN( $x_1, x_2, \dots$ )	minimum	minimum of $x_1$ , etc.
MOD(x,y)	remainder (modulo)	$x - y \cdot (\text{INT}(x/y))$
PI(x)	multiples of pi ( $\pi$ )	$x \cdot \pi$
RSS( $X_1, X_2, \dots, X_n$ )	square root of sum of squares	$\sqrt{\sum_{i=1}^n X_i^2}$
SIN(x)	sine	sin x
SINH(x)	hyperbolic sine	sinh x
SQRT(x)	square root	$\sqrt{x}$

Table 9 DEQATN Entry Functions (continued)

Format	Description	Mathematical Expressions
SSQ( $X_1, X_2, \dots, X_n$ )	sum of squares	$\sum_{i=1}^n X_i^2$
SUM( $X_1, X_2, \dots, X_n$ )	summation	$\sum_{i=1}^n X_i$
TAN(x)	tangent	tan x
TANH(x)	hyperbolic tangent	tanh x

where

- $X_1, X_2, \dots, X_n, P$  = structure responses or acoustic pressure
- PREF = reference pressure
- F = forcing frequency
- DB = acoustic pressure in decibel
- DBA = perceived acoustic pressure in decibel

$$Ta1 = \frac{K3 \cdot F^4}{(F^2 + P2^2)(F^2 + P3^2)}$$

$$Ta2 = \frac{K1 \cdot F^4}{(F^2 + P1^2)^2 (F^2 + P4^2)^2}$$

$$K1 = 2.242882e+16$$

$$K3 = 1.562339$$

$$P1 = 20.598997$$

$$P2 = 107.65265$$

$$P3 = 737.86223$$

$$P4 = 12194.22$$

5. If the DEQATN entry is referenced by the:

- DVCREL2, DVMREL2, or DVPREL2 entry, then  $X_j$  represents the DVIDj and LABELk fields.

- DRESP2 entry, then  $X_i$  represents the DVIDj, LABELk, NRm, Gp, DPIPq, DCICr, DMIMs, DPI2Pt, DCI2Cu, DMI2Mv, and NRRw fields in that order.
6. If the DEQATN entry is referenced by DRESP2, DVCREL2, DVMREL2 or DVPREL2 entries, constants must be specified in single precision regardless of your machine's word length.
  7. The DMAP logical operators NOT, AND, OR, XOR, and XQV cannot be used as  $X_i$  names.
  8. Input errors on the DEQATN entry often result in poor messages. Substituting a “[” for a parenthesis or violating the restriction against large field format are examples. Known messages are UFM 215, SFM 233 and UFM 5199. If any of these messages are encountered then review the DEQATN entry input.
  9. Intrinsic functions AVG, MAX, MIN, RSS, SSQ and SUM are limited to <97 arguments. If more arguments are desired, the functions may be divided up. For example, to perform a sum of squares on 170 items, use:

$$SSQ(X1,X2,...X95) + SSQ(X96,X97,...X170).$$

For MAX/MIN, the following concatenation can be used:

$$MAX1 = MAX(X1,X2,..X95), MAX2=MAX(X96,X97...X170); MAXT=MAX(MAX1,MAX2).$$

For AVG, an example to average 170 terms is:

$$AVG1=AVG(X1,X2,...X95);AVG2=AVG(X96,X97..X170); AVGT = (95. * AVG1 + 75. * AVG2) / 170.$$

10. Arithmetic is carried out using the type of the input data. For example, in the expression:

$$X = A^{**}(1/2)$$

both values in the exponent are integers so that the value returned for the exponent is calculated using integer arithmetic or  $1/2 = 0$ . In this case  $1/2$  should be replaced by (.5).

**DESVAR**

## Design Variable

Defines a design variable for design optimization.

**Format:**

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	DELXV	DDVAL		

**Example:**

DESVAR	2	BARA1	35.0	10.	100.	0.2			
--------	---	-------	------	-----	------	-----	--	--	--

Describer	Meaning
ID	Unique design variable identification number. (Integer > 0)
LABEL	User-supplied name for printing purposes. (Character)
XINIT	Initial value. (Real; $XLB \leq XINIT \leq XUB$ )
XLB	Lower bound. (Real; Default = $-1.0E+20$ )
XUB	Upper bound. (Real; Default = $+1.0E+20$ )
DELXV	Fractional change allowed for the design variable during approximate optimization. (Real > 0.0; for Default see Remark 2.)
DDVAL	ID of a DDVAL entry that provides a set of allowable discrete values. (Blank or Integer > 0; Default=blank for continuous design variables. See Remark 3.)

**Remarks:**

1. DELXV can be used to control the change in the design variable during one optimization cycle.
2. If DELXV is blank, the default is taken from the specification of the DELX parameter on the DOPTPRM entry. If DELX is not specified, then the default is 0.5.
3. If the design variable is to be discrete (Integer > 0 in DDVAL field), and if either of the XLB and/or XUB bounds are wider than those given by the discrete list of values on the corresponding DDVAL entry, XLB and/or XUB will be replaced by the minimum and maximum discrete values.

## DETSPH

## Spherical Detonation Wave

Defines the ignition point from which a spherical detonation wave travels, causing the reaction of high explosive materials. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
DETSPH	DID	MID	X	Y	Z	VEL	TIME		

### Example:

DETSPH	100	10	96.5	177.6	37.4	2379.	1.7E-6		
--------	-----	----	------	-------	------	-------	--------	--	--

Describer	Meaning
DID	Unique detonation number. (Integer > 0; Required)
MID	References MATDEUL id of the exploding material. (Integer > 0; Required)
X, Y, Z	Coordinates of the ignition point. (Real, 0.0)
VEL	Velocity of the detonation wave. (Real ≥ 0.0, 0.0)
TIME	Detonation time. (Real ≥ 0.0, 0.0)

### Remark:

1. An element detonates when a spherical detonation wave originating from the detonation point at the specified time reaches the element.

**DIVERG**

## Divergence Analysis Data

Defines Mach numbers (m) for a static aeroelastic divergence analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
DIVERG	SID	NROOT	M1	M2	M3	M4	M5	M6	
	M7	M8	-etc.-						

**Example:**

DIVERG	70	2	.5	.8	.9				
--------	----	---	----	----	----	--	--	--	--

Describer	Meaning
SID	Unique set identifier. (Integer > 0)
NROOT	Number of divergence roots that are to be output and their eigenvectors printed. (Integer; Default = 1)
Mi	Mach number. (Real $\geq$ 0.0)

**Remarks:**

1. The DIVERG entry is referenced in Case Control by “DIVERG = SID”.
2. The NROOT lowest divergence dynamic pressures are printed. If there are fewer than NROOT pressures, all available dynamic pressures are printed.
3. Mi values must be distinct.
4. A blank Mach number field terminates the input.



## DLINK Multiple Design Variable Linking

Relates one design variable to one or more other design variables.

### Format:

1	2	3	4	5	6	7	8	9	10
DLINK	ID	DDVID	C0	CMULT	IDV1	C1	IDV2	C2	
	IDV3	C3	-etc.-						

### Example:

DLINK	10	2	0.1	0.33	4	2.0	6	-1.0	
	8	7.0							

Describer	Meaning
ID	Unique identification number. (Integer > 0)
DDVID	Dependent design variable identification number. (Integer > 0)
C0	Constant term. (Real; Default = 0.0)
CMULT	Constant multiplier. (Real; Default = 1.0)
IDVi	Independent design variable identification number. (Integer > 0)
Ci	Coefficient i corresponding to IDVi. (Real)

### Remarks:

- DLINK defines the relationship

$$DDVID = C0 + CMULT * \sum_i C_i * IDV_i$$

- This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.
- CMULT provides a simple means of scaling the Ci. For example if Ci = 1/7, 2/7, 4/7, etc. is desired, then CMULT = 1/7 and Ci = 1, 2, 4, etc., may be input.
- An independent IDVi must not occur on the same DLINK entry more than once.
- ID is for user reference only.
- If a design variable is specified as dependent on a DLINK entry, then it cannot be specified as independent on another DLINK entry.

## DLOAD

### Dynamic Load Combination or Superposition

Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries for frequency response or TLOAD1 or TLOAD2 entries for transient response.

#### Format:

1	2	3	4	5	6	7	8	9	10
DLOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-	*					

#### Examples:

DLOAD	17	1.0	2.0	6	-2.0	7	2.0	8	
	-2.0	9							

Describer	Meaning
SID	Load set identification number. See Remarks 1. and 4. (Integer > 0)
S	Scale factor. See Remarks 2. and 8. (Real)
Si	Scale factors. See Remarks 2., 7. and 8. (Real)
Li	Load set identification numbers of RLOAD1, RLOAD2, TLOAD1, TLOAD2, and ACSRCE entries. See Remarks 3. and 7. (Integer > 0)

#### Remarks:

- Dynamic load sets must be selected in the Case Control Section with DLOAD = SID.
- The load vector being defined by this entry is given by

$$\{P\} = S \sum_i S_i \{P_i\}$$

- Each Li must be unique from any other Li on the same entry.
- SID must be unique from all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries.
- Nonlinear transient load sets (NOLINi entries) may not be specified on DLOAD entries. NOLINi entries are selected separately in the Case Control Section by the NONLINEAR command.
- A DLOAD entry may not reference a set identification number defined by another DLOAD entry.
- The scale factor Si will apply to all dynamic load entries with load set identification number of Li.
- For RC network solver in thermal analysis, the S and Si factors are always taken as 1.
- If Modules are present then this entry may only be specified in the main Bulk Data section.

## DMI Direct Matrix Input

Defines matrix data blocks. Generates a matrix of the following form:

$$[\text{NAME}] = \begin{bmatrix} X_{11} & X_{12} & \dots & X_{1n} \\ X_{21} & X_{22} & \dots & X_{2n} \\ \cdot & \cdot & \cdot & \cdot \\ X_{m1} & \dots & \dots & X_{mn} \end{bmatrix}$$

where the elements  $X_{ij}$  may be real ( $X_{ij} = A_{ij}$ ) or complex ( $X_{ij} = A_{ij} + iB_{ij}$ ). The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

### Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	"0"	FORM	TIN	TOUT		M	N	

### Column Entry Format for Real Matrices:

DMI	NAME	J	I1	A(I1,J)	A(I1+1,J)		-etc.-	I2	
	A(I2,J)		-etc.-						

### Column Entry Format for Complex Matrices:

DMI	NAME	J	I1	A(I1,J)	B(I1,J)	A(I1+1,J)	B(I1+1,J)	-etc.-	
	I2	A(I2,J)	B(I2,J)	-etc.-					

### Example of a Real Matrix:

DMI	BBB	0	2	1	1		4	2	
DMI	BBB	1	1	1.	3.	5.			
DMI	BBB	2	2	6.	4	8.			

$$\text{BBB} = \begin{bmatrix} 1.0 & 0.0 \\ 3.0 & 6.0 \\ 5.0 & 0.0 \\ 0.0 & 8.0 \end{bmatrix}$$

**Example of a Complex Matrix:**

DMI	QQQ	0	2	3	3		4	2	
DMI	QQQ	1	1	1.0	2.0	3.0	0.0	3	
		5.0	6.0						
DMI	QQQ	2	2	6.0	7.0	4	8.0	9.0	

$$[QQQ] = \begin{bmatrix} 1.0 + 2.0i & , & 0.0 + 0.0i \\ 3.0 + 0.0i & , & 6.0 + 7.0i \\ 5.0 + 6.0i & , & 0.0 + 0.0i \\ 0.0 + 0.0i & , & 8.0 + 9.0i \end{bmatrix}$$

Describer	Meaning
NAME	Name of the matrix. See Remark 1. Name is used to reference the data block in the DMAP sequence. (One to eight alphanumeric characters, the first of which must be alphabetic.)
FORM	Form of matrix, as follows: (Integer) <ul style="list-style-type: none"> <li>1 Square matrix (not symmetric)</li> <li>2 General rectangular matrix</li> <li>3 Diagonal matrix (M=number of rows, N = 1)</li> <li>4 Lower triangular factor</li> <li>5 Upper triangular factor</li> <li>6 Symmetric matrix</li> <li>8 Identity matrix (M=number of rows, N = M)</li> </ul>
TIN	Type of matrix being input, as follows: (Integer) <ul style="list-style-type: none"> <li>1 Real, single precision (one field used/element)</li> <li>2 Real, double precision (one field used/element)</li> <li>3 Complex, single precision (two fields used/element)</li> <li>4 Complex, double precision (two fields used/element)</li> </ul>

Describer	Meaning
TOUT	Type of matrix being output, as follows: (Integer) 0 Set by precision cell 1 Real, single precision 2 Real, double precision 3 Complex, single precision 4 Complex, double precision
M	Number of rows in NAME. (Integer > 0)
N	Number of columns in NAME. Except for FORM 3 and 8. (Integer > 0)
"0"	Indicates the header entry.
J	Column number of NAME. (Integer > 0)
I1, I2, etc.	Row number of NAME, which indicates the beginning of a group of nonzero elements in the column. See Remark 13. (Integer > 0)
A(Ix,J)	Real part of element (see TIN). (Real)
B(Ix,J)	Imaginary part of element (see TIN). (Real)

**Remarks:**

1. In order to use the DMI feature, the user must write a DMAP, or make alterations to a solution sequence that includes the DMIIN module. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. The total number of DMIs and DTIs may not exceed 1000.
3. Field 3 of the header entry must contain an integer of zero (0).
4. For symmetric matrices, the entire matrix must be input.
5. Only nonzero terms need be entered.
6. Leading and trailing zeros in a column do not have to be entered. However, a blank field between nonzero fields on this entry is not equivalent to a zero. If a zero input is required, the appropriate type zero must be entered (i.e., 0.0 or 0.0D0).
7. Complex input must have both the real and imaginary parts entered if either part is nonzero; i.e., the zero component must be input explicitly.
8. If A(Ix,J) is followed by "THRU" in the next field and an integer row number "IX" after the THRU, then A(Ix,J) will be repeated in each row through IX. The "THRU" must follow an element value. For example, the entries for a real matrix RRR would appear as follows:

1	2	3	4	5	6	7	8	9	10
DMI	NAME	J	I1	A(I1,J)			I1	A(I2,J)	
DMI	RRR	1	2	1.0	THRU	10	12	2.0	

These entries will cause the first column of the matrix RRR to have a zero in row 1, the values 1.0 in rows 2 through 10, a zero in row 11, and 2.0 in row 12.

9. Each column must be a single logical entry. The terms in each column must be specified in increasing row number order.
10. The “FORM” options 4, 5, and 8 are nonstandard forms and may be used only in conjunction with the modules indicated in [Table 10](#).

**Table 10**      **DMI FORM Options**

FORM	Matrix Description	Modules			
		ADD	FBS	MATPRN	MPYAD
4	Lower Triangular Factor		X	X	
5	Upper Triangular Factor		X	X	
8	Identity	X	X	X	X

11. Form 3 matrices are converted to Form 6 matrices, which may be used by any module.
12. Form 7 matrices may not be defined on this entry.
13. I1 must be specified. I2, etc. are not required if their matrix elements follow the preceding element in the next row of the matrix. For example, in the column entry for column 1 of QQQ, neither I2 nor I3 is specified.
14. The DMIG entry is more convenient for matrices with rows and columns that are referenced by grid or scalar point degrees-of-freedom.
15. If Modules are present then this entry may only be specified in the main Bulk Data section.

## DMIAX

### Direct Matrix Input for Axisymmetric Analysis

Defines axisymmetric (fluid or structure) related direct input matrix terms.

The matrix is defined by a single header entry and one or more column entries. Only one header entry is required. A column entry is required for each column with nonzero elements.

#### Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIAX	NAME	"0"	IFO	TIN	TOUT				

#### Column Entry Format:

DMIAX	NAME	GJ	CJ	NJ					
	G1	C1	N1	A1	B1				
	G2	C2		-etc.-					

#### Example:

DMIAX	B2PP	0	1	3	4				
DMIAX	B2PP	32							
	1027	3		4.25+6	2.27+3				

Describer	Meaning
NAME	Name of the matrix. See Remark 2. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix: (Integer) <ul style="list-style-type: none"> <li>1            Square matrix</li> <li>2            General rectangular matrix</li> <li>6            Symmetric matrix</li> </ul>
TIN	Type of matrix being input: (Integer) <ul style="list-style-type: none"> <li>1            Real, single precision (One field is used per element.)</li> <li>3            Complex, single precision (Two fields are used per element.)</li> </ul>

Describer	Meaning
TOUT	Type of matrix that will be created: (Integer) 1           Real, single precision 2           Real, double precision 3           Complex, single precision 4           Complex, double precision
GJ, Gi	Grid, scalar, RINGFL fluid point, PRESPT pressure point, FREEPT free surface displacement, or extra point identification number. (Integer > 0)
CJ, Ci	Component number for GJ or Gi grid point (0 ≤ Integer ≤ 6; Blank or zero if GJ or Gi is a scalar, fluid, or extra point.)
NJ, Ni	Harmonic number of RINGFL point. Must be blank if a point type other than RINGFL is used. A negative number implies the “sine” series; a positive number implies the “cosine” series. (Integer)
Ai, Bi	Real and imaginary parts of matrix element; row (Gi, Ci, Ni) column (GJ, CJ, NJ). If the matrix is real (TIN = 1), then Bi must be blank.

1. This entry is allowed only if an AXIF entry is also present.
2. Matrices defined on this entry may be used in dynamics by selection with the Case Control commands K2PP = NAME, B2PP = NAME, or M2PP = NAME for  $[K_{pp}^2]$ ,  $[B_{pp}^2]$ , or  $[M_{pp}^2]$ , respectively. See [Superelement Analysis](#) in the *MSC Nastran Reference Guide*.
3. Field 3 or the header entry must contain an integer 0.
4. For symmetric matrices, either the upper or the lower triangle terms may be specified, but not both.
5. Only nonzero terms need be entered.
6. If any DMIAX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, DMIAX entry changes or additions are not recommended on restart.



## DMIG Direct Matrix Input at Points

Defines direct input matrices related to grid, extra, and/or scalar points. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

### Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

### Column Entry Format:

DMIG	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

### Example:

DMIG	STIF	0	1	3	4				
DMIG	STIF	27	1		2	3	3.+5	3.+3	
	2	4	2.5+10	0.	50		1.0	0.	

Describer	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) 1 Square 2 or 9 Rectangular 6 Symmetric
TIN	Type of matrix being input: (Integer) 1 Real, single precision (One field is used per element.) 2 Real, double precision (One field is used per element.) 3 Complex, single precision (Two fields are used per element.) 4 Complex, double precision (Two fields are used per element.)
TOUT	Type of matrix that will be created: (Integer) 0 Set by precision system cell (Default) 1 Real, single precision 2 Real, double precision 3 Complex, single precision

Describer	Meaning
	4            Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5. and 6. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

### Remarks:

- Matrices defined on this entry may be used in dynamics by selection in the Case Control with K2PP = NAME, B2PP = NAME, M2PP = NAME for  $[K_{\rho\rho}]$ ,  $[B_{\rho\rho}]$ , or  $[M_{\rho\rho}]$ , respectively. Matrices may also be selected for all solution sequences by K2GG = NAME, B2GG = NAME, and M2GG = NAME. The g-set matrices are added to the structural matrices before constraints are applied, while p-set matrices are added in dynamics after constraints are applied. Load matrices may be selected by P2G = NAME for dynamic and superelement analyses.
- The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column follows. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but input of an element of the matrix more than once will produce a fatal message.
- Field 3 of the header entry must contain an integer 0.
- For symmetric matrices (IFO = 6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both below and above the diagonal.
- The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIG matrices is always either p-set or g-set size, depending on the context.) The GJ term is used for the column index. The CJ term is ignored.
- If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).

- If  $IFO = 2$ , the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the g- or p-size matrix will be truncated.
7. The matrix names must be unique among all DMIGs.
  8. TIN should be set consistent with the number of decimal digits required to read the input data adequately. For a single-precision specification on a short-word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double-field format. If more digits are needed, a double precision specification should be used instead. However, note that a double precision specification requires a “D” type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
  9. On long-word machines, almost all matrix calculations are performed in single precision and on short-word machines, in double precision. It is recommended that DMIG matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIG input to be used on any machine. If TOUT is contrary to the machine type specified (for example, a TOUT of 1 on a short-word machine), unreliable results may occur.
  10. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

**DMIG,UACCEL**

## Direct Matrix Input of Enforced Static Acceleration

Defines rigid body accelerations in the basic coordinate system.

**Format:**

1	2	3	4	5	6	7	8	9	10
DMIG	UACCEL	"0"	"9"	TIN				NCOL	
DMIG	UACCEL	L			G1	C1	X1		
	G2	C2	X2		G3	C3	X3		

**Example:**

DMIG	UACCEL	0	9	1				4	
DMIG	UACCEL	2			2	3	386.4		
DMIG	UACCEL	3			2	4	3.0		
DMIG	UACCEL	4			2	6	1.0		

Descriptor	Meaning
TIN	Type of matrix being input. (Integer 1 or 2) 1        Real, single precision (One field is used per element.) 2        Real, double precision (One field is used per element.)
NCOL	Number of columns, see Remark 2. Default is the number of columns specified. (Integer > 0)
L	Load sequence number. (Integer > 0)
Gi	Grid point identification number of a single reference point. (Integer > 0)
Ci	Component number for Gi in the basic coordinate system. See Remark 4. (0 < Integer ≤ 6)
Xi	Value of enforced acceleration term in the basic coordinate system. (Real)

**Remarks:**

1. DMIG,UACCEL is an optional entry when PARAM,INREL,-1 is specified in SOLs 101 or 200. If DMIG,UACCEL is present, the loads applied to the structure are the sum of the conventional applied loads plus the inertia loads resulting from the rigid body accelerations defined on this entry. If it is not present, conventional inertia relief calculations are performed.
2. The load sequence number interpretation depends on the value of the NCOL field. The recommended method is to set it equal to the number of loading conditions. The load sequence number L is then the sequence number of the subcase to which the applied acceleration will be applied.

3. The grid point identification number listed on Gi defines a single grid point on the model where loads will be applied to cause the enforced acceleration state. Gi must also appear on a SUPORT Bulk Data entry. It must also appear on a PARAM,GRDPNT entry. In superelement analysis, it must be a residual structure point exterior to all superelements.
4. The Xi value is the enforced acceleration at grid point Gi. The translation and rotation components are in consistent units and will be applied in the basic coordinate system regardless of the displacement coordinate system specified for Gi (CD field on GRID entry).
5. Only nonzero terms need be entered.
6. See [Superelement Analysis](#) in *the MSC Nastran Reference Guide* for the theoretical basis of inertia relief with superelements.
7. If any DMIG entry is changed or added on restart then a complete re-analysis is performed. Therefore, DMIG entry changes or additions are not recommended on restart.

**DMIGOUT****DMIG Matrices to be Output from the Marc Portion of SOL 600**

Defines DMIG matrices to be output from the Marc Portion of SOL 600.

**Header Entry Format:**

1	2	3	4	5	6	7	8	9	10
DMIGOUT	ID	ISTIFF	IDIFF	IMASS	IDAMP	ICOND	ISPECIF	ISOL	
		ICTRL	IFREQ	ICORD	KIND	AMIN	IUSEK	IUSEM	
		IE1	THRU	IE2		IE3	THRU	IE4	

**Example:**

DMIGOUT	100	1	1	0	0	0	0	103	
		2	-1	2	2	1.0E-16			

Describer	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example if the Case Control contains SUBCASE 20, ID would be 20. To output matrices in Marc's phase zero, etc ID=0. (Integer; Default = 1, see Remark 11.)
ISTIFF	Flag to output stiffness matrices. (Normally contains the differential stiffness and all other stiffness terms.) (Integer; Default = 0) 0 Do not output the matrix 1 Output the matrix
IDIFF	Flag to output differential stiffness matrices. (Only available for buckling analysis or subcases.) (Integer; Default = 0) 0 Do not output the matrix 1 Output the matrix
IMASS	Flag to output mass matrices. (Integer; Default = 0) 0 Do not output the matrix 1 Output the matrix (must be a dynamic analysis)
IDAMP	Flag to output damping matrices. (Integer; Default = 0) 0 Do not output the matrix 1 Output the matrix (must be a dynamic analysis)
ICOND	Flag to output conductivity matrices. (Integer; Default = 0) 0 Do not output the matrix 1 Output the matrix
ISPECIF	Flag to output specific heat matrices. (Integer; Default = 0)

Describer	Meaning
	0 Do not output the matrix
	1 Output the matrix
ISOL	Solution sequence to run using the DMIG matrices. (Integer absolute value > 100; Default = 0, which means do not run any solution sequence using the DMIG's created by Marc in this execution). To speed up the solution, use DOMAINSOLVER ACMS (PARTOPT=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112).
ICTRL	Controls type of matrix. (Integer; Default = 2)
	1 Element matrices
	2 Global matrices
	3 Element and global matrices
IFREQ	Controls how often the matrices are output (Integer; no Default)
	1 Output at every increment
	2 Output every other increment, etc.
	3 Output every 3rd increment, etc.
	-1 Output only at start of the subcase. (See Remark 12.)
ICORD	Controls matrix output coordinate system (Integer; no Default)
	1 Nastran basic coordinate system
	2 Current transformed coordinate system (at the start of the run, this is the Nastran global coordinate system)
KIND	Controls which elements are written (Integer; Default = 2)
	1 A list of elements starting with the 3 <sup>rd</sup> entry will be specified
	2 All elements will be written
AMIN	Values below AMIN will not be written (Real; Default = 1.0E-15). Values below AMIN will be skipped.
IUSEK	Increment to use in SOL 600 CONTINUE option for the stiffness matrix (if CONTINUE is specified on the SOL 600 entry). (Integer; Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global stiffness matrix for increment 10 will have the name jid.marc_cglsti_0020.
IUSEM	Increment to use in SOL 600 CONTINUE option for the mass matrix (If CONTINUE is specified on the SOL 600 entry). (Integer, Default = -1) Increments 0 to 9999 may be specified. If -1 is specified, the last increment output by Marc will be used. As an example, the Marc DMIG file for the global mass matrix for increment 20 will have the name jid.marc_cglmas_0020.

Describer	Meaning
IE1, IE3	Starting element number of a range of elements (Integer; no Default) Only enter IE1 is KIND=1
IE2, IE4	Ending element number of a range of elements (Integer; no Default) Only enter IE2 is KIND=1 (IE2 is required if IE1 is entered).

**Remarks:**

1. The first continuation line is required.
2. The second continuation line should only be entered if KIND=1 and may be repeated as many times as necessary to define all applicable elements.
3. ICTRL, IFREQ, ICORD, KIND and AMIN apply to all types of matrices to be written for the subcase.
4. Only one DMIGOUT entry can be entered per subcase. If more than one is entered, only the first encountered will be used.
5. DMIGOUT entries may be made for each subcase desired.
6. Marc DMIG element output will be in files named jid.marc\_dmigXX\_inc where XX is shown below and inc is the increment number.

ST	Stiffness Matrices
DF	Differential stiffness matrices
MS	Mass matrices
DM	Damping
CO	Conductivity matrices
SP	Specific heat matrices

Global matrices are named jid\_marc\_cglsti\_inc.

7. If the SOL 600 CONTINUE options is invoked, Case Control commands and a Bulk Data include statement to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
8. ID may not be 600 or 700 in the Executive Control statement, SOL 600,ID.
9. The following Bulk Data parameters are usually required in addition to the DMIGOUT entry:  

```
$2345678x234567890123456x34567890123456
param* mrspawn2      nastcmd
pram,mrmtxnam,KAAX
param,marcfile,nastb.rc
```

 where



- a. `nastcmd` is the name of the command to run the primary and continuation jobs (examples are `nastran`, `nast2006t1`, `nast2006t2`, etc.)
  - b. `nastb.rc` should be changed to the name of the rc file to be used for the continuation run. It usually will specify `mem=` with a larger value than that of the primary run and also include a line `bat=no` (except for windows systems).
  - c. `PARAM,MARCFILi` should not be used.
10. For standard nonlinear static or dynamic analyses the stiffness matrix contains all contributions including the differential stiffness matrix and it is not possible to obtain the differential stiffness matrix separately. For a buckling analysis, the differential stiffness matrix may be obtained separately.
  11. To obtain stiffness and mass matrices for SOL 600,103, ID in field 2 of this entry must be must be zero.
  12. If IFREQ is 1 or a small number, the number of matrices output for dynamic analyses (SOL 600,109 or SOL 600,120) can be extremely large.
  13. If ID=0, IFREQ should be 1.
  14. Setting IFREQ to a value larger than the actual number of increments in a subcase will produce no matrices.

## DMIGROT

## DMIG Matrices for which large rotation in SOL 600

Defines large rotation and other characteristics of a matrix entered using DMIG in SOL 600.

### Format for K2GG and K2PP Stiffness DMIG Matrices:

1	2	3	4	5	6	7	8	9	10
DMIGROT	Name	G1	G2	G3	SCALE	ITRAN	IACT	IDEACT	

### Format for M2GG and M2PP Mass DMIG Matrices:

DMIGROT	Name				SCALE		IACT		
---------	------	--	--	--	-------	--	------	--	--

### Example:

DMIGROT	KAAX	375							
DMIGROT	KAAY	401	402	403					
DMIGROT	KAAX				1.25	1	2	5	
DMIGROT	MAAX				0.001		7	8	

Describer	Meaning
Name	Name of the DMIG matrix for which large rotation is specified. See Remark 1. (Character; Required field; no Default)
G1	Grid ID specifying the rotation of the matrix. See Remark 2. (Integer; Required field; no Default)
G2	Grid ID which combined with G1 and G3 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0; )
G3	Grid ID which combined with G1 and G2 specifies a plane whose rotation provides the rigid body rotation of the matrix. See Remark 2. (Integer; Default = 0)
SCALE	Scale factor. Each term in the DMIG matrix will be scaled by this value. (Real; Default = 1.0)
ITRAN	Flag indicating whether to transform the DMIG's or not. (Integer, Default = 0) 0 Do not transform the DMIG matrix. 1 Transform the DMIG using G1, G2, G3 to form the transformation matrix.
IACT	Subcase ID for which this DMIG matrix will be activated. See Remark 5. (Integer; Default = 0). 0 The matrix is active for all subcases. N The matrix is activated starting with subcase N
IDEACT	Subcase ID for which this DMIG matrix will be deactivated. See Remark 5. (Integer; no Default; leave blank if the matrix should be active fro the entire analysis or starting with subcase IACT)

**Remarks:**

1. All DMIG matrices with the name specified in field 2 will rotate with grids  $G_i$ .
2. If  $G_1$  has 6 dof it is not necessary to specify  $G_2$  or  $G_3$  and the DMIG matrix will rotate with the same rotation as  $G_1$ . If  $G_1$  has fewer than 6 dof, or if it is desired that the DMIG matrix rotate with a plane defined by  $G_1$ ,  $G_2$  and  $G_3$ ,  $G_2$  and  $G_3$  must be entered and may not all be collinear.
3. If a transformation is to be applied to the stiffness matrix, the DMIG **must** contain all of the degrees of freedom associated with the node to which the transformation is applied.
4. This bulk data entry maps to Marc's K2GG (and K2PP) model definition entry.
5. IACT and IDEACT are ID's of the Case Control SUBCASE entry.
6.  $G_1$ ,  $G_2$ ,  $G_3$  and ITRAN may not be entered for mass matrices.

## DMIJ

### Direct Matrix Input at js-Set of the Aerodynamic Mesh

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for CAERO1, CAERO3, CAERO4 and CAERO5 and for the slender body elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the interference elements of a CAERO2, use DMIJI or DMI.

#### Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIJ	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

#### Column Entry Format:

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

#### Example:

DMIJ	ALPH1	0	9	2	0			1	
DMIJ	ALPH1	1	1		1	1	.1		
	2	1	.1						

Descriptor	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer) 1            Square 9 or 2      Rectangular 6            Symmetric
TIN	Type of matrix being input: (Integer) 1            Real, single precision (One field is used per element) 2            Real, double precision (One field is used per element) 3            Complex, single precision (Two fields are used per element) 4            Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer) 0            Set by precision system cell (Default) 1            Real, single precision

Describer	Meaning
	2      Real, double precision
	3      Complex, single precision
	4      Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

**Remarks:**

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. Also, DMIJ may also be used for the W2GJ and FA2J entries. Again, a single column is required. If both DMI and DMIJ are specified for W2GJ or FA2J, the DMI entry will be used. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJ data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X		
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9, the number of columns in the matrix is NCOL. (The number of rows in all DMIJ matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIJ.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIJ matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIJ input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIJ entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIJ entry changes or additions are not recommended on restart.

**DMIJ**

**Direct Matrix Input at js-Set of the Interference Body**

Defines direct input matrices related to collation degrees-of-freedom (js-set) of aerodynamic mesh points for the interference elements of CAERO2. These include W2GJ, FA2J and input pressures and downwashes associated with AEPRESS and AEDW entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements. For entering data for the slender elements of a CAERO2, or a CAERO1, 3, 4 or 5 use DMIJ or DMI.

**Header Entry Format:**

1	2	3	4	5	6	7	8	9	10
DMIJ	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

**Column Entry Format:**

DMIJ	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

**Example:**

DMIJ	ALPH1	0	9	2	0			1	
DMIJ	ALPH1	1	1		1	1	.1		
	2	1	.1						

Describer	Meaning
NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix being input. (Integer) 1 Square 9 or 2 Rectangular 6 Symmetric
TIN	Type of matrix being input: (Integer) 1 Real, single precision (One field is used per element) 2 Real, double precision (One field is used per element) 3 Complex, single precision (Two fields are used per element) 4 Complex, double precision (Two fields are used per element)
TOUT	Type of matrix being created: (Integer) 0 Set by precision system cell (Default) 1 Real, single precision 2 Real, double precision



Describer	Meaning
	3          Complex, single precision
	4          Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ ≤ 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

**Remarks:**

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEDW and/or AEPRESS entries. In that paradigm, a single column is required. DMI may NOT be used for AEDW and AEPRESS.
2. The js-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIJI data will be partitioned to the j-set, not reduced. No warnings are issued about truncated data. The j-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO2-Y		X			
CAERO2-Z			X		
CAERO2-ZY		X	X		

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.
4. Field 3 of the header entry must contain an integer 0.



5. For symmetric matrices (very rare in the j-set!) (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. the number of columns in the matrix is NCOL. (The number of rows in all **DMJI** matrices is always the js-set size--the union of the j-set and the permanently SPC'd partition). The GJ term is used for the column index. the CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered nonnull column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all **DMJI**.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that **DMIJ** matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same **DMJI** input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any **DMIJ** entry is changed or added on restart then a complete reanalysis is performed. Therefore, **DMIJ** entry changes or additions are not recommended on restart.

## DMIK

### Direct Matrix Input at ks-Set of the Aerodynamic Mesh

Defines direct input matrices related to physical (displacement) degrees-of-freedom (ks-set) of aerodynamic grid points. These include WKK, WTFAC and input forces associated with AEFORCE entries. The matrix is described by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

#### Header Entry Format:

1	2	3	4	5	6	7	8	9	10
DMIK	NAME	"0"	I FO	TIN	TOU T	POLAR		NCOL	

#### Column Entry Format:

DMIK	NAME	GJ	CJ		G1	C1	A1	B1	
	G2	C2	A2	B2	-etc.-				

#### Example:

DMIK	ALPH1	0	9	2	0			1	
DMIK	ALPH1	1	1		1	1	1.0		
	2	1	1.0						

Descriptor	Meaning
------------	---------

NAME	Name of the matrix. See Remark 1. (One to eight alphanumeric characters, the first of which is alphabetic.)
I FO	Form of matrix being input. (Integer) 1 Square 9 or 2 Rectangular 6 Symmetric
TIN	Type of matrix being input: (Integer) 1 Real, single precision (One field is used per element) 2 Real, double precision (One field is used per element) 3 Complex, single precision (Two fields are used per element) 4 Complex, double precision (Two fields are used per element)
TOU T	Type of matrix being created: (Integer) 0 Set by precision system cell (Default) 1 Real, single precision 2 Real, double precision 3 Complex, single precision

Describer	Meaning
	4          Complex, double precision
POLAR	Input format of Ai, Bi. (Integer = blank or 0 indicates real, imaginary format. Integer > 0 indicated magnitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. (Integer > 0)
GJ	Grid, scalar or extra point identification number for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer ≤ 6; blank or zero if GJ is a scalar or extra point.)
Gi	Grid, scalar, or extra point identification number for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

**Remarks:**

1. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFAC T entries. Matrices defined on this entry are referenced in static aeroelastic analysis by reference on AEFORCE entries. In that paradigm, a single column is required. Also, DMIK may also be used for the WKK and WTFAC T entries. If both DMI and DMIK are specified for WKK or WTFAC T, the DMI entry will be used. DMI may NOT be used for AEFORCE.
2. The ks-set DOF's for each aerodynamic theory are limited to the six-DOF paradigm (3 translations and 3 rotations). However, particular DOF's are permanently SPC'd based on the theory's ability to support those degrees-of-freedom. Unlike the DMIG entry, DMIK data will be partitioned to the k-set, not reduced. No warnings are issued about truncated data. The k-set DOF's for each aerodynamic element/grid are highly method dependent. The following data define the known set, but the j-set definition is somewhat arbitrary in the general (external aerodynamics) paradigm.

Entry Type	COMP				
	1	2	3	5	6
CAERO1			X	X	
CAERO2-Y		X		X	
CAERO2-Z			X		X
CAERO2-ZY		X	X	X	X

3. The header entry containing IFO, TIN and TOUT is required. Each nonnull column is started with a GJ, CJ pair. The entries for each row of that column then follow. Only nonzero terms need be entered. The terms may be input in arbitrary order. A GJ, CJ pair may be entered more than once, but the input of an element of the matrix more than once results in a fatal message.

4. Field 3 of the header entry must contain an integer 0.
5. For symmetric matrices (IFO=6), a given off-diagonal element may be input either below or above the diagonal. While upper and lower triangle terms may be mixed, a fatal message will be issued if an element is input both above and below the diagonal.
6. The recommended format for rectangular matrices requires the use of NCOL and IFO = 9. The number of columns in the matrix is NCOL. (The number of rows in all DMIK matrices is always the ks-set size--the union of the k-set and the permanently SPC'd partition). The GJ term is used for the column index. The CJ term is ignored.
7. If NCOL is not used for rectangular matrices, two different conventions are available:
  - If IFO = 9, GJ and CJ will determine the sorted sequence, but will otherwise be ignored; a rectangular matrix will be generated with the columns submitted being in the 1 to N positions, where N is the number of logical entries submitted (not counting the header entry).
  - If IFO = 2, the number of columns of the rectangular matrix will be equal to the index of the highest numbered non-null column (in internal sort). Trailing null columns of the js-size matrix will be truncated.
8. The matrix names must be unique among all DMIK.
9. TIN should be consistent with the number of decimal digits required to read the input data adequately. For a single precision specification on a short word machine, the input will be truncated after about eight decimal digits, even when more digits are present in a double field format. If more digits are needed, a double precision specification should be used instead. However, not that a double precision specification requires a "D" type exponent even for terms that do not need an exponent. For example, unity may be input as 1.0 in single precision, but the longer form 1.0D0 is required for double precision.
10. On long word machines, almost all matrix calculations are performed in single precision and, on short word machines, in double precision. It is recommended that DMIK matrices also follow these conventions for a balance of efficiency and reliability. The recommended value for TOUT is 0, which instructs the program to inspect the system cell that measures the machine precision at run time and sets the precision of the matrix to the same value. TOUT = 0 allows the same DMIK input to be used on any machine. If TOUT is contrary to the machine type specified, unreliable results may occur.
11. If any DMIK entry is changed or added on restart then a complete reanalysis is performed. Therefore, DMIK entry changes or additions are not recommended on restart.

## DOPTPRM

## Design Optimization Parameters

Overrides default values of parameters used in design optimization.

### Format:

1	2	3	4	5	6	7	8	9	10
DOPTPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	PARAM4	VAL4	
	PARAM5	VAL5	-etc.-						

### Example:

DOPTPRM	IPRINT	5	DESMAX	10					
---------	--------	---	--------	----	--	--	--	--	--

Describer	Meaning
PARAMi	Name of the design optimization parameter. Allowable names are given in <a href="#">Table 11</a> . (Character)
VALi	Value of the parameter. (Real or Integer, see <a href="#">Table 11</a> .)

Table 11 PARAMi Names and Descriptions

Name	Description, Type, and Default Value
APRCOD	Approximation method to be used. (Integer 1, 2, or 3; Default = 2)
	1 Direct Linearization. APRCOD = 1 is recommended for shape, topology and topography optimization problems.
	2 Mixed Method based on response type
	3 Convex Linearization
AUTOSE	Flag to request an AESO job. AUTOSE = 1 activates an AESO creation run. (Integer $\geq 0$ ; Default = 0)
CONV1	Relative criterion to detect convergence. If the relative change in objective between two optimization cycles is less than CONV1, then optimization is terminated. Topology optimization can be terminated with $CONV1 < 5.00E-3$ (Real $> 0.0$ ; Default = 0.001).
CONV2	Absolute criterion to detect convergence. If the absolute change in objective between two optimization cycles is less than CONV2, then optimization is terminated. (Real $> 0.0$ ; Default = $1.0E-20$ )
CONVDV	Relative convergence criterion on design variables. (Real $> 0.0$ ; Default = 0.001 for non-topology; Default = 0.0001 for topology optimization)
CONVPR	Relative convergence criterion on properties. (Real $> 0.0$ ; Default = 0.001)

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
CT	Constraint tolerance. Constraint is considered active if current value is greater than CT. (Real < 0.0; Default = -0.03)	
CTMIN	Constraint is considered violated if current value is greater than CTMIN. (Real > 0.0; Default = 0.003)	
DABOBJ	Maximum absolute change in objective between two consecutive iterations to indicate convergence at optimizer level. F0 is the initial objective function value. (Real > 0.0; Default = MAX[0.001*ABS(F0), 0.0001])	
DELB	Relative finite difference move parameter. (Real > 0.0; Default = 0.0001; 0.001 for fatigue responses)	
DELOBJ	Maximum relative change in objective between two consecutive iterations to indicate convergence at optimizer level. (Real > 0.0; Default = 0.001)	
DELP	Fractional change allowed in each property during any optimization design cycle. This provides constraints on property moves. (Real, 0.0<DELP<1.0, default=0.2)	
DELX	Fractional change allowed in each design variable during any optimization cycle. (Real > 0.0; Default = .5 for sizing/shape/topometry optimization; Default = 0.2 for topology and topography optimization)	
DELXESL	Fractional change allowed in each design variable during the ESLNRO loop. (Real > 0.0, Default = 0.5)	
DESMAX	Maximum number of design cycles (not including FSD cycle) to be performed. (Integer ≥ 0; Default = 5 for sizing/shape optimization; Default = 30 for topology, topography, and topometry optimization)	
DISCOD	Discrete Processing Method: (Integer 0, 1, 2, 3 or 4; Default = 1)	
	0	No Discrete Optimization
	1	Design of Experiments
	2	Conservative Discrete Design
	3	Round up to the nearest design variable
4	Round off to the nearest design variable	
DISBEG	Design cycle ID for discrete variable processing initiation. Discrete variable processing analysis is carried out for every design cycle after DISBEG. (Integer ≥ 0; Default = 0=the last design cycle)	
DPMAX	Maximum fraction of change on designed property (Default = 0.5), used by Trust Region Method.	
DPMIN	Minimum move limit imposed. (Real > 0.0; Default = 0.01)	
DRATIO	Theshold value that can be used to turn off an active AESO job. An AESO job is terminated if the ratio of the size of a design model to that of an analysis model is greater than DRATIO. (Real > 0.; Default = 0.1)	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
DSMXESL	Maximum number of design cycles applied to the ESLNRO loop. (Integer $\geq 0$ ; Default = 20)	
DXMAX	Maximum fraction of change on design variable (Default = 1.0), used by Trust Region Method.	
DXMIN	Minimum design variable move limit. (Real $> 0.0$ ; Default = 0.05 for sizing/shape/topometry optimization; Default = 1.0E-5 for topology and topography optimization)	
ETA1 ( $\eta_1$ )	the cutting ratio 1 (Default = 0.01), used by Trust Region Method.	
ETA2 ( $\eta_2$ )	the cutting ratio 2 (Default = 0.25), used by Trust Region Method.	
ETA3 ( $\eta_3$ )	the cutting ratio 3 (Default = 0.7), used by Trust Region Method.	
FSDALP	Relaxation parameter applied in Fully Stressed Design. (Real, $0.0 < \text{FSDALP} \leq 1.0$ ; Default = 0.9)	
FSDMAX	Specifies the number of Fully Stressed Design Cycles that are to be performed. (Integer; Default = 0)	
GMAX	Maximum constraint violation allowed at the converged optimum. (Real $> 0.0$ ; Default = 0.005)	
GSCAL	Constraint normalization factor. See Remarks under the DSCREEN and DCONSTR entries. (Real $> 0.0$ ; Default = 0.001)	
IGMAX	If IGMAX = 0, only gradients of active and violated constraints are calculated. If IGMAX $> 0$ , up to IGMAX gradients are calculated including active, violated, and near active constraints. For many constraint problems, a reasonable value of IGMAX = 2*NDV (number of design variables). A smaller IGMAX has a faster performance of MSCADS. (Integer $> 0$ ; Default = 0)	
IPRINT	Print control during approximate optimization phase. Increasing values represent increasing levels of optimizer information. ( $0 \leq \text{Integer} \leq 7$ ; Default = 0)	
	0	No output (Default)
	1	Internal optimization parameters, initial information, and results
	2	Same, plus objective function and design variables at each iterations
	3	Same, plus constraint values and identification of critical constraints
	4	Same, plus gradients
	5	Same, plus search direction
	6	Same, plus scaling factors and miscellaneous search information
7	Same, plus one dimensional search information	
ISCAL	Design variables scaling. (Integer $\geq -1$ ; Default = 0) = -1 no scaling; $\geq 0$ scaling	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
METHOD	Optimization Method: (Integer $\geq 0$ ; Default = 0)	
	0	Automatic selection for a better performance based on number of design variables, number of constraints, number of active/violated constraints and computer memory.
	1	Modified Method of Feasible Directions for MSCADS.
	2	Sequential Linear Programming for MSCADS
	3	Sequential Quadratic Programming for MSCADS
	4	SUMT method for MSCADS
	IJK	See Remark 3.
NASPRO	First cycle analysis output control. (Integer 0 or 1)	
	0	Print analysis output of first cycle. (Default)
	1	Do NOT print analysis output of first cycle.
OBJMOD	Objective function modification. (Integer; Default = 0)	
	0	Objective function will not be modified.
	1	Objective function will be reset to 0.0. Subsequently, printed objective function value represents the change of objective function.
OPTCOD	OPTCOD. See Remark 2. (Character; Default = Blank)	
	Blank	Taken from system cell number 413
	"MSCADS"	MSCADS is used
	"IPOPT"	IPOPT is used
	"External Optimizer SCA Service Identifier"	
	Must be defined in the CONNECT Service statement (see <i>MSC Nastran User Defined Services User's Guide</i> )	
P1	Print control items specified for P2. (Integer $\geq 0$ ; Default = 0) Initial results are always printed prior to the first approximate optimization. If an optimization task is performed, final results are always printed for the final analysis unless PARAM,SOFTEXIT,YES is specified. These two sets of print are not controllable.	
	n	Print at every n-th design cycle.



Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
P2	Items to be printed according to P1: (Integer; Default = 1)	
	0	No print.
	1	Print objective and design variables. (Default for sizing/shape optimization) Print objective. (Default for topology optimization) P2 ≥ 13 Print design variables for topology, topography, and topometry optimization
	2	Print properties.
	4	Print constraints.
	8	Print responses.
	16	Print weight as a function of a material ID (note that this is not a design quantity so that only inputs to the approximate design are available).
	n	Sum of desired items. For example, P2 = 10 means print properties and responses.
P2CALL	Maximum number of retained constraints of all categories to be printed per category. This single parameter can be used in place of the individual parameters P2CBL, P2CC, P2CDDV, P2CM, P2CP and P2CR. If any of these six parameters are non-zero, the P2CALL value is overridden for that constraint type. (Integer > 0; default is to print all retained constraints.)	
P2CBL	Maximum number of constraints on beam library dimensions to be printed. (Integer ≥ 0; default is to print all beam library constraints.)	
P2CC	Maximum number of constraints on connectivity properties to be printed. (Integer ≥ 0; default is to print all connectivity property constraints.)	
P2CDDV	Maximum number of constraints on dependent design variables to be printed. (Integer ≥ 0; default is to print all dependent design variable constraints.)	
P2CM	Maximum number of constraints on material properties to be printed. (Integer ≥ 0; default is to print all material property constraints.)	
P2CP	Maximum number of constraints on element properties to be printed. (Integer ≥ 0; default is to print all element property constraints.)	
P2CR	Maximum number of constraints on design responses to be printed. (Integer ≥ 0; default is to print all retained design response constraints.)	
P2RSET	ID of a SET1 Bulk Data entry to identify the set of retained responses (DRESP1, DRESP2 and/or DRESP3) to be printed. (Integer; Default is to print all responses associated with printed constraints. If P2CR is > 0, the set associated P2RSET > 0 will be printed independent of the responses associated with the printed constraint. If P2CR > 0 and PR2SET = -1, all retained responses will be printed.	

Table 11 PARAMi Names and Descriptions (continued)

Name	Description, Type, and Default Value	
PENAL	Penalty parameter used to transform an infeasible approximate optimization task to a feasible one. Setting this parameter to to; e.g., 2.0 may improve optimizer performance when the starting design is infeasible. (Real; Default = 0.0)	
PLVIOL	Flag for handling of property limit violation. By default, the job will terminate with a user fatal message if the property derived from design model (DVPRELi, DVMRELi, DVCRELi) exceeds the property limits. Setting PLVIOL to a non-zero number will cause the program to issue a user warning message by ignoring the property limits violation and proceed with the analysis. (Integer; Default = 0)	
PTOL	Maximum tolerance on differences allowed between the property values on property entries and the property values calculated from the design variable values on the DESVAR entry (through DVPRELi relations). PTOL is provided to trap ill-posed design models. (The minimum tolerance may be specified on user parameter DPEPS. See <a href="#">Parameters</a> ) (Real > 0.0; Default = 1.0E+35)	
STPSCL	Scaling factor for shape finite difference step sizes, to be applied to all shape design variables. (Real > 0.0; Default = 1.0)	
TCHECK	Topology Checkerboarding/minimum member size control option. (Integer $\geq$ -1)	
	1	Filtering algorithm
	2	Density constraint
	0	No control
-1	Automatic selection of filtering or density constraint algorithm for a better performance. (Default)	
TDMIN	Minimum diameter of members in topology optimization. This option is applied on 2 and 3D elements only. (Real > 0.0)	
TREGION	Flag to invoke Trust Region method.	
	0	Don't employ trust regions (Default)
	1	Turn Trust Region on
UPDFAC1	Updating factor 1 (Default = 2.0), used by Trust Region Method.	
UPDFAC2	Updating factor 2 (Default = 0.5), used by Trust Region Method.	

**Remarks:**

1. Only one DOPTPRM entry is allowed in the Bulk Data Section. All defaults recommended.
2. OPTCOD specifies which optimization code to be used in SOL 200 and METHOD specifies which optimization method to be used. The default is recommended.
3. METHOD = IJK enables a user selectable optimization strategy as documented in Vanderplaats, G. N., ADS -- A Fortran Program for Automated Design Synthesis -- Version 1.10, NASA CR 177985, 1985. The default is recommended.

The I selects one of ten available strategy options:

0	None -- Go directly to the optimizer
1	Sequential unconstrained minimization using the exterior penalty function method
2	Sequential unconstrained minimization using the linear extended interior penalty function method
3	Sequential unconstrained minimization using the quadratic extended interior penalty function method
4	Sequential unconstrained minimization using the cubic extended interior penalty function method
5	Augmented Lagrange multiplier method
6	Sequential linear programming
7	Method of centers
8	Sequential quadratic programming
9	Sequential convex programming

The J selects one of five available optimizer options:

1	Fletcher-Reeves algorithm for unconstrained minimization
2	Davidon-Fletcher-Powell (DFP) variable metric method for unconstrained minimization
3	Broydon-Fletcher-Goldfarb-Shanno (BFGS) variable metric method for unconstrained minimization
4	Method of feasible directions for constrained minimization
5	Modified method of feasible directions for constrained minimization

And K selects one of eight available one-dimensional search strategies:

1	Find the minimum of an unconstrained function using the Golden Section method
2	Find the minimum of an unconstrained function using the Golden Section method followed by polynomial interpolation
3	Find the minimum of an unconstrained function by first finding bounds and then using the Golden Section method followed by polynomial interpolation
4	Find the minimum of an unconstrained function by polynomial interpolation/extrapolation without first finding bounds on the solution

5	Find the minimum of a constrained function using the Golden Section method
6	Find the minimum of a constrained function using the Golden Section method followed by polynomial interpolation
7	Find the minimum of a constrained function by first finding bounds and then using polynomial interpolation
8	Find the minimum of a constrained function by polynomial interpolation/extrapolation without first finding bounds on the solution

## DPHASE      Dynamic Load Phase Lead

Defines the phase lead term  $\theta$  in the equation of the dynamic loading function.

### Format:

1	2	3	4	5	6	7	8	9	10
DPHASE	SID	P1	C1	TH1	P2	C2	TH2		

### Example:

DPHASE	4	21	6	2.1	8	6	7.2		
--------	---	----	---	-----	---	---	-----	--	--

Describer	Meaning
SID	Identification number of DPHASE entry. (Integer > 0)
Pi	Grid, extra, or scalar point identification number. (Integer > 0)
Ci	Component number. (Integers 1 through 6 for grid points; zero or blank for extra or scalar points)
THi	Phase lead $\theta$ in degrees. (Real)

### Remarks:

1. One or two dynamic load phase lead terms may be defined on a single entry.
2. SID must be referenced on a RLOADi entry. Refer to the RLOAD1 or RLOAD2 entry for the formulas that define how the phase lead  $\theta$  is used.
3. A DAREA, LSEQ or static load entry should be used to define a load at Pi and Ci.
4. In superelement analysis, DPHASE entries may only be applied to loads on points in the residual structure.
5. RC network solver does not support DPHASE for thermal analysis.

## DRESP1

## Design Sensitivity Response Quantities

Defines a set of structural responses that is used in the design either as constraints or as an objective.

### Format:

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
	ATT2	-etc.-							

### Example:

DRESP1	1	DX1	STRESS	PROD	2	3		102	
	103								

DRESP1	10	BMWV	WMPID	PSHELL		10		10	
	20	30							

Describer	Meaning
ID	Unique entry identifier. (Integer > 0)
LABEL	User-defined label. (Character, no default)
RTYPE	Response type. See <a href="#">Table 12</a> . (Character)
PTYPE	Element flag (PTYPE = "ELEM") or property entry name. Used with element type responses (stress, strain, force, etc.) to identify the property type, since property entry IDs are not unique across property types. (Character: "ELEM", "PBAR", "PSHELL", etc.)
REGION	Region identifier for constraint screening. See <a href="#">Remark 10</a> for defaults. (Integer > 0)
ATTA, ATTB, ATTi	Response attributes. See <a href="#">Table 12</a> . (Integer > 0 or Real or blank)

Table 12      Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
WEIGHT	Row Number ( $1 \leq \text{ROW} \leq 6$ ) See Remark 23.	Column Number ( $1 \leq \text{COL} \leq 6$ )	SEID <sub>i</sub> or All or blank. See Remark 12.
VOLUME	Blank	Blank	SEID <sub>i</sub> or ALL or blank. See Remark 12.
FRMASS See Remarks 27. & 28.	Blank	Blank	Blank or Property ID (PID). See Remark 36.
COMP See Remark 27.	Blank	Blank	Blank
EIGN	Normal Modes Mode Number. See Remark 32.	Approximation Code. See Remark 19.	Blank
CEIG	Complex Eigenvalue Mode Number. (Integer > 0)	ALPHA or OMEGA (Default = ALPHA)	Blank
FREQ	Normal Modes Mode Number. See Remarks 18. and 32.	Approximation Code. See Remark 19.	Blank
LAMA	Buckling Mode Number	Approximation Code. See Remark 19.	Blank
DISP	Displacement Component	Blank or Mode Number	Grid ID
STRAIN	Strain Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
ESE	Strain Energy Item Code See Remark 20.	Blank or Mode Number	Property ID (PID) or Element ID (EID)
STRESS	Stress Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
FORCE	Force Item Code	Blank or Mode Number	Property ID (PID) or Element ID (EID)
FATIGUE (pseudo-static) See Remark 38.	Fatigue Item Code. See Remark 42.	ID of a FATIGUE case control. See Remark 39.	Property ID (PID) or Element ID (EID)

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
FRFTG (frequency response - random vibration fatigue) See Remark 38.	Fatigue Item Code. See Remark 42.	ID of a FATIGUE case control. See Remark 39.	Property ID (PID) or Element ID (EID)
SPCFORCE	SPC Force Component	Blank	Grid ID
CSTRAIN See Remark 2.	Strain Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRESS See Remark 2.	Stress Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CFailure See Remark 2.	Failure Indices Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
CSTRAT See Remark 2.	Composite Strength Ratio Item Code	LAMINA Number or GPLYIDi (Integer; Default = 1)	Property ID (PID) or Element ID (EID)
TOTSE (Total Strain Energy)	Blank	Blank or Mode Number	SEIDi or All or blank. See Remark 12.
GPFORCE	GPFORCE Component Code (1-6; see Remark 24.)	Blank	Element ID
GPFORCP	Grid Point (see Remark 25.)	Blank	Grid ID connected to ATTA grid to specify orientation.
ABSTRESS	Arbitrary Beam Stress Item Code (see Remark 29.)	Blank	Property ID (PID) or Element ID (EID)
FRDISP	Displacement Component	Frequency Value. (Blank; Real $\geq 0.0$ or Character) See Remarks 15. and 20.	Grid ID
PRES	Acoustic Pressure Component (= 1 or 7)	Frequency Value. (Blank; Real $\geq 0.0$ or Character) See Remarks 15. and 20.	Grid ID



Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
FRVELO	Velocity Component	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRACCL	Acceleration Component	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRSPCF	SPC Force Component	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID
FRSTRE	Stress Item Code	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Property ID (PID) or Element ID (EID)
FRFORC	Force Item Code	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Property ID (PID) or Element ID (EID)
PSDDISP	Displacement Component (see Remarks 26. and 30.)	Frequency Value. (Blank; Real $\geq$ 0.0 or Character). See Remarks 15. and 20.	Grid ID
PSDVELO	Velocity Component (see Remarks 26. and 30.)	Frequency Value (Blank; Real $\geq$ 0.0 or Character). See Remarks 15. and 20.	Grid ID
PSDACCL	Acceleration Component (see Remarks 26. and 30.)	Frequency Value. (Blank; Real $\geq$ 0.0 or Character). See Remarks 15. and 20.	Grid ID

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
RMSDISP	Displacement Component (see Remark 30.)	RANDPS ID	Grid ID
RMSVELO	Velocity Component (see Remark 30.)	RANDPS ID	Grid ID
RMSACCL	Acceleration Component (see Remark 30.)	RANDPS ID	Grid ID
ACPWR (Acoustic Power radiated through a panel) See Remark 33.	Blank	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Blank
ACINTS (Acoustic Intensity)	Blank	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID of wetted surface.
AFPRES (Acoustic Pressure for AFPM) See Remark 34.	Acoustic Pressure Component (Integer = 1 or 7)	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID of AFPMID.
AFINTS (Acoustic Intensity for AFPM) See Remark 34.	Component Code - 0 - normal to AFPM, 1 - x-dir 2 - y-dir 3 - z-dir	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID of AFPMID.
AFVELO (Velocity for AFPM) See Remark 34.	Component Code - 11 - Real/Mag in x-dir 12 - Real/Mag in y-dir 13 - Real/Mag in z-dir 71 - Img/Ph in x-dir 72 - Img/Ph in y-dir 73 - Img/Ph in z-dir	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID of AFPMID.
AFPWR (Acoustic Power for AFPM) See Remark 34.	Blank	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Blank

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
DYSTIFF	Dynamic Stiffness Component	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Grid ID
ERP See Remarks 40. and 41.	ERP Item Code	Frequency Value. (Blank; Real $\geq$ 0.0 or Character) See Remarks 15. and 20.	Set3 ID or Blank
TDISP	Displacement Component	Time Value. (Blank; Real; or Character) See Remarks 16. and	Grid ID
TVELO	Velocity Component	Time Value. (Blank; Real, or Character) See Remarks 16. and	Grid ID
TACCL	Acceleration Component	Time Value. (Blank; Real, or Character) See Remarks 16. and	Grid ID
TSPCF	SPC Force Component	Time Value. (Blank; Real, or Character) See Remarks 16. and	Grid ID
TSTRE	Stress Item Code	Time Value. (Blank; Real, or Character) See Remarks 16. and	Property ID (PID) or Element ID (EID)
TFORC	Force Item Code	Time Value. (Blank; Real, or Character) See Remarks 16. and	Property ID (PID) or Element ID (EID)
STMONP1 Structural MONPNT1	Component See Remark 35.	Blank	Blank
STMOND1 Structural MONDSP1	Component See Remark 35.	Blank	Blank
MONPNT3	Component See Remark 35.	Blank	Blank

Table 12 Design Sensitivity Response Attributes

Response Type (RTYPE)	Response Attributes		
	ATTA (Integer > 0)	ATTB (Integer > 0 or Real > 0.0)	ATTI (Integer > 0)
AEMONP1 Aerodynamic MONPNT1	Component See Remark 35.	Blank	Blank
AEMOND1 Aerodynamic MONDSP1	Component See Remark 35.	Blank	Blank
TRIM	AESTAT or AESURF entry ID	Blank	Blank
STABDER	AESTAT or AESURF entry ID	Restraint Flag. (Integer 0 or 1) See Remark 13.	Component
FLUTTER	Blank	Blank	See Remark 14.
DIVERG	Divergence Root Number See remark 37.	Blank	Mach No.
WMPID See Remarks 43. and 44.	MID	SEID	PID

**Remarks:**

1. Stress, strain, force, fatigue and ERP item codes can be found in [Item Codes](#). For stress or strain item codes that have dual meanings, such as von Mises or maximum shear, the option specified in the Case Control Section will be used; i.e., STRESS(VONM) or STRESS(MAXS).
2. RTYPE = “CSTRESS”, “CSTRAIN”, “CFAILURE”, and “CSTRAT” are used only with the PCOMP/PCOMPG entry. “CSTRESS” and “CSTRAIN” item codes are described under Table 1 (Element Stress/Strain Item Codes) in [Item Codes](#). “CFAILURE” and “CSTRAT” item codes are described under Table 2 (Element Force Item Codes) in [Item Codes](#). Only force item codes that refer to failure indices of direct stress and interlaminar shear stress are valid.

The CFAILURE and CSTRAT response types requires the following specifications on the applicable entries:

- Failure theory in the FT field on PCOMP/PCOMPG entry.
  - Allowable bonding shear stress in the SB field on PCOMP/PCOMPG entry.
  - Stress limits in the ST, SC, and SS fields on all MATi entries.
  - When the PCOMPG is invoked, there is no default for the ATTB field.
3. ATTB can be used for responses of weight, composite laminae, dynamics, real and complex eigenvalues, WMPID and stability derivatives. For eigenvector responses, such as DISP, the ATTB identifies the mode of interest. For other responses, this field must be blank.

4. All grids associated with a DRESP1 entry are considered to be in the same region for screening purposes. Only up to NSTR displacement constraints (see DSCREEN entry) per group per load case will be retained in the design optimization phase.
5. DRESP1 identification numbers must be unique with respect to DRESP2 identification numbers.
6. If PTYPE = "ELEM", the ATT<sub>i</sub> correspond to element identification numbers.
7. If RTYPE = "DISP", "SPCFORCE", "GPFORCE", "TDISP", "TVELO", "TACCL" or "TSPCF", multiple component numbers (any unique combination of the digits 1 through 6 with no embedded blanks) may be specified on a single entry. Multiple response components may not be used on any other response types.
8. If RTYPE = "FRDISP", "FRVELO", "FRACCL", or "FRSPCF" only one component number may be specified in the ATTA field. Numbers 1 through 6 correspond to real (or magnitude) components and 7 through 12 imaginary (or phase) components. If more than one component for the same grid is desired, then a separate entry is required.
9. Real/imaginary representation is the default for complex response types. Magnitude/phase representation must be requested by the corresponding Case Control command; e.g., DISP(PHASE) = ALL for FRDISP type responses.
10. REGION is used for constraint screening. The NSTR field on DSCREEN entries gives the maximum number of constraints retained for each region per load case.  
If RTYPE = "WEIGHT", "VOLUME", "LAMA", "EIGN", "FREQ", "CEIG", "TOTSE", "RMSDISP", "RMSVELO", "RMSACCL", no REGION identification number should be specified. If the region field is left blank for a grid response, one region is created for each DRESP1 ID. If the region field is left blank for an element response, one region is created for each property ID invoked. Usually, the default value is appropriate.  
If the REGION field is not blank, all the responses on this entry as well as all responses on other DRESP1 entries that have the same RTYPE and REGION identification number will be grouped into the same region.
11. REGION is valid only among the same type of responses. Responses of different types will never be grouped into the same region, even if they are assigned the same REGION identification number by the user.
12. If RTYPE = "WEIGHT", "VOLUME", or "TOTSE" field ATT<sub>i</sub> = "ALL" implies total weight/volume/total strain energy of all superelements except external superelements, 0 implies residual only and i implies SEID=i. Default="ALL". RTYPE="TOTSE" is not supported for shape optimization.
13. RTYPE = "STABDER" identifies a stability derivative response. ATTB is the restraint flag for the stability derivative. ATTB = 0 means unrestrained, and ATTB = 1 means restrained. For example, ATTA = 4000, ATTB = 0, and ATT1 = 3 reference the unrestrained C<sub>z</sub> derivative for the AESTAT (or AESURF) entry ID = 4000.
14. RTYPE = "FLUTTER" identifies a set of damping responses. The set is specified by ATT<sub>i</sub>:  
ATT1 = Identification number of a SET1 entry that specifies a set of modes.  
ATT2 = Identification number of an FLFACT entry that specifies a list of densities.

ATT3 = Identification number of an FLFACT entry that specifies a list of Mach numbers.

ATT4 = Identification number of an FLFACT entry that specifies a list of velocities.

If the flutter analysis is type PKNL, it is necessary to put PKNL in the PTYPE field of this entry.

15. For RTYPE = “FRXXXX”, “PSDXXXX”, “PRES” and “ERP” a real value for ATTB specifies a frequency value in cycles per unit time. If a real ATTB value is specified, then the responses are evaluated at the closest excitation frequency. The default for ATTB is all excitation frequencies. See Remark for additional ATTB options. The OFREQ Case Control command has no effect on the selection of the frequencies.
16. For RTYPE = “TDISP”, “TVELO”, “TACCL”, “TSPCF”, “TFORC”, and “TSTRE”, ATTB specifies a time value. If ATTB is specified, then the responses are evaluated at the closest time selected by the OTIME command. The default for ATTB is all time steps selected by the OTIME command.
17. Intermediate station responses on CBAR elements due to PLOAD1 and/or CBARAO entries may not be defined on the DRESP1 entry.
18. RTYPE = “EIGN” refers to normal modes response in terms of eigenvalue (radian/time)\*\*2 while RTYPE = “FREQ” refers to normal modes response in terms of natural frequency or units of cycles per unit time.
19. For RTYPE = LAMA, EIGN or FREQ, the response approximation used for optimization can be individually selected using the ATTB field when APRCOD = 1 is being used.

For RTYPE = LAMA, ATTB = blank or 1 selects direct linearization, ATTB = 2 = inverse linearization.

For RTYPE = EIGEN or FREQ, ATTB = blank = Rayleigh Quotient Approximation, = 1 = direct linearization, = 2 = inverse approximation.

The default Rayleigh Quotient Approximation should be preferred in most cases.

Character input for ATTB is available for RTYPE of “FRXXXX”, “PSDXXXX”, “TXXXX”, “PRES”, “ERP”, “ACPWR”, “ACINTS”, “AFPRES”, “AFINTS”, “AFVELO”, “AFPWR” and “DYSTIFF”. The character input represents a mathematical function and the options for character input are SUM, AVG, SSQ, RSS, MAX and MIN. The expression of mathematical function is shown as follows:

$$\text{SUM}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i$$

$$\text{AVG}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i/n$$

$$\text{SSQ}(X_1, X_2, \dots, X_n) = \sum_{i=1}^n X_i^2$$

$$RSS(X_1, X_2, \dots, X_n) = \sqrt{\sum_{i=1}^n X_i^2}$$

MAX( $X_1, X_2, \dots, X_n$ ) = Maximum value among  $X_i$ ( $i=1$  to  $n$ )

MIN( $X_1, X_2, \dots, X_n$ ) = Minimum value among  $X_i$ ( $i=1$  to  $n$ )

where  $X_i$  is the response for a forcing frequency or time step. For example

DRESP1,10,DX1,FRSTRE,ELEM,,3,AVG,10

yields a response which is equal to the average stress for element 10 across all forcing frequencies.

**CAUTION!**

1. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP2" flag, the results of the mathematical function are used to evaluate the DRESP2.
2. If the ID of a DRESP1 with ATTB character input is referenced by a DRESP2 using the "DRESP1" flag, the individual values of the DRESP1 are used to compute individual DRESP2 values and the mathematical function is not used by the DRESP2.

20. Element strain energy item codes can be found under [Table 7-12](#) in [Item Codes](#). Only element strain energy and element strain energy density can be referenced on a DRESP1 entry. RTYPE="ESE" is not supported for shape optimization.
21. For RTYPE=RMSDISP, RMSVELO, or RMSACCL the ATTB specifies the appropriate RANDPS ID.
22. Input other than 1 or 7 of ATTA field, acoustic pressure component, for PRES response type will be reset to 1 (if less than 7) or 7 (if greater than 6 and less than 13).
23. Design response weight is obtained from Grid Point Weight Generator for a reference point GRDPNT (see parameter GRDPNT). If GRDPNT is either not defined, equal to zero, or not a defined grid point, the reference point is taken as the origin of the basic coordinate system. Fields ATTA and ATTB refer to the row and column numbers of the rigid body weight matrix, which is partitioned as follows:

$$[W] = \begin{bmatrix} W_x & W_{12} & W_{13} & W_{14} & W_{15} & W_{16} \\ W_{21} & W_y & W_{23} & W_{24} & W_{25} & W_{26} \\ W_{31} & W_{32} & W_z & W_{34} & W_{35} & W_{36} \\ W_{41} & W_{42} & W_{43} & I_x & W_{45} & W_{46} \\ W_{51} & W_{52} & W_{53} & W_{54} & I_y & W_{56} \\ W_{61} & W_{62} & W_{63} & W_{64} & W_{65} & I_z \end{bmatrix}_{6 \times 6}$$

The default values of ATTA and ATTB are 3, which specifies weight in the Z direction. Field ATT1 = "ALL" implies total weight of all superelements except external superelements. SEIDi refers to a superelement identification number. SEIDi = "0" refers to the residual superelement. The default of ATT1 is blank which is equivalent to "ALL".

24. For RTYPE = GPFORCE, the PTYPE field is used to designate the GRID ID at which the force is defined. Output that is produced using PARAM NOELOF > 0 is not supported for the DRESP1 entry.
25. For RTYPE = GPFORCP, the PTYPE field is blank. The grid point force is for the sum of all elements from the GRID ID listed in ATTA to the GRID (orient ID) listed in ATTi. This response corresponds to that produced with PARAM NOELP > 0. It is not necessary to set PARAM NOELP > 0 to compute the GPFORCP response.
26. For RTYPE = PSDXXXX, the PTYPE field specifies the RANDPS ID.
27. RTYPE=COMP (compliance of structures =  $\mathbf{P}^T \mathbf{u}$ ) and FRMASS (mass fraction of topology designed elements) entries are used for topology optimization or combined topology, sizing/shape optimization.
28. RTYPE=FRMASS can be used for topology and topometry optimization. For topology, FRMASS, is the mass divided by the mass calculated if all topology design variables are 1.0. FRMASS is calculated for designed elements only. FRMASS = 1.0 if all design variables are 1.0. For topometry, the initial FRMASS is defined as 1.0 for the initial design specified on the TOMVAR entries. For non-volume elements like CELAS, an artificial mass = 1.0 is assumed for each element.
29. Response type ABSTRESS is for element type code of 238 (CBAR) and 239 (CBEAM) only.
30. If a PSDxxxx or RMSxxxx response is being applied with RANDPS Bulk Data entries that include multiple subcases, the DESSUB or DESOBJ Case Control command that invokes the PSDxxxx or RMSxxxx response must be in the first subcase used by the RANDPS entry.
31. Response types, ESE, TOTSE, GPFORCE and GPFORCP are not supported for shape optimization.
32. For RTYPE=EIGN or FREQ, PTYPE field can be utilized to identify the source of the mode. Valid options are 'STRUC' or 'FLUID'. The default is 'STRUC'.
33. For RTYPE=ACPWR, PTYPE field can be utilized to identify the panel name. The default is 'total' which covers the whole interface between structural and fluid field.



34. For RTYPE=AFxxxx, PTYPE field can be utilized to specified the ID of Acoustic Field Point Mesh, AFPM.
35. For monitor point responses (RTYPE = STMONP1, STMOND1, MONPNT3 AEMONP1 or AEMOND1) the ATTA field specifies the components to be extracted. These can be any subset of the integers 1 through 6 that appear on the monitor quantity with the NAME provided in the PTYPE field. All of these responses can be invoked in a static aeroelastic (ANALYSIS=SAERO) subcase. STMOND1 and MONPNT3 can be invoked from a static (ANALYSIS=STAT) subcase. The responses are not available in a dynamic response or normal modes subcase. The response types have the following meaning:
  - a. STMONP1 – A structural MONPNT1
  - b. STMOND1 – A structural MONDSP1
  - c. MONPNT3 – A MONPNT3
  - d. AEMONP1 – An aerodynamic MONPNT1
  - e. AEMOND1 – An aerodynamic MONDSP1For all but the STMONP1, the response is the elastic monitor point value. For the STMONP1, it is the elastic “minus” inertial “plus” elastic applied load value.
36. For RTYPE=FRMASS, if the PID in the ATTi field is not blank, the PTYPE field must specify the type of property the PID references.
37. RTYPE=DIVERG is for subcases with ANALYSIS=DIVERG (aeroelastic static divergence). Only a single Mach number can be specified for this RTYPE.
38. For RTYPE = FATIGUE (pseudo-static fatigue), PTYPE must be set to ELEM or PSOLID, PSHELL, PBAR, PBEAM, or PWELD. For RTYPE = FRFTG (frequency response fatigue), PTYPE must be set to ELEM or PSOLID or PSHELL. ATTB is the FID of a FATIGUE case control and must be the same FID for all RTYPEs of type FATIGUE or FRFTG. RTYPEs of type FATIGUE and FRFTG cannot be mixed; either all are RTYPE=FATIGUE or all are RTYPE=FRFTG.
39. If a FATIGUE case control references a SET ID, then ATTB must reference one of the IDs referenced by the SET. Design optimization using fatigue responses is currently limited to one fatigue analysis, that is, one loading sequence. Multiple fatigue responses referencing different loading sequences, i.e., FATIGUE case control IDs, will result in an error.
40. For the ERP response, the PTYPE field must be ERPPNL.
41. If the ATT1 field is blank for the ERP response, all panels are invoked.
42. Fatigue Item Codes can be found in [Fatigue Item Codes](#). Item codes that are available as design responses are 4-9 and 12 for the element centroid or first node/layer/location and the comparable items for ensuing nodes/layers/locations. The use of NODA on the FTGPARM entry is not supported.
43. For the WMPID (weight as a function of material and property ID), the following conventions apply:
  - a. ATTA is required, ATTB and ATTi are optional.
  - b. ATTB specifies the superelement with ATTB=0 indicating the residual, >0, indicates superelement ATTB and ATTB = ALL or blank means all superelements)

- c. The ATTi fields designate the property ID's of interest. If ATTi is specified, the associated property type (e.g., PSHELL) must be specified by the PTYPE input of field 5 on the DRESP1 entry.
  - d. If there is no ATTi, all material with MID=ATTA will be used to generate the response.
  - e. If there are multiple ATTi, a single scalar weight is calculated that is the sum of the all the properties
  - f. Materials that are supported with this entry are: MAT1, MAT2, MAT3, MAT8, MAT9 and MAT10
  - g. Properties that are supported are: PSHELL, PROD, PBEAM, PTUBE, PSHEAR, PBAR, PSOLID, PBEND, PCOMP, PCOMPG, PBARL, PBEAML, PBRSECT and PBMSECT.
44. Certain restrictions apply for RTYPE=WMPID
- a. The density of the material referenced on the ATTA field cannot also be designed using a DVMRELx entry
  - b. The property referenced on the ATTI field cannot also be invoked by a TOMVAR entry.
  - c. The WMPID response is supported for topology optimization or for shape sizing optimization, but not for both topology and shape sizing optimization.

**DRESP2****Design Sensitivity Equation Response Quantities**

Defines equation responses that are used in the design, either as constraints or as an objective.

**Format:**

1	2	3	4	5	6	7	8	9	10
DRESP2	ID	LABEL	EQID or FUNC	REGION	METHOD	C1	C2	C3	
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						
	“DRESP1”	NR1	NR2	NR3	NR4	NR5	NR6	NR7	
		NR8	-etc.-						
	“DNODE”	G1	C1	G2	C2	G3	C3		
		G4	C4	etc.					
	“DVPREL1”	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7	
		DPIP8	DPIP9	-etc.-					
	“DVCREL1”	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7	
		DCIC8	DCIC9	-etc.-					
	“DVMREL1”	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7	
		DMIM8	DMIM9	-etc.-					
	“DVPREL2”	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7	
		DPI2P8	DPI2P9	-etc.-					
	“DVCREL2”	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7	
		DCI2C8	DCI2C9	-etc.-					
	“DVMREL2”	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7	
		DMI2M8	DMI2M9	-etc.-					
	“DRESP2”	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7	
		NRR8	-etc.-						
	“DVLREL1”	DLIL1	DLIL2	DLIL3	DLIL4	DLIL5	DLIL6	DLIL7	
		DLIL8	-etc.-						

**Example:**

DRESP2	1	LBUCK	5	3					
	DESVAR	101	3	4	5	1	205	209	
		201							
	DTABLE	PI	YM	L					
	DRESP1	14	1	4	22	6	33	2	

	DNODE	14	1	4	1	22	3		
		2	1	43	1				
	DVPREL1	101	102						
	DVCREL1	201	202						
	DVMREL1	301							
	DVPREL2	401	402						
	DVCREL2	501							
	DVMREL2	601	602	603					
	DRESP2	50	51						

Describer	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character, no default)
EQID	DEQATN entry identification number. (Integer > 0)
FUNC	Function to be applied to the arguments. See Remark 8. (Character)
REGION	Region identifier for constraint screening. See Remark 5. (Integer > 0)
METHOD	When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN)  When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS)
Ci	Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. (Real; Defaults: C1 = 1.0., C2 = .005, and C3=10.0)
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABELs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character). See Remark 13.
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag indicating grid point and component identification numbers. (Character)
Gm	Identification number for any grid point in the model. (Integer > 0)
Cm	Component number of grid point Gm. (1 ≤ Integer ≤ 3)
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)

Describer	Meaning
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCICi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVREL1 entry identification number. (Character)
DMIMi	DVMREL1 entry identification number. (Integer > 0)
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number. (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character). See Remark 13.
NRRk	DRESP2 entry identification number. (Integer > 0)
“DVLREL”	Flag indicating DVLREL1 identification numbers. (Character)
DLILi	DVLREL1 entry identification number. (Integer > 0)

**Remarks:**

1. DRESP2 entries may only reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2 and DVLREL1 entries. They may also reference other DRESP2 entries. However, a DRESP2 entry cannot reference itself directly or recursively.
2. a) If the referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify DRESP1 IDs for each subcase. DRESP2 entries that span subcases must be invoked above the subcase level by DESGLB on DESOBJ commands.  
 b) Referenced DRESP entries that span superelements are supported automatically.  
 c) Referenced DRESP2 entries cannot span subcases or superelements.
3. DRESP2 entries must have unique identification numbers with respect to DRESP1 entries.
4. The “DESVAR”, “DTABLE”, “DRESP1”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2” and DVLREL1 flags in field 2 must appear in the order given above. Any of these words, along with the identification numbers associated with them, may be omitted if they are not involved in this DRESP2 relationship. However, at least one of these ten types of arguments must exist.
5. The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP2 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP2 entry in the same region.

6. The variables identified by DVIDi, LABLj, NRk, the Gm, CMPM pairs, DPIPi, DCICm, DMIMn, DPI2Po, DCI2Cp, DMI2Mq, and NRRu are assigned (in that order) to the variable names (x1, x2, x3, etc.) specified in the left-hand side of the first equation on the DEQATN entry referenced by EQID. In the example below,

DESVARs 101 and 3 are assigned to arguments A and B.

DTABLEs PI and YM are assigned to arguments C and D.

Grid 14, Component 1 is assigned to argument R.

DRESP2	1	LBUCK	5	3						
	DESVAR	101	3							
	DTABLE	PI	YM							
	DNODE	14	1							
DEQATN	5	F1(A, B, C, D, R)=A+B*C-(D**3+10.0)+sin(C*R)								

7. (Gm, Cm) can refer to any grid component and is no longer limited to a designed grid component.
8. The FUNC attributes can be used in place of the EQID and supports the functions shown in the following table:

Function	Description
SUM	Sum of the arguments
SFMAX	Minimization of compliance and maximization of frequency. See remark 14.
AVG	Average of the arguments
SSQ	Sum of the squares of the arguments
RSS	Square root of the sum of the squares of the arguments
MAX	The maximum value of the argument list
MIN	The minimum value of the argument list
BETA	Minimize the maximum response. See Remark 10.
MATCH	Match analysis results with user specified values. See Remark 11.

When EQID has character input, the DEQATN entry is no longer needed. The functions are applied to all arguments on the DRESP2 regardless of the type. See Remark of the DRESP1 entry for the explanations of SUM, AVG, SSQ, RSS, MAX and MIN.

9. The number of arguments of a DEQATN can be more than the number of values defined on the DRESP2 if the DRESP1s referenced have RTYPE with 'FR' or 'PSD' prefix. Arguments are still positional. The extra arguments in the DEQATN must appear at the end of the argument list. The discrepancy is resolved internally with the forcing frequency(ies) associated with DRESP1s. An example is shown as follows:

DRESP1	10	FDISP1	FRDISP			1	10.	1001	
DRESP1	20	FDISP2	FRDISP			1	20.	1001	

DRESP2	30	AVGFD	100							
	DRESP1	10	20							
DEQATN	100	AVG(D1,D2,F1,F2) = (D1/F1+D2/F2)*0.5								

In the above example, the DEQATN has two more additional terms than have been defined on the DRESP2. The first additional term is the forcing frequency (in hertz) of the first DRESP1 ID on the DRESP2. The second additional term is the forcing frequency of second DRESP1 ID in the list. When all DRESP1s involved have the same frequency, the user is not required to name all the additional terms in the argument list of DEQATN.

10. FUNC = BETA facilitates a design task where the objective is to minimize the maximum response. Only DRESP1 entries can be invoked by DRESP2 and the DRESP's cannot span subcases. The BETA function creates the following design task:

$$\text{Minimize } \phi = C_1 X_\beta$$

$$\text{Subject to } g = \frac{r_j - \gamma X_\beta}{C_3} \leq 0$$

where  $\gamma$  is determined from

$$C_2 = (r_{jmax} - \gamma X_\beta) / C_3$$

User input parameters  $C_1$ ,  $C_2$ ,  $C_3$  therefore have the following meaning:

$C_1$  (Default = 1.0) weights the spawned design variable,  $X_\beta$ , to create the objective. Since  $X_\beta$  starts at 1.0,  $C_1$  is the initial objective.

$C_2$  sets the initial value of the maximum constraint created by this process. The default values of 0.005 is equal to DOPTPRM parameter GMAX.

$C_3$  (Default = 10.0) is an offset value to avoid dividing by zero when creating constraints.

11. FUNC = MATCH creates a response from the difference between analysis results,  $r_j$ , that are associated with DRESP1s and target values,  $r_j^T$ , that are input using DTABLE data. Only DRESP1 entries and DTABLE entries can be invoked by the DRESP2 entry.

When METHOD = LS, a least square minimization is performed where the response is

$$r_2 = \sum_{j=1}^m \left( \frac{r_j - r_j^T}{r_j^T} \right)^2$$

When METHOD = BETA, the design task becomes one of minimizing an objective that is the maximum normalized difference between the analysis and target values

$$\frac{r_j - r_j^T}{|r_j^T|}$$

in the same manner as outlined in Remark 10.

12. With FUNC=MATCH, if the DTABLE LABELj invokes an integer VALUi on the DTABLE entry, the integer points to a TABLEDi entry that provides tabular input as a function of frequency or time. This is used in conjunction with a single DRESP1 NRk that provides a response across a range of frequencies or times (i.e., RTYPE=FRxxxx, PSDxxxx, ACxxxx or Txxxx). When the TABLEDi option is invoked, the response can only be constrained, it cannot be the objective. This further implies that FUNC=MATCH, METHOD=BETA is not supported with TABLEDi.
13. If the mathematical function from a DRESP1 that has character input in the ATTB field (see remark 20 of the DRESP1) is to be used in the DRESP2 evaluation, the DRESP1 must be referenced under the “DRESP2” flag, not the “DRESP1” flag. If the “DRESP1” flag is used in this situation, it will result in a DRESP2 being evaluated for each of the individual values that contribute to the mathematical function.
14. The function SFMAX requires n+1 DRESP1 IDs. The first ‘n’ IDs correspond to compliance constraints for the first ‘n’ static subcases. The final ID refers to a DRESP1 with RTYPE=FREQ and mode=1. The function minimizes the sum of the compliance constraints while maximizing the fundamental frequency.



**DRESP3****Design Sensitivity Response using user-supplied routine(s)**

Defines an external response using user-supplied routine(s).

**Format:**

1	2	3	4	5	6	7	8	9	10	
DRESP3	ID	LABEL	GROUP	TYPE	REGION					
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7		
		DVID8	etc.							
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7		
		LABL8	etc.							
	“DRESP1”	NR1	NR2	NR3	NR4	NR5	NR6	NR7		
		NR8	etc.							
	“DNODE”	G1	C1	G2	C2	G3	C3			
		G4	C4	etc.						
	“DVPREL1”	DPIP1	DPIP2	DPIP3	DPIP4	DPIP5	DPIP6	DPIP7		
		DPIP8	DPIP9	etc.						
	“DVCREL1”	DCIC1	DCIC2	DCIC3	DCIC4	DCIC5	DCIC6	DCIC7		
		DCIC8	DCIC9	-etc.-						
	“DVMREL1”	DMIM1	DMIM2	DMIM3	DMIM4	DMIM5	DMIM6	DMIM7		
		DMIM8	DMIM9	-etc.-						
	“DVPREL2”	DPI2P1	DPI2P2	DPI2P3	DPI2P4	DPI2P5	DPI2P6	DPI2P7		
		DPI2P8	DPI2P9	-etc.-						
	“DCREL2”	DCI2C1	DCI2C2	DCI2C3	DCI2C4	DCI2C5	DCI2C6	DCI2C7		
		DCI2C8	DCI2C9	-etc.-						
	“DVMREL2”	DMI2M1	DMI2M2	DMI2M3	DMI2M4	DMI2M5	DMI2M6	DMI2M7		
		DMI2M8	DMI2M9	-etc.-						
	“DRESP2”	NRR1	NRR2	NRR3	NRR4	NRR5	NRR6	NRR7		
		NRR8	-etc.-							
	DVLRREL1	DLIL1	DLIL2	DLIL3	DLIL4	DLIL5	DLIL6	DLIL7		
		DLIL8	DLIL9	-etc.-						
	“USRDATA”	String								
		-etc.-								

**Example:**

DRESP3	1	LBUCK	TAILWNG	BUCK					
	DESVAR	101	3	4	5	1	205	209	

		201								
	DTABLE	PI	YM	L						
	DRESP1	14	1	4	22	6	33	2		
	DNODE	14	1	4	1	22	3			
		2	1	43	1					
	DVPREL1	101	102							
	DVCREL1	201	202							
	DVMREL1	301								
	DVPREL2	401	402							
	DVCREL2	501								
	DVMREL2	601	602	603						
	DRESP2	50	51							
	USRDATA	Constants: 12345.6 789.0 99.								

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
LABEL	User-defined label. (Character, no default)
GROUP	Group name the external response type belongs to (Character). See Remark 2.
TYPE	External response type (Character). See Remark 3.
“DESVAR”	Flag indicating DESVAR entry identification numbers. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABELs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABELi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)
“DRESP1”	Flag indicating DRESP1 entry identification numbers. (Character)
NRk	DRESP1 entry identification number. (Integer > 0)
“DNODE”	Flag signifying that the following fields are grid points.
Gm	Grid point identification number. (Integer > 0)
Cm	Degree-of-freedom number of grid point Gm. ( $1 \leq \text{Integer} \leq 3$ )
“DVPREL1”	Flag indicating DVPREL1 entry identification number. (Character)
DPIPi	DVPREL1 entry identification number. (Integer > 0)
“DVCREL1”	Flag indicating DVCREL1 entry identification number. (Character)
DCiCi	DVCREL1 entry identification number. (Integer > 0)
“DVMREL1”	Flag indicating DVMREL1 entry identification number. (Character)
DMiMi	DVMREL1 entry identification number. (Integer > 0)

Describer	Meaning
“DVPREL2”	Flag indicating DVPREL2 entry identification number. (Character)
DPI2Pi	DVPREL2 entry identification number. (Integer > 0)
“DVCREL2”	Flag indicating DVCREL2 entry identification number. (Character)
DCI2Ci	DVCREL2 entry identification number. (Integer > 0)
“DVMREL2”	Flag indicating DVMREL2 entry identification number. (Character)
DMI2Mi	DVMREL2 entry identification number (Integer > 0)
“DRESP2”	Flag indicating other DRESP2 entry identification number. (Character)
NRRk	DRESP2 entry identification number. (Integer > 0)
“DVLREL1”	Flag indicating DVLREL1 identification number (character)
DLILi	DVLREL1 entry identification number (integer)
“USRDATA”	Flag indicating user input data (Character). See Remark 8.

### Remarks:

- DRESP3 entries may reference DESVAR, DTABLE, DRESP1, DNODE, DVPREL1, DVCREL1, DVMREL1, DVPREL2, DVCREL2, DVMREL2, DRESP2 and DVLREL1 entries. However, a DRESP3 entry cannot reference another DRESP3 entry.
- The group name must be referenced by an FMS CONNECT entry.
- Multiple types of external responses can be defined in one group. Each type name identifies a specific external response evaluated in the user-supplied routines. See [Building and Using the Sample Programs MSC Nastran Utilities Guide](#) for a discussion of how to incorporate external responses.
- Referenced DRESP2 entries cannot span subcases or superelements.
  - If referenced DRESP1 entries span subcases, the DRSPAN Case Control command is required to identify the DRESP1 IDs for each subcase. DRESP3 entries that span subcases must be invoked above the subcase level by DESGLB or DESOBJ commands.
  - Referenced DRESP1 entries that span superelements are supported automatically.
- DRESP3 entries must have unique identification numbers with respect to DRESP2 and DRESP1 entries.
- The “DESVAR”, “DTABLE”, “DNODE”, “DVPREL1”, “DVCREL1” and “DVMREL1”, “DVPREL2”, “DVCREL2”, “DVMREL2”, “DRESP2”, “DVLREL1” and “USRDATA” keywords on the continuation entries must appear in the order given above. Any of these words, along with the subsequent data associated with them, may be omitted if they are not involved in this DRESP3 relationship. However, at least one of these types of arguments must exist.
- The REGION field follows the same rules as for the DRESP1 entries. DRESP1 and DRESP3 responses will never be contained in the same region, even if they are assigned the same REGION identification number. The default is to put all responses referenced by one DRESP3 entry in the same region.

8. The data in the USRDATA field is character string based. It provides a convenient way to pass constants to the external response server routines. The maximum number of characters allowed is 32000.

## DSCREEN

## Design Constraint Screening Data

Defines screening data for constraint deletion.

### Format:

1	2	3	4	5	6	7	8	9	10
DSCREEN	RTYPE	TRS	NSTR						

### Example:

DSCREEN	STRESS	-0.7	2						
---------	--------	------	---	--	--	--	--	--	--

Describer	Meaning
RTYPE	Response type (or ALL, See Remark 9.) for which the screening criteria apply. (Character)
TRS	Truncation threshold. (Real; Default = -0.5)
NSTR	Maximum number of constraints to be retained per region per load case. See Remark 3. (Integer > 0; Default = 20)

### Remarks:

1. Grid responses associated with one particular load case are grouped by the specification of DRESP1 entries. From each group, a maximum of NSTR constraints are retained per load case.
2. Element responses are grouped by the property; i.e., all element responses for one particular load case belonging to the set of PIDs specified under ATTi on a DRESPi entry are regarded as belonging to the same region. In superelement sensitivity analysis, if the property (PID) is defined in more than one superelement, then separate regions are defined. A particular stress constraint specification may be applied to many elements in a region generating many stress constraints, but only up to NSTR constraints per load case will be retained.
3. For aeroelastic responses, that is RTYPE = "TRIM", "STABDER", and "FLUTTER", the NSTR limit is applied to all DRESP1 IDs that are the same RTYPE and have the same REGION specified.
4. For responses that are not related to grids or elements, that is RTYPE = "WEIGHT", "VOLUME", "EIGN", "FREQ", "LAMA", "CEIG", "FRMASS", "COMP", and "TOTSE", NSTR is not used. TRS is still applicable.
5. The RTYPE field is set to EQUA if constraints that are associated with DRESP2 entries are to be screened. The RTYPE field is set to DRESP3 if constraints that are associated with DRESP3 entries are to be screened. If the REGION field on the DRESP2 or DRESP3 is blank, one region is established for each DRESP2/DRESP3 entry.
6. If a certain type of constraint exists but no corresponding DSCREEN entry is specified, all the screening criteria used for this type of constraint will be furnished by the default values.
7. Constraints can be retained only if they are greater than TRS. See the Remarks under the [DCONSTR](#) entry for a definition of constraint value.

8. Constraint screening is applied to each superelement.
9. RTYPE=ALL specifies that the specified TRS and NSTR applies to all response types and other DSCREEN entries are ignored.

**DTABLE** Table Constants

Defines a table of real constants that are used in equations (see DEQATN entry).

**Format:**

1	2	3	4	5	6	7	8	9	10
DTABLE	LABL1	VALU1	LABL2	VALU2	LABL3	VALU3	LABL4	VALU4	
	LABL5	VALU5	LABL6	VALU6	LABL7	VALU7	LABL8	VALU8	
		-etc.-							

**Example:**

DTABLE	PI	3.142	H	10.1	E	1.0E6			
	G	5.5E5	B	100.					

Describer	Meaning
LABLi	Label for the constant. (Character)
VALUi	Value of the constant or TABLEDx ID. (Real or Integer, respectively)

**Remarks:**

- Multiple DTABLE entries may be specified in the Bulk Data Section.
- LABLi are referenced by the LABj on the DRESP2, DRESP3, DVCREL2, DVMREL2, or DVPREL2 entries.
- Trailing blank fields are permitted at the end of each line of LABLi/VALUi pairs, but intermediate blanks are not. (See the example above for permitted trailing blanks.)
- For PART SE, if LABLi is referenced on SEDRSP2 and/or SEDRSP3, DTABLE entries must be placed in a PART SE where companion design model entries, such as DESVAR, DRESP1 and etc, are available.
- If the VALUi is an integer, this points to the ID of a TABLEDx Bulk Data entry that lists the constants as a function of frequency or time. See Remark 12. on the DRESP2 entry.
- LABLi must be unique across all DTABLE and DTABLE2 entries.

**DTABLE2** Table Constants

Defines real constants from a field of property, material or connections bulk data entries which then can be invoked by a DV<sub>x</sub>REL2, DRESP2, or DRESP3 entry.

**Format:**

1	2	3	4	5	6	7	8	9	10
DTABLE2	LABL1	PNAME1	PID1	FNAME1	LABL2	PNAME2	PID2	FNAME2	
	LABL3	PNAME3	PID3	FNAME3					

**Example:**

DTABLE2	PTHK10	PSHELL	10	T	MATIE	MAT1	38	E	
	CBARX1	CBAR	3888	X1					

Describer	Meaning
LABLi	Label for the constant. (Character)
PNAMEi	Property, material or connection bulk data entry name. (Character)
PIDi	ID of PNAMEi entry. (Integer > 0)
FNAMEi	Field name of PNAMEi. (Character)

**Remarks:**

1. LABLi on DTABLE2 and DTABLE must be unique.
2. LABLi on DTABLE2 can be referenced under DTABLE flag of DV<sub>x</sub>REL2 (where x=P, M or C)/DRESP2/DRESP3.
3. Values for the FNAMEi field of the PNAMEi Bulk Data entry with the ID of PIDi are taken from analysis model before updating of analysis values with the designed value. If the updated value is desired, use the DV<sub>x</sub>REL2 flag on DRESP2 or DRESP3 entries instead.
4. FNAMEi must be the same as the character string that appears on the PNAMEi Bulk Data entry.
5. DTABLE2 must be utilized along with 'NASTRAN SYSTEM(444)=1' in input file or 'sys444=1' during job submittal.



**DTI** Direct Table Input

Defines table data blocks.

**Format:**

1	2	3	4	5	6	7	8	9	10
DTI	NAME	"0"	T1	T2	T3	T4	T5	T6	
	V01	V02	-etc.-						
DTI	NAME	IREC	V1	V2	V3	V4	V5	V6	
	V7	V8	V9	V10	-etc.-	"ENDREC"			

**Example: (The first logical entry is the header entry.)**

DTI	XXX	0	3	4	4096		1	0	
	1.2	2.3							
DTI	XXX	1	2.0	-6	ABC	6.000	-1	2	
	4	-6.2	2.9	1	DEF	-1	ENDREC		

Describer	Meaning
NAME	Any character string that will be used in the DMAP sequence to reference the data block. See Remark 1. (Character; the first character must be alphabetic.)
Ti	Trailer values. (Integer $\geq 0$ ; Default = 32767)
IREC	Record number. (Integer $> 1$ )
V0i, Vi	Value. (Integer, Real, Character or blank)
"ENDREC"	Flags the end of the string of values (V0i or Vi) that constitute record IREC. (Character)

**Remarks:**

1. The user defines the data block and therefore must write a DMAP (or ALTER a solution sequence), which includes the DTIIN modules, in order to use the DTI feature. See the [MSC Nastran DMAP Programmer's Guide](#). All of the rules governing the use of data blocks in DMAP sequences apply.
2. All fields following ENDREC must be blank.
3. The entry using IREC = 0 is called the header entry and is an optional entry. The values T1 through T6 go to a special record called the trailer. Other values on the optional continuation go to the header record. If the header entry or the trailer is not specified, T1 through T6 = 32767. On this entry, "ENDREC" may be used only if there is at least one continuation.
4. In addition to the optional header entry, there must be one logical entry for each record in the table. Null records require no entries.

5. “ENDREC” is used to input blank values at the end of a record. If “ENDREC” is not specified, the string for a record ends with the last nonblank field.
6. The maximum number of DMI and DTI data blocks is 1000.
7. If Ti is not an integer, a machine-dependent error will be issued that may be difficult to interpret.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

## DTI,ESTDATA

## Superelement Estimation Data Overrides

Provides override data for time and space estimation for superelement processing operations.

### Format:

1	2	3	4	5	6	7	8	9	10
DTI	ESTDATA	"0"							
	kd1	vd1	kd2	vd2	-etc.-				

The next entries are repeated for any superelement for which estimate data overrides are desired. IREC must be incremented by 1.

DTI	ESTDATA	IREC	SEFLAG	SEID	k1	v1	k2	v2	
	k3	v3	-etc.-						

### Example:

DTI	ESTDATA	0							
	NOMASS	-1							
DTI	ESTDATA	1	SE	10	C1	5.5	C3	4.5	
	C7	7.3							

Describer	Meaning
kdi	Keyword for estimation parameter. (Character from <a href="#">Table 9-9</a> .)
vdi	Value assigned to the estimation parameter kdi. (The type given in <a href="#">Table 9-9</a> .)
IREC	Record number beginning with 1. (Integer > 0)
SEFLAG	SEFLAG = "SE" or "SEID" indicates the next field containing a superelement identification number. (Character)
SEID	Superelement identification number. (Integer > 0)
ki	Keyword for override of estimation parameter for indicated superelement. (Character from <a href="#">Table 9-9</a> .)
vi	Value for keyword ki. (Type depends on ki as shown in the <a href="#">Table 9-9</a> .)

Table 9-9 DTI,ESTDATA Input Parameter Descriptions

Input Parameters				Meaning and Comments
Keyword	Type	Default Value	Math Symbol	
CRMS*	Real	-1.0	C	Number of active columns in $[K_{oo}]$ .
FCRMS*	Real	0.10		If $FCRMS \leq 0.0$ , FCRMS is used (c/o).
C1	Real	6.0	$c_1$	Average number of degrees-of-freedom per grid point in o-set.
C3	Real	8.0	$c_3$	Average number of connections per grid point.
C4	Real	0.15	$c_4$	I/O time (seconds) per block transferred.
C5	Real	6.0	$c_5$	Average number of effective degrees-of-freedom per grid point in a-set.
C6	Real	1.2	$c_6$	Total CPU factor.
C7	Real	6.0	$c_7$	Number of equivalent KGG data blocks for space estimation.
WF	Real	-1.0	W	If $WF \leq 0.0$ then use available working storage in units of single-precision words.
NOMASS	Integer	1		If $NOMASS \neq 1$ then exclude mass terms from estimates.
TSEX	Real	0.5 (min)		Threshold limit for CPU.
SSEX	Real	50.0 (blocks)		Threshold limit for space.
TWALLX	Real	5.0 (min)		Threshold limit for wall time.
BUFSIZ	Integer	Machine Buffsize	B	Buffsize. See <a href="#">The NASTRAN Statement (Optional)</a> in the <i>MSC Nastran Reference Guide</i> .
ML	Real	Machine Loop Time	M	Arithmetic time for the multiply/add loop. See the <i>SOL 700 Explicit Nonlinear User's Guide</i> .
CONIO	Integer	Machine I/O ratio		I/O count/CPU equivalence
PREC	Integer	1 or 2		Machine Word Length (1 = long, 2 = short). See <a href="#">The NASTRAN Statement (Optional)</a> in the <i>MSC Nastran Reference Guide</i> .
NLOADS	Integer	1	$N_L$	Number of loading conditions
SETYPE	Character	"T"		Superelement type (T = Tip)
CMAX	Real	-1.0	$C_{max}$	Maximum bandwidth

### Parameters Obtained from SEMAP

NGI	Number of interior grid points.
NPE	Number of exterior grid points.
NS	Number of scalar points
NE	Number of elements.

### Derived Parameters

$O = C1 + NGI$	Size of o-set.
$A = C5(NPE - NS) + NS$	Size of a-set.
$T = \text{BUFFSIZE}/\text{PREC}$	Number of matrix terms in a buffer.

### Estimation Equations

For each superelement, estimates of CPU time and disk space are made using the following equations.

Table 9-10      Equations Used for CPU Time and Disk Space Estimate

Printout Symbol	Math Symbol	Equations
TD	$T_1$	$T_1 = 1/2 \cdot M \cdot O \cdot C^2$
TFBS	$T_2$	$T_2 = 2 \cdot M \cdot C \cdot O \cdot a$
TMAA	$T_3$	$T_3 = M \cdot O \cdot a^2$ (set to 0.0 if NOMASS $\neq$ +1)
TSE	$T_{SE}$	$T_{SE} = C_6(T_1 + T_2 + T_3)$
SLOO	$S_1$	$S_1 = O \cdot C \cdot \frac{\text{PREC}}{b}$
SGO	$S_2$	$S_2 = O \cdot a \cdot \frac{\text{PREC}}{B}$
SKGG	$S_3$	$S_3 = 36(NG_i + NG_e - NS)(c_3 + 1.0) \left( \frac{\text{PREC}}{B} \right)$

Table 9-10 Equations Used for CPU Time and Disk Space Estimate

Printout Symbol	Math Symbol	Equations
SSE	$S_{SE}$	$S_{SE} = S_1 + S_2 + c_7 \cdot S_3$
PASSES	$p$	$FBS \text{ passes} = p = a \cdot O \cdot \frac{PREC}{WF}$
BKSTRN	$BT$	Blocks Transferred = $BT = 2 \cdot p \cdot S_1 + S_2 + p \cdot S_2$ . (Last term omitted if $NOMASS \neq +1$ )
TWALL	$T_W$	Wall Time = $T_W = T_{SE} + c_4 \cdot BT$

**Remarks:**

- In the superelement solution sequences, this data is stored automatically.
- The header record continuation entries are optional if no global override data is to be specified. In this case, the complete header entry is optional.
  - Active column data can come from one of several places. The value for CRMS is determined as follows:
    - RMS from the entry when  $IREC > 0$  and field 4 is "SE".
    - RMS from entries with  $IREC = 0$ .
    - Computed bandwidth when  $PARAM, OLDSEQ$  is specified.
    - If  $FCRMS$  is specified when  $IREC > 0$  and field 4 is "SE", then  $CRMS = FCRMS \cdot O$ .
    - If  $FCRMS$  is specified when  $IREC = 0$ , then  $CRMS = FCRMS \cdot O$ .
    - $CRMS = 0.1 \cdot O$ .
- If  $C_{MAX}$  is not specified, then it is defaulted to  $CRMS$ .
- In the example above, mass terms are excluded for all superelements and new values are given for parameters  $C_1$ ,  $C_3$ , and  $C_7$  for Superelement 10 only.
- The estimates for  $TSEX$ ,  $SSEX$ , and  $TWALLX$  are not printed unless at least one estimate exceeds the threshold.

## DTI,INDTA Stress, Strain and/or Force Sort/Filter Item Code Override

Specifies or overrides default item codes for the sorting and filtering of element stresses, strains, and forces.

### Format:

1	2	3	4	5	6	7	8	9	10
DTI	INDTA	"0"							

To specify/override items for a sort of stress quantities:

DTI	INDTA	"1"	B1	C1	B2	C2	"ENDREC"		
-----	-------	-----	----	----	----	----	----------	--	--

To specify/override items for a sort of force quantities:

DTI	INDTA	"2"	B1	C1	B2	C2	"ENDREC"		
-----	-------	-----	----	----	----	----	----------	--	--

### Examples:

DTI	INDTA	0							
-----	-------	---	--	--	--	--	--	--	--

To specify/override items for a sort of stress quantities:

DTI	INDTA	1	64	18	75	18	ENDREC		
-----	-------	---	----	----	----	----	--------	--	--

To specify/override items for a sort of force quantities:

DTI	INDTA	2	34	2	2	4	ENDREC		
-----	-------	---	----	---	---	---	--------	--	--

Describer	Meaning
Bi	Element type identification number. See the table in <a href="#">Item Codes</a> for allowable values. (Integer > 0)
Ci	Item code identification number for the stress, strain, or force quantity on which the sort or filter is to be performed. See the table in the <a href="#">Item Codes</a> for allowable values. (Integer)

### Remarks:

- This table is recognized only in SOLs 101, 103, 105, 106, 108, 109, 111, 112, 114, 115, 144, 153, and for stress, strain or force quantities only. One or more of the user parameters S1, S1G, or S1M must be specified with a value greater than or equal to zero in order to request sorting and/or filtering. See user parameter S1 in [Parameters](#). In order to sort force or strain quantities, a DMAP Alter is required.

2. If the Ci value is -1, the element type will be suppressed on the output file. An example of this feature could be as follows: If an element type is to be sorted on two different values and output twice, this can be accomplished by two calls to the STRSORT module with two unique DTI tables. However, other element types will be printed twice. This additional print can be suppressed by setting their sort codes to -1.
3. [Table 13](#) lists the elements currently that are sortable. In addition, the element type identification number, the default stress output quantity, and the associated stress code identification numbers are provided. If this entry is not specified, then the stresses are sorted based on the default quantity given in [Table 13](#).

The following should be noted:

- a. The element type identification number is used internally by the program to differentiate element types.
- b. The stress code identification number is merely the word number in the standard printed output for the stress quantity of interest. For example, the thirteenth word of stress output for the CHEXA element is the octahedral shear stress. For this element type, the element identification number and the grid point ID each count as a separate word. Stress codes for the elements are tabulated in [Item Codes](#).
- c. By default, stress sorting for the membrane and plate elements will be performed on the Hencky-von Mises stress. For maximum shear stress, the STRESS (MAXS) Case Control command should be specified.



Table 13 Sortable Elements

Element	Element Type ID Number	Default Stress Output Quantity and Identification Number	
		Quantity	Stress Code ID Number
CBAR	34	Maximum stress at end B	14
CBEAM	2	Maximum stress at end B	108
CBEND	69	Maximum stress at end B	20
CONROD	10	Axial stress	2
CELAS1	11	Stress	2
CELAS2	12	Stress	2
CELAS3	13	Stress	2
CHEXA	67	Hencky-von Mises or Octahedral stress	13
CQUAD4	33	Maximum shear or Hencky-von Mises stress at $Z_2$	17
CQUAD4*	144	Maximum shear or Hencky-von Mises stress at $Z_2$	19
CQUAD8	64	Maximum shear or Hencky-von Mises stress at $Z_2$	19
CQUADR	82	Maximum shear or Hencky-von Mises stress at $Z_2$	19
CPENTA	68	Octahedral stress	13
CROD	1	Axial stress	2
CSHEAR	4	No default	---
CTETRA	39	No default	---
CTRIA3	74	Maximum shear or Hencky-von Mises stress at $Z_2$	17
CTRIA6	75	Maximum shear or Hencky-von Mises stress at $Z_2$	19
CTRIAR	70	Maximum shear or Hencky-von Mises stress at $Z_2$	19
CTRIAX6	53	No default	---
CTUBE	3	Axial stress	2

\*CORNER output

## DTI,SETREE

### Superelement Tree Definition

Defines a superelement tree that determines the superelement processing order.

#### Format:

1	2	3	4	5	6	7	8	9	10
DTI	SETREE	"1"	SEUP1	SEDOWN1	SEUP2	SEDOWN2	SEUP3	SEDOWN3	
	SEUP4	SEDOWN4	SEUP5	SEDOWN5	-etc.-				

#### Example:

DTI	SETREE	1	1	14	2	14	3	14	
	4	14	14	0					

Describer	Meaning
SEUP <sub>i</sub>	Identification number of the superelement upstream from SEDOWN <sub>i</sub> . (Integer > 0)
SEDOWN <sub>i</sub>	Identification number of the superelement into which SEUP <sub>i</sub> is assembled. (Integer ≥ 0)

#### Remarks:

1. SETREE entries or the DTI,SETREE entry are required for multi-level superelement configurations.
2. If an DTI,SETREE entry is provided, then SETREE entries are not required.
3. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.
4. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one of more of the superelements are not in the same path.
5. If this entry is not present, the superelement tree and the processing order are determined automatically.
6. A superelement identification may appear only once in a SEUP<sub>i</sub> field.
7. On restart, if a superelement identification does not appear in a SEUP<sub>i</sub> field, its matrices will not be assembled, even though they may be present in the database.
8. See the *MSC Nastran Superelements and Modules User's Guide* for a description of user-designated trees.

9. This entry is stored in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.
10. In the example above, the following superelement tree is defined:

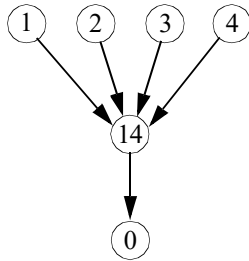


Figure 9-89 Sample Superelement Tree

## DTI,SPECSEL

## Response Spectra Input Correlation Table

Correlates spectra lines specified on TABLED1 entries with damping values.

### Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPECSEL	RECNO		TYPE	TID1	DAMP1	TID2	DAMP2	
	TID3	DAMP3	TID4	DAMP4	TID5	DAMP5	-etc.-		

### Example:

DTI	SPECSEL	1		A	1	.02	2	.04	
	3	.06							
DTI	SPECSEL	3		V	4	.01			

Describer	Meaning
RECNO	Spectrum number. (Integer > 0)
TYPE	Type of spectrum. (Character: "A" for acceleration, "V" for velocity, or "D" for displacement.)
TID <sub>i</sub>	TABLED1 entry identification number. (Integer > 0)
DAMP <sub>i</sub>	Damping value assigned to TID <sub>i</sub> . (Real)

### Remarks:

1. The RECNO is the number of the spectrum defined by this entry. It is referenced on DLOAD Bulk Data entries.
2. The TID<sub>i</sub>, DAMP<sub>i</sub> pairs list the TABLED1 entry, which defines a line of the spectrum and the damping value assigned to it. The damping value is in the units of fraction of critical damping.
3. This entry is placed in the database automatically. Once stored, the Bulk Data entry may be removed from the input file.

## DTI,SPSEL

### Response Spectra Generation Correlation Table

Correlates output requests with frequency and damping ranges.

#### Format:

1	2	3	4	5	6	7	8	9	10
DTI	SPSEL	RECNO	DAMPL	FREQ1	G1	G2	G3	G4	
	G5	G6	G7	-etc.-					

#### Example:

DTI	SPSEL	1	2	1	11	12			
DTI	SPSEL	2	4	3	1	7	11	12	
	13	14							

Describer	Meaning
DAMPL	Identification number of the FREQ, FREQ1, or FREQ2 Bulk Data entry that specifies the list of damping values. (Integer > 0)
FREQ1	Identification number of the FREQi Bulk Data entry that specifies the list of frequencies. (Integer > 0)
Gi	Grid point number where response spectra will be calculated. (Integer > 0)
RECNO	Record number of spectra to be generated. (Sequential integer beginning with 1.)

#### Remarks:

1. This table is used in SOLs 109 and 112.
2. Damping values are in the units of fraction of critical damping.
3. Output of response spectra requires the use of the XYPLOT...SPECTRA(RECNO)/Gi... command, where Gi is restricted to the grid points listed on the (RECNO) record of this entry.
4. The SPSEL table is stored in the database automatically in SOLs 109 and 112. Once stored, the Bulk Data entry may be removed from the input file.
5. There must be case control output request for displacements and velocities of the points to be output. For example you could use case control commands:

DISP(PLOT) = ALL

VELO(PLOT) = ALL

**DTI,UNITS**

## Unit Definitions

Defines units necessary for conversion during the analysis for the Nastran/ADAMS interface or a Nastran fatigue analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
DTI	UNITS	1	MASS	FORCE	LENGTH	TIME	STRESS		

**Example:**

DTI	UNITS	1	KG	N	M	S	MPA		
-----	-------	---	----	---	---	---	-----	--	--

**Remarks:**

1. The DTI,UNITS Bulk Data entry is required for a ADAMSMNF FLEXBODY=YES run. See the [ADAMSMNF\\* \(Case\)](#) case control entry. ADAMS is not a unitless code (as is Nastran). Units must be specified. A DTI Bulk Data entry provides 'UNITS' (a unique identifier) input as the above example illustrates. Once identified, the units will apply to all superelements in the model. Acceptable character input strings are listed in the table below. MASS, FORCE, LENGTH, and TIME are required for ADAMS interface.
2. MSC Nastran is a unitless code and it is the user's responsibility to ensure compatible units. During a fatigue analysis, stress is converted to SI units of MPa because the fatigue material property stress parameters as defined on the MATFTG entry are internally converted to standard SI units of MPa. The stresses from the analysis must match. Thus it is necessary for the user to use DTI,UNITS to define the stress units to ensure proper conversion. The default units are MPa and this entry is only necessary if stresses are not in MPa.

Mass	Force	Length	Time	Stress*
KG - kilogram	N - newton	KM - kilometer	H - hour	MPA - megapascal
LBM - pound-mass	LBF - pounds-force	M - meter	MIN - minute	PA - pascal
SLUG - slug	KGF - kilograms-force	CM - centimeter	S - second	PSI - pound per square inch
GRAM - gram	OZF - ounce-force	MM - millimeter	MS - millisecond	KSI - kilo pound per square inch
OZM - ounce-mass	DYNE - dyne	MI - mile	US - microsecond	PSF - pound per square foot

Mass	Force	Length	Time	Stress*
KLBM - kilo pound-mass (1000 lbm)	KN - kilonewton	FT - foot	NANOSEC - nanosecond	KSF - kilo pound per square foot
MGG - megagram	KLBF - kilo pound-force (1000 lbf)	IN- inch	D - day	DYNECM2 - dyne per square centimeter
SLINCH - 12 slugs	MN - millinewton	UM - micrometer		BAR - bar
UG - microgram	UN - micronewton	NM - nanometer		ATM - physical atmosphere
NG - nanogram	NN - nanonewton	ANG - angstrom		
USTON - US ton		YD - yard		
		MIL - milli-inch		
		UIN - micro-inch		

\* For fatigue analysis of spot welds, only MPA, PA, PSI, KSI, PSF, and KSF are supported. For random vibration fatigue analysis using SOL 108 or 111, only MPA, PA, PSI, and KSI are supported.

**DVBSHAP**

## Design Variable to Boundary Shapes

Associates a design variable identification number to a linear combination of boundary shape vectors from a particular auxiliary model.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVBSHAP	DVID	AUXMOD	COL1	SF1	COL2	SF2	COL3	SF3	

**Example:**

DVBSHAP	4	1	1	1.6					
---------	---	---	---	-----	--	--	--	--	--

Describer	Meaning
DVID	Design variable identification number of a DESVAR entry. (Integer > 0)
AUXMOD	Auxiliary model identification number. (Integer > 0)
COLi	Load sequence identification number from AUXMODEL Case Control command. (Integer > 0)
SFi	Scaling factor for load sequence identification number. (Real; Default = 1.0)

**Remarks:**

1. Design variable DVID must be defined on a DESVAR entry.
2. Multiple references to the same DVID and/or COLi will result in the vector addition of the referenced boundary shape vectors.
3. Multiple DVBSHAP entries may be specified.



**DVCREL1****Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVCREL1	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

**Example:**

DVCREL1	5	CQUAD4	1	ZOFFS		1.0			
	1	1.0							

Describer	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as “CBAR”, “CQUAD4”, etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as “X1”, “X2”, “X3”, “ZOFFS”, etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
CPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default =1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. (If i = 1, Real or Character; if i > 1, Real)

**Remarks:**

1. The relationship between the connectivity property and design variables is given by:

$$CP_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc), ZOFFS (case insensitive) must be specified on the CPNAME field.

4. The default values for CPMIN and CPMAX are not applied when the linear property is a function of a single design variable and  $C0=0$ . It is expected that the limits applied on the associated DESVAR entry will keep the designed property within meaningful bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the connectivity bulk data entry. If a DVCREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
6. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL1 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL1 entry should refer to CQUAD4/CTRIA3 types.

**DVCREL2****Design Variable to Connectivity Property Relation**

Defines the relation between a connectivity property and design variables with a user-supplied equation.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVCREL2	ID	TYPE	EID	CPNAME	CPMIN	CPMAX	EQID		
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

**Example:**

DVCREL2	1	CBAR	100	X1	0.05	1.0	100		
	DESVAR	1001							
	DTABLE	X10							

Describer	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of an element connectivity entry, such as “CBAR”, “CQUAD4”, etc. (Character)
EID	Element Identification number. (Integer > 0)
CPNAME	Name of connectivity property, such as “X1”, “X2”, “X3”, “ZOFFS”, etc. (Character)
CPMIN	Minimum value allowed for this property. If CPNAME references a connectivity property that can only be positive, then the default value of CPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
CPMAX	Maximum value allowed for this property. (Real; Default =1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
“DESVAR”	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

**Remarks:**

1. The variable identified by  $DVID_i$  and  $LABL_i$  correspond to variable names ( $x_1, x_2$ , etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names  $x_1$  through  $x_N$  (where  $N = m + n$ ) are assigned in the order  $DVID_1, DVID_2, \dots, DVID_m, LABL_1, LABL_2, \dots, LABL_n$ .
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, CPNAME, only accepts string characters. These string values must be the same as those given in the connectivity entry descriptions in this Guide. For example, if the plate offset is to be designed (CQUAD4, CTRIA3, etc.), ZOFFS (case insensitive) must be specified on the CPNAME field.
4. If the user inputs CQUAD4/CTRIA3 entries and then uses QRMETH = 5 to convert them to CQUADR/CTRIAR entries, the design of items on these entries using the DVCREL2 entry should refer to the converted type (i.e., CQUADR/CTRIAR). Similarly, if QRMETH=2 or 3 is used, the DVCREL2 entry should refer to CQUAD4/CTRIA3 types.

**DVGRID**

## Design Variable to Grid Point Relation

Defines the relationship between design variables and grid point locations.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVGRID	DVID	GID	CID	COEFF	N1	N2	N3		

**Example:**

DVGRID	3	108	5	0.2	0.5	0.3	1.0		
--------	---	-----	---	-----	-----	-----	-----	--	--

Describer	Meaning
DVID	DESVAR entry identification number. (Integer > 0)
GID	Grid point (GRID) or geometric point (POINT) identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
COEFF	Multiplier of the vector defined by Ni. (Real; Default = 0.0)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni ≠ 0.0)

**Remarks:**

1. A CID of zero or blank (the default) references the basic coordinate system.
2. Multiple references to the same grid ID and design variable result in vectorial addition of the participation vectors defined by CID, COEFF, and Ni. There is no restriction on the number of DVGRID entries that may reference a given grid (GID) or design variable (DVID).
3. The coordinate update equation is given as

$$\{g\}_i - \{g\}_i^0 = \sum_j \text{COEFF}_j (X_{\text{DVID}j} - X_{\text{DVID}j}^0) \{N\}_j$$

where  $\{g\}_i$  is the location of the  $i$ -th grid,  $[g_x g_y g_z]^T$ .

The vector  $\{N\} = [N_x N_y N_z]^T$  is determined from CID and Ni. Note that it is a change in a design variable from its initial value  $X^0$ , and not the absolute value of the design variable itself, that represents a change in a grid point location,  $\{g\}_i - \{g\}_i^0$ .

4. The DVGRID entry defines the participation coefficients (basis vectors) of each design variable for each of the coordinates affected by the design process in the relationship

$$\{\Delta g\}_i = \sum_j \{T\}_{ij} \cdot \Delta X_j$$

5. DVGRID entries that reference grid points on MPCs or RSSCON entries produce incorrect sensitivities. Often the sensitivities are 0.0 which may result in a warning message indicating zero gradients which may be followed by UFM 6499. Other rigid elements produce correct results.

**DVLREL1**

Defines the linear relation between analysis model loading and design variables in SOL 200 with Analysis = STATICS

**Format:**

1	2	3	4	5	6	7	8	9	10
DVLREL1	ID	TYPE	SID	LNAME	LMIN	LMAX	C0		
	ATT1	ATT2	ATT3	ATT4	ATT5				
	DVID	COEF	DVID2	COEF2	DVID3	Etc			

**Example: Design the N1 FORCE at GRID 100 to be equal to DESVAR=10**

DVLREL1	10	FORCE	300	N1					
	100								
	10	1.0							

Describer	Meaning
ID	Unique identification number (Integer>0)
TYPE	Name of Load, such as FORCE, see Remark 2. for supported types (Character)
SID	Load set ID (Integer>0)
LNAME	Load Name, such as F or N1 on the FORCE entry. See Remark 2. (Character)
LMIN	Minimum value for the load. See Remark 5. (Real, default=-1.0e35)
LMAX	Maximum value for the load (Real, Default=1.0e20)
C 0	Constant term of relation (Real, Default=0.0)
ATTi	Attributes of the designed load, see Remark 2. (Integer>0 or blank)
DVIDi	DESVAR entry identification number. (Integer>0)
COEFi	Coefficient of linear relation or keyword="PVAL", See Remark 3. (If i=1, Real or Character; if i>1, Real )

**Remarks:**

1. The relationship between the analysis model load and the design variables is given by:

$$L_i = C_0 + \sum COEFF_i X_{DVID_i}$$

2. Supported TYPEs and their ATTi meaning are given in the following table:

Load Type	LNAME (character)	ATT1 (integer)	ATT2 (integer)	ATT3 (integer)	ATT4 (integer)	ATT5 (integer)
FORCE	F, N1, N2 or N3	G	CID (see remark 6)			
LOAD	S or Si	0 if LNAME is S, Li otherwise				
MOMENT	M, N1, N2 or N3	G	CID (see remark 6)			

- When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the load bulk data entry. If a DVLREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
- If there are multiple loads that satisfy the designation of the request (for example two FORCE entries with identical SID, GRID and CID values), it is a user input error.
- The default values of LMIN and LMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds
- Input of these data is optional. In most cases, leaving it off will result in a single load that qualifies with the remaining attributes. If there are multiple instances that qualify, it is a user input error.
- The use of this entry is limited to statics and buckling analyses in SOL 200.



## DVMREL1      Design Variable to Material Relation

Defines the relation between a material property and design variables.

### Format:

1	2	3	4	5	6	7	8	9	10
DVMREL1	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	-etc.-		

### Example:

DVMREL1	5	MAT1	1	RHO	0.05	1.0			
	1	1.0							

Describer	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as “MAT1”, “MAT2”, etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as “E” or “RHO”. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 4. (Real)
MPMAX	Maximum value allowed for this property. See Remark 4. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. (If i = 1, Real or Character; if i > 1, Real)

### Remarks:

1. The relationship between the material property and design variables is given by:

$$MP_i = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

4. The default value for MPMIN and MPMAX are not applied when the linear property is a function of a single design variable and  $C0=0.0$ . It is expected that the limits applied to the DESVAR entry will keep the designed property within reasonable bounds.
5. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the material bulk data entry. If a DVMREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.

**DVMREL2****Design Variable to Material Relation**

Defines the relation between a material property and design variables with a user-supplied equation.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVMREL2	ID	TYPE	MID	MPNAME	MPMIN	MPMAX	EQID		
	DESVAR	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	DTABLE	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

**Example:**

DVMREL2	5	MAT1	1	E	0.05	1.0	100		
	DESVAR	1	2						
	DTABLE	E0							

Describer	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a material property entry, such as “MAT1”, “MAT2”, etc. (Character)
MID	Material Identification number. (Integer > 0)
MPNAME	Name of material property, such as “E” or “RHO”. (Character)
MPMIN	Minimum value allowed for this property. If MPNAME references a material property that can only be positive, then the default value for MPMIN is 1.0E-15. Otherwise, it is -1.0E35. (Real)
MPMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
EQID	DEQATN entry identification number. (Integer > 0)
DESVAR	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
DTABLE	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

**Remarks:**

1. The variables identified by  $DVID_i$  and  $LABL_i$  correspond to variable names ( $x_1, x_2$ , etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names  $x_1$  through  $x_N$  (where  $N = m + n$ ) are assigned in the order  $DVID_1, DVID_2, \dots, DVID_m, LABL_1, LABL_2, \dots, LABL_n$ .
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. The fifth field of the entry, MPNAME, only accepts string characters. It must be the same as the name that appears in the [Bulk Data Entries](#) for various material properties. For example, if the isotropic material density is to be designed, RHO (case insensitive) must be specified on the MPNAME field.

**DVPREL1****Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/ FID	PMIN	PMAX	C0		
	DVID1	COEF1	DVID2	COEF2	DVID3	-etc.-			

**Example:**

DVPREL1	12	PBAR	612	6	0.2	3.0			
	4	0.25	20	20.0	5	0.3			

Describer	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as “PBAR”, “PBEAM”, etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If PMIN references a property that can only be positive, then the default value for PMIN is 1.0E-15. Otherwise, it is -1.0E35. See Remark 6. (Real)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E+20)
C0	Constant term of relation. (Real; Default = 0.0)
DVIDi	DESVAR entry identification number. (Integer > 0)
COEFi	Coefficient of linear relation or keyword = “PVAL”. See Remark 7. (If i = 1, Real or Character; if i > 1, Real)

**Remarks:**

1. The relationship between the analysis model property and design variables is given by:

$$P_j = C_0 + \sum_i COEF_i \cdot X_{DVID_i}$$

2. The continuation entry is required.
3. TYPE=“PBEND” is not supported. TYPE=“PBEAML” supports only PNAME and not FID.

4. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in the element property table. For example, to specify the area of a PBAR, either PNAME=A, FID=+4 or FID=-3 can be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
5. Designing PBEAML or PBEAM requires specification of both property name and station. [Table 14](#) shows several examples.

Table 14

PTYPE	Property Name	END A	END B	i-th Station
PBEAML	DIM1	DIM1 or DIM1(A)	DIM1(B)	DIM1(i)
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM or PBEAML entry can be referenced by a DVPREL1. For example, referencing an END B property name on a DVPREL1 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.

6. The default values of PMIN and PMAX are not applied when the linear property is a function of a single design variable and C0=0. It is expected that the limits applied on the DESVAR entry will keep the designed property within reasonable bounds.
7. When "PVAL" is used for the COEF1 field, this is a flag to indicate that the COEF1 value is to be obtained from the property bulk data entry. If a DVPREL1 entry references more than one design variable with the PVAL option, a User Fatal Message will be issued.
8. With GPLY for TYPE field and GPLYID for PID field, a ply identified with GPLYID across all PCOMPG entries in the model can be designed. Internally, a DVPREL1 will be spawned for each PCOMPG has a ply ID of GPLYID. For TYPE=GPLY, the relationship between the analysis model property and design variables is given by.

$$P_i = C0 + (T0_i \text{ or } THETA0_i) \cdot \sum_j (COEF_j \cdot X_{DVID_j}) \text{ for PNAME=T or THETA}$$

Where T0 and THETA0 are value of thickness and theta angle on the original PCOMPG. Note that non-zero C0 is not recommended for TYPE=GPLY or PCOMPG. For THETA0 with original value equal to 0.0, THETA0 is taken as 1.0 and it is recommended to have XINIT of DVID set to 0.0.

**DVPREL2****Design Variable to Property Relation**

Defines the relation between an analysis model property and design variables with a user-supplied equation.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVPREL2	ID	TYPE	PID	PNAME/ FID	PMIN	PMAX	EQID		
	“DESVAR”	DVID1	DVID2	DVID3	DVID4	DVID5	DVID6	DVID7	
		DVID8	-etc.-						
	“DTABLE”	LABL1	LABL2	LABL3	LABL4	LABL5	LABL6	LABL7	
		LABL8	-etc.-						

**Example:**

DVPREL2	13	PBAR	712	5	0.2	0.4	50		
	DESVAR	4	11	13	5				
	DTABLE	PI	YM						

Descriptor	Meaning
ID	Unique identification number. (Integer > 0)
TYPE	Name of a property entry, such as PBAR, PBEAM, etc. (Character)
PID	Property entry identification number. (Integer > 0)
PNAME/FID	Property name, such as “T”, “A”, or field position of the property entry, or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referred to by field position. (Character or Integer ≠ 0)
PMIN	Minimum value allowed for this property. If FID references a stress recovery location field, then the default value for PMIN is -1.0+35. PMIN must be explicitly set to a negative number for properties that may be less than zero (for example, field ZO on the PCOMP entry). (Real; Default = 1.E-15)
PMAX	Maximum value allowed for this property. (Real; Default = 1.0E20)
EQID	DEQATN entry identification number. (Integer > 0)
“DESVAR”	DESVAR flag. Indicates that the IDs of DESVAR entries follow. (Character)
DVIDi	DESVAR entry identification number. (Integer > 0)
“DTABLE”	DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)
LABLi	Label for a constant on the DTABLE or DTABLE2 entry. (Character)

**Remarks:**

1. The variables identified by DVID<sub>i</sub> and LABL<sub>i</sub> correspond to variable names (x<sub>1</sub>, x<sub>2</sub>, etc.) listed in the left-hand side of the first equation on the DEQATN entry identified by EQID. The variable names x<sub>1</sub> through x<sub>N</sub> (where N = m+n) are assigned in the order DVID<sub>1</sub>, DVID<sub>2</sub>, ..., DVID<sub>n</sub>, LABL<sub>1</sub>, LABL<sub>2</sub>, ..., LABL<sub>m</sub>.
2. If both “DESVAR” and “DTABLE” are specified in field 2, “DESVAR” must appear first.
3. FID may be either a positive or a negative number. If FID > 0, it identifies the field position on a property entry. If FID < 0, it identifies the word position of an entry in EPT. For example, to specify the area of a PBAR, either PNAME=A, FID = +4 or FID = -3 may be used. In general, use of PNAME is recommended. For Type "PBUSH", PNAME is recommended, if FID is used it must be < 0.
4. Types “PBEND”, “PBARL” and “PBEAML” are not supported for the DVPREL2.
5. Designing PBEAM requires specification of both property name and station. [Table 15](#) shows one example.

Table 15

PTYPE	Property Name	END A	END B	i-th Station
PBEAM	A	A or A(A)	A(B)	A(i)

Only stations that are input on a PBEAM entry can be referenced by a DVPREL2. For example, referencing an END B property name on a DVPREL2 entry when the referenced PBEAM does not explicitly specify the END B station, is not allowed.



**DVSHAP****Design Variable to Basis Vector(s)**

Defines a shape basis vector by relating a design variable identification number (DVID) to columns of a displacement matrix.

**Format:**

1	2	3	4	5	6	7	8	9	10
DVSHAP	DVID	COL1	SF1	COL2	SF2	COL3	SF3		

**Example:**

DVSHAP	2	1	2.0	4	1.0				
--------	---	---	-----	---	-----	--	--	--	--

Describer	Meaning
DVID	Design variable identification number on the DESVAR entry. (Integer > 0)
COLi	Column number of the displacement matrix. See Remark 2. ( $1 \leq \text{Integer} \leq \text{maximum column number in the displacement matrix.}$ )
SFi	Scaling factor applied to the COLi-th column of the displacement matrix. (Real; Default = 1.0)

**Remarks:**

1. DVID must be defined on a DESVAR entry.
2. COLi must be a valid column number in the displacement matrix.
3. Multiple references to the same DVID and/or COLi will result in a linear combination of displacement vectors. In the example above, the shape basis vector is a linear combination of the fourth column and twice the second column.
4. The displacement matrix must have been created by SOL 101 or 200 with analysis = statics and be available on a database, which is attached via the DBLOCATE FMS statement shown below:

```
ASSIGN    DISPMAT=' physical filename of MASTER DBset '
DBLOCATE DATABLK=(UG/UGD,GEOM1/GEOM1D,GEOM2/GEOM2D) ,
LOGICAL=DISPMAT
```

## DVPSURF Design Variable to Control Surface Setting Relation

Defines the relationship between a control surface setting in a particular subcase and a design variable.

### Format:

1	2	3	4	5	6	7	8	9	10
DVPSURF	ID	AELABEL	TRIMID	DVID	COEF				

### Example:

DVPSURF	10	OBDFLAP	1	100	0.01746				
---------	----	---------	---	-----	---------	--	--	--	--

Describer	Meaning
ID	Unique identification number (Integer>0)
AELABEL	LABEL of the AESURF entry that is being designed (Character ,no default)
TRIMID	Associated trim set identification number (Integer>0)
DVID	DESVAR entry identification number (Integer>0)
COEF	Coefficient of linear relation (Real)

### Remarks:

1. The relationship between the control surface setting and the design variable is given by  $\delta_{SURF} = COEF * X_{DVID}$
2. The surface called out by AELABEL must also appear on the trim entry specified by TRIMID. The value specified on the trim entry will be overwritten by the value obtained from the relationship of Remark 1.
3. Limits on the control deflection are not provided on this entry but can be specified on the underlying DESVAR
4. The DVID called out on this entry cannot be associated with any other designed property.
5. Note that since the DVPSURF calls out a TRIM ID, it is associated only with a single subcase.

## DYFSISW

## Fluid Structure Interaction Activation Switch

Allows activating or deactivating Fluid Structure Interaction and Eulerian solver. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
DYFSISW	SWID	TID	INITV						

### Example:

DYFSISW	14	10							
---------	----	----	--	--	--	--	--	--	--

### Describer

### Meaning

SWID	Unique number of a DYFSISW entry. (Integer > 0; Required)
TID	TABLED1 ID. Fluid Structure Interface and Euler solver are switched on and off, depending on the y-value of the table. The x-value of the table represents the time; the y-value denotes: ON        y > 0.0 OFF       y < 0.0
INITV	Euler element initialization flag. See remark 3. (Character, Default=NO) NO        Element initialization at cycle 0 YES       Element initialization when the elements becomes active

### Remarks:

1. The default is that COUPLE, AIRBAG and Euler solver is active at all times.
2. When Euler solver is deactivated, no output will be generated for the Eulerian elements.
3. For simulations involving a prestress phase the Euler elements can be initially deactivated. When the Euler elements are activated the Eulerian masses still originate from the Euler initiation at cycle 0. If during prestressing the structure did not move much, then these Eulerian masses of cycle 0 can give a stable run. But if there has been substantial movement of the structure then Eulerian masses can be erroneously compressed. This shows up by large pressures and velocities in the Euler elements and a time step too small. To avoid this instability the Euler initialization can be postponed until the Euler elements become active.

**DYPARAM, ATBAOUT**      Output Frequency to Main Output File of ATB - SOL 700

Defines the frequency at which output is written to the main output file of ATB. Used in SOL700 only.

**Format:**

DYPARAM, ATBAOUT, value

**Example:**

DYPARAM, ATBAOUT, 5.0E-3

Option	Content
Value	Every multiple of ATBAOUT seconds, the main output file of ATB is updated. (Real > 0.0; default=10.0E-3)

**Remarks:**

1. Only active when field 3 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of the output of segment acceleration, velocity and displacement, joint forces and moments.

## DYPARAM, ATBHOUT

### Write ATB Output to Time history files - SOL 700

A time-history file is created containing the output as requested in the ATB input file on cards H.1 to H.11. Used in SOL700 only.

#### Format:

DYPARAM, ATBHOUT, option

#### Example:

DYPARAM, ATBHOUT, NO

Option	Content
Option	Flag of ATBHOUT. (Character; default=YES)
	YES: The time history files are created.
	NO: The time history files are not created.

## **DYPARAM, ATBTOUT**

### Output Frequency to Time-History Files of ATB - SOL 700

Defines the frequency at which output is written to the time-history files of ATB. Used in SOL700 only.

#### **Format:**

DYPARAM, ATBTOUT, value

#### **Example:**

DYPARAM, ATBTOUT, 1.0E-4

Option	Content
Value	Every multiple of ATBTOUT seconds, the time-history files of ATB are updated. (Real > 0.0; default=1.0E-3)

#### **Remarks:**

1. Only active when field 26 on the A5 card of the ATB input file is set to a value of -1.
2. Controls the frequency of all output requested on the H-cards, and of the tabular time-histories that are controlled by field 18 on the A5 card of the ATB input file.

## DYPARAM,AUTOCOUP

## Automatic Coupling

Defines the automatic coupling algorithm. Used in Sol700 only.

### Format:

DYPARAM,AUTOCOUP,ACTIVE,CLEAN,DUMMY,TOL\_AREA, OUTPUT, FAIL

### Example:

DYPARAM, AUTOCOUP, ON

Describer	Meaning
ACTIVE	Flag of activation of the sub element approach. (Character; default=OFF)
	ON                    Turn on the sub element approach.
	OFF                   Turn off the sub element approach.
CLEAN	Flag of cleaning obsolete sub elements. (Character; default=OFF)
	ON                    Cleans obsolete sub elements
	OFF                   Obsolete elements will not be removed from memory.
DUMMY	Flag of putting dummy segments on coupling surface archives. (Character; default=OFF)
	ON                    Puts dummy segments on the coupling surface archive
	OFF                   Does not put dummy segments on the coupling surface archive.
TOL_AREA	If the area of a hole is smaller than TOL_AREA a 1D flow computation method is used. (Real $\geq$ 0.0; default=0.0).
OUTPUT	Determines how results of Euler elements that are intersected by the structure are written to the Euler archive. (Character, Default = ZERO).
	ZERO                   Write zero for the results of these intersected elements.
	AVERAGE            Average across sub elements. Each sub element has the same weight.
	AVEREAGEU          Average across sub element using the uncover fraction as weights.
	MAXUNC              The results are taken from the sub element with the largest volume uncover fraction.
	COVER                Uses the cover field of COUPLE to select the proper sub element. Only supported for COVER=OUTSIDE or COVER=INSIDE. Not supported for COVER =NONE.
FAIL	Activates fluid flow through failed shell segments in the coupling surface. (Character; there is no default, if failure is not used, then FAIL should be left blank)
	POROUS              Makes failed segment porous. This is the fast coupling approach.
	REMOVE              Removes the failed segments from the coupling surface.

**Remarks:**

1. Features that are not supported are:
  - Multiple Euler domain
  - Adaptive Euler meshes
  - Fluid flow through failed segments in the coupling surface failure can only be defined by option FAIL of DYPARAM, AUTOCOUP. Defining this failure by use of COUP1FL and COUPINT is not supported.
  - COUPLE with option AIRBAG
  - Graded meshes
  - Euler import
  - Viscosity
  - Porosity
  - Option FAIL=POROUS and REMOVE are not supported by DMP.
  - Markers
2. With automatic coupling fluid can be defined on both sides of the coupling surface. To do this option COVER of the COUPLE entry has to be set to NONE. When using COVER = OUTSIDE fluid is only initialized in the inside region of the coupling surface. For COVER = INSIDE the opposite applies. When the coupling surface is not closed, only COVER=NONE is allowed.
3. In the auto coupling approach holes in the coupling surface are meshed with dummy segments. For holes with large deformations PARAM DUMSEGS, ON can be used to maintain good dummy segments.
4. For adding FAIL = POROUS failed segments have to be made fully porous. This is straightforward.
5. FAIL = REMOVE. When the structure fails, holes in the coupling surface are formed. For a closed coupling surface each side of a segment is connected to another segment. Therefore, the segments have no free edges. But a segment adjacent to a hole has a free edge. At this edge it is not connected to any other segment. A list of free edges of the coupling surface specifies the holes. By maintaining a list of free edges auto coupling handles these holes automatically. If a segment fails, this list of free edges has to be updated. Also the failed segment has to be removed from the coupling surface.

**Note:**

This entry also supports DMP. While using for DMP, no changes to input is needed. In the auto coupling method Euler elements that are intersected by the structure are split into sub elements. For details refer to [1]. To enable dmp for auto coupling some of the properties of sub elements have to be communicated across CPU's. If holes in the coupling surface extend across multiple CPUs, also communication across CPU's is needed.



## DYPARAM,AXIALSYM

## Axial Symmetric Analyses - SOL 700

Enables an efficient and accurate 2-D axial symmetry for Eulerian materials. A much larger time step becomes possible by not taking into account the mesh-size in circumferential direction. Used in SOL 700 only.

### Format:

DYPARAM,AXIALSYM,MESHTYPE,AXIALAXIS,SYMPLAN,PHI,ALIGN,PHI2

### Example:

DYPARAM,AXIALSYM,RECT,X,XY,2.5,YES,0.0

Describer	Meaning
MESHTYPE	Two types of Euler meshes are supported: (Character; Required) AXIAL      Axial symmetric meshes. RECT        Rectangular meshes
AXIAL AXIS	X            X-Axis (Character; Required) Y            Y-Axis Z            Z-Axis
SYMPLAN	The approximate symmetry plan of the Euler mesh. On for MESHTYPE=AXIAL. See Remark 7. (Character; Required) XY          XY-Plane YZ          YZ-Plane ZX          ZX-Plane
PHI	Only used for MESHTYPE = RECT. Used to create a 2D axial symmetric mesh with angles +PHI/2 and -PHI/2. (Real; Default = 0.0)
ALIGN	Only used for MESHTYPE = AXIAL. (Character; Default = YES) YES         Align normals of oblique Euler element faces. This prevents errors in strains that can arise from small errors in Euler face normals. NO          Do not align normals.
PHI2	As a final operation rotate the mesh around the axial axis by the angle PHI2. See Remark 6. (Real; Default = 0.0)

### Remarks:

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.

2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one layer can be used. Using the angle specified by PHI this Euler mesh is mapped into a 2d axial symmetric mesh.
3. The Euler mesh has to consist of one layer.
4. Rectangular meshes that can be made 2d symmetric using the angle PHI should satisfy:
  - All boundary Euler faces are aligned with a coordinate direction
  - Only one layer thick.
  - The axial symmetry axis is either on the boundary of the Euler mesh or outside the Euler mesh. It is not allowed that the axial axis is inside the Euler mesh.Initialization of Euler element using geometric regions as defined by the TICEUL entry is carried out onto the transformed 2d axial mesh.
5. In the time step computation the circumferential mesh-size will not be taken into account.
6. Use option PHI2 with caution. Euler initialization is done using the mesh rotated by the angle PHI2. So after including the angle PHI2 or modifying its value the Euler initialization should be revised.
7. It is assumed that one of the coordinate planes is an approximate symmetry plane of the Euler mesh. Although approximate symmetry is sufficient, the coordinate plane can always be made an exact symmetry plane by the use of PHI2. If for example the Euler mesh has angles 0 and 2.5, PHI2 has to be set to -1.25 to get exact symmetry.

## DYPARAM,AXREMAP

## 2D Axial symmetric Euler archive remap - SOL 700

Allows import of a 2D axial symmetric Euler archive into a 3D simulation.

### Format:

1	2	3	4	5	6	7	8	9	10
DYPARAM	AXREMAP	x0	y0	z0	xn	yn	zn	range	

### Example:

DYPARAM	AXREMAP	0.0	0.5	0.5	1.	0.	0.		
---------	---------	-----	-----	-----	----	----	----	--	--

Describer	Meaning
x0,y0,z0	x,y,z-coordinate of the point at which the 2D axial symmetric mesh is remapped. (Real; Default = 0.0)
xn,yn,zn	Unit vector that specifies the direction of the axial axis of the 2D axial symmetric mesh as viewed in the 3D mesh. (Real; Default = (1.0, 0.0, 0.0) )
range	Only material whose distance from the axial axis is smaller than "range" will be initialized with the 2D axi-symmetric Euler archive. (Real, Default = 1e+20)

### Remarks:

1. Since 2D axial symmetric simulations run much faster than 3D simulation it can save much cpu time to do the first part of the simulation with a 2D axial symmetric mesh. Afterwards the 2D-axial symmetric Euler archive is imported into the 3D simulation. By default the 2D axial symmetric archive will not be expanded in 3D. To enable this expansion DYPARAM, AXREMAP has to be used. It is useful for blast wave simulations. The 2D axial symmetric simulation has to be terminated before the blast wave approaches any 3D structure.
2. This import of Euler archives is done by means of the eid option in the pth file.
3. To generate axial symmetric meshes DYPARAM,AXIALSYM can be used.

## **DYPARAM,BULKL**

## Linear Bulk Viscosity Coefficient- SOL700

Defines the default value of the linear bulk viscosity coefficient.

### **Format:**

DYPARAM, BULKL, VALUE

### **Example:**

DYPARAM, BULKL, 0 . 1

<b>Describer</b>	<b>Meaning</b>
value	Value of the linear coefficient in the bulk viscosity equation. (Real $\geq$ 0.0; default=0.0)

### **Remarks:**

1. The default value works well for the majority of problems.
2. The value defined on this entry is used as the default whenever BULKL is blank on the MATBV material entry.
3. When BULKL is specified on a material definition entry, the default value is overridden for that specific material.

## DYPARAM,BULKQ

## Quadratic Bulk Viscosity Coefficient - SOL700

Defines the default value of the quadratic bulk viscosity coefficient.

### Format:

DYPARAM, BULKQ, VALUE

### Example:

DYPARAM, BULKQ, 1.6

Describer	Meaning
value	Value of the quadratic coefficient in the bulk viscosity equation. (Real $\geq$ 0.0; default=0.0)

### Remarks:

1. The default value works well in the majority of situations.
2. The value defined on this entry is used as the default whenever BULKQ is blank on the MATBV material entry.
3. When BULKQ is specified on a material definition entry, the default value is overridden for that specific material.

## **DYPARAM,BULKTYP**

Bulk Viscosity Type - SOL700

Defines the default type of bulk viscosity.

### **Format:**

DYPARAM, BULKTYP, option

### **Example:**

DYPARAM, BULKTYP, DYNA

<b>Describer</b>	<b>Meaning</b>
Option	bulk viscosity type. (Character; Default=DYNA)
DYNA	Standard DYNA3D model
DYTRAN	Enhanced DYNA model

### **Remarks:**

## DYPARAM,CFULLRIG

Converts 123456 Constraints to FULLRIG on RBE2 - SOL700

Converts all 123456 constraints to the FULLRIG option on all entries. Used in SOL 700 only.

### Format:

DYPARAM,CFULLRIG,value

### Example:

DYPARAM,CFULLRIG,NO

Describer	Meaning
value	Activation flag. (Character; Default=YES)
YES	123456 constraints are converted to FULLRIG.
NO	123456 constraints are not converted to FULLRIG.

## DYPARAM,CLUFLIM

## Limiter of Volume Stain Rate for Clumps - SOL700

In some cases, airbag runs become instable. Often, this is caused by a much too large volume strain rate in a clump that consists of too many elements. These clumps typically have a small average volume uncovered fraction. The large volume strain rate causes a huge compression work and this blows up the specific internal energy. When this happens it is clearly visible in the OUT file and in the results. This DYPARAM activates a limiter that scales down the volume strain rate for clumps with a small average uncovered fraction. It can keep an instable airbag run stable, just like PARAM, VELMAX can keep runs stable. Used in SOL 700 only.

### Format:

DYPARAM, CLUFLIM, value

### Example:

DYPARAM, CLUFLIM, 0.22

Describer	Meaning
value	The volume strain rate in a clump will be reduced when the average Uncovered Fraction of elements in a CLump falls below CLUFLIM. CLUFLIM has to be smaller than FBLEND. The default value of FBLEND is 0.66 giving a value of 0.22 for CLUFLIM. For more details on FBLEND refer to PARAM,FBLEND. (0<Real<FBLEND; default=FBLEND/3)

### Remarks:

- The Volume strain rate  $\frac{DIV}{\Delta t}$  in clumps will be limited by

$$\left(\frac{DIV}{\Delta t}\right)_{Lim} = MIN\left(1, \frac{U}{CLUFLIM}\right)\left(\frac{DIV}{\Delta t}\right)$$

Here, U is the average uncovered fraction of elements in the clump as given by

$$U = \frac{\sum_{eleclump} Uncf_{el} \cdot Vol_{el}}{\sum_{eleclump}}$$

Here Uncf and Vol denote the uncovered fraction and volume of an element inside the clump.

Therefore, only when the average uncover fraction falls below CLUFLIM, the volume strain rate is limited.



**DYPARAM,CLUMPENR**

Switch for Kinetic Energy Calculation Scheme of Blended Clumps - SOL700

Sets the definition of the kinetic energy calculation method for Eulerian blended clumps. Used in SOL 700 only.

**Format:**

DYPARAM, CLUMPENR, value

**Example:**

DYPARAM, CLUMPENR, SUM

Describer	Meaning
value	Flag for CLUMPENR.(Character; default=AVERAGE)
	<p><b>AVERAGE</b>     The kinetic energy of a Eulerian blended clump is calculated from the average velocity of the clump. The average velocity of the blended clump is computed as the sum of the momentum of each member of the clump divided by the total clump mass.</p>
	<p><b>SUM</b>             The kinetic energy of a Eulerian blended clump is calculated as the sum of the kinetic energy of each member of the clump.</p>

**DYPARAM,CONM2OUT**

CONM2 Summary Output - SOL700

Determines if a summary of concentrated masses and their energy and momentum is written to the output file. Used in SOL 700 only.

**Format:**

DYPARAM, CONM2OUT, value

**Example:**

DYPARAM, CONM2OUT, NO

Describer	Meaning
value	Flag for CONM2OUT. (Character; default=NO)
NO	No information about concentrated masses is written to the cycle and material summaries on the output file.
YES	A complete summary of concentrated masses including the associated mass, momentum, and energy is written to the output file.

**Remarks:**

1. When DYPARAM,CONM2OUT is set to NO, there is no summary of the concentrated mass. This means that the mass, momentum, and energy of the concentrated masses, is not added to the material and cycle summaries. Setting DYPARAM,CONM2OUT,NO saves memory and CPU time.

## DYPARAM,CONTACT

## Sets Defaults for CONTACT - SOL700

Defines certain defaults for the contact definitions. Used in SOL 700 only.

### Format:

DYPARAM, CONTACT, option, value1, value2, value3, value4, value5, value6

### Example:

DYPARAM, CONTACT, VERSION, V4

Describer	Meaning
VERSION	Defined the default version of contact. Only support value1. (Character; required) V4 General Contact algorithm of SOL700
BELT	Suited for modeling contact between a belt element and a rigid structure. Primary secondary contact only. The contact logic doesn't apply a contact force, but applies an enforced displacement and velocity that keeps the secondary nodes exactly on top of the primary face. The secondary node does not slide relative to the primary face when the friction coefficient (FS) is set to 1E20.
BELT1	Identical to BELT algorithm, except that the secondly nodes are initially repositioned on top of the closest primary face.
DRAWBEAD	Suited for modeling a drawbead.
THICK	Defines the default value for THICK. Only support vaule1. (Real>0.0; required.)
GAP	Defines the default value for GAP. Only support vaule1. (Real>0.0; required.)
LIMITS	Definition of a three dimensional contact region where contact in the analysis model takes place. Significant CPU time savings can be achieved when used in adaptive contact. 6 values are used to define area.  The format of this option is DYPARAM,CONTACT,LIMITS,xmin,xmax,ymin,ymax,zmin.zmax. (Real; default=1.0E20 for xmax, ymax and zmax and -1.0E20 for xmin, ymin and zmin) XMIN(value1) Lower limit in x-direction where main contact occurs XMAX(value2) Upper limit in x-direction where main contact occurs YMIN(value3) Lower limit in y-direction where main contact occurs YMAX(value4) Upper limit in y-direction where main contact occurs ZMIN(value5) Lower limit in z-direction where main contact occurs ZMAX(value6) Upper limit in z-direction where main contact occurs
DAMPING	Defines the default for usage of damping. Only support value1. (Character; required) YES Use damping in CONTACT

Describer	Meaning
	NO Not use damping in CONTACT
COPOR	Activates contact based porosity. Only support value1. (Character; default=NO)
	YES Use porosity in CONTACT
	NO Not use porosity in CONTACT
DAMPFOR	Defines whether the noncontact forces acting on the grid points need to be taken into account in the contact damping. This option is only used if DAMPING is set to YES. This option prevents large penetrations that might occur when the forces acting on the grid points tend to push them into the contact surface. This happens, for example, in airbag analyses, where a large pressure exists inside the bag. Only support value1. (Character; required)
	YES Damping is considered in noncontact forces
	NO Damping is not considered in noncontact forces
DYNA	The following parameters of the contact definition get the default values consistent with Dyna. (Character ; required)
	THICK 1.0
	THICKOF 0.0
INFO	Information on the contact state of grid points G1,G2,... is printed to ASCII files, named CNT. This information can be useful in debugging models with contacts. Supports many values as you want. (Integer > 0; required.)
EVIEW	Defines the default value of the view angle of Edge to Edge contacts. The value of the angle must be in degrees. Only support value1. (Real > 0.0; required)
FORCE	Controls the contact forces on the grid points. Supports value1, value2, value3. See Remark 1. (value1: Integer > 0; default=10, value2: Real > 0; default=10.0; value3: Character; default=ZERO)
	NMCYC Frequency check of contact force; applied on each grid point. (Integer > 0; (value1) default = 100).
	SCALE Scale factor for maximum allowable contact force. (value2) $F_{max} = SCALE \cdot F_{last\ check}$ (Integer > 0; default = 10)
	TYPE Contact force limitation. (Character; default=ZERO) (value3)
	ZERO 0.0
	FMAXF Update contact limit forces using new contact forces (where non allowable forces are not taken into account)

**Remarks:**

1. The DYPARAM, FORCE check takes up some CPU time and, therefore, do not make this value too small. Furthermore, when the check is performed at each cycle, the force will be too limited and the bag will not unfold. Recommended values are between 5 and 200. The same problems can occur for SCALE (value2). In case this value is too small, the bag will not unfold either. The minimum value for air bags that should be used is about 5. The maximum is about 20. When this value is too big a difference will not be noticed. TYPE (value3)=ZERO is a bigger restriction. In some cases, TYPE=FMAXF might yield better results.

## **DYPARAM,COSUBMXT**

Time-dependent Subcycle Limit in Euler/Lagrange Coupling - SOL700

Defines the maximum number of subcycles that can occur in Euler/Lagrange coupling. During a subcycle, the geometry of the coupling surface is not updated. This number can vary in time and is given by a table. Used in SOL 700 only.

### **Format:**

DYPARAM, COSUBMXT, value

### **Example:**

DYPARAM, COSUBMAT, 10

<b>Describer</b>	<b>Meaning</b>
value	TABLED1 ID that specifies for each time the maximum number of time steps between updating the coupling surface geometry in the coupling calculations. (Integer > 0; required)

### **Remarks:**

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of this parameter, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. If the geometry of the coupling surface is changing rapidly, smaller values on the table should be used.

## DYPARAM,COUFRIC

### Coupling Surface Friction for Nonmetallic Eulerian Solids - SOL700

Defines the Coulomb friction scheme. Please check DYPARAM,COHESION too. Used in SOL 700 only.

#### Format:

DYPARAM, COUFRIC, value

#### Example:

DYPARAM, COUFRIC, NO-METAL

Describer	Meaning
value	Flag for COUFRIC usage. (Character; default=METAL)
METAL	A tensile condition will result in zero load on the structure part on cohesive coupling.
NO-METAL	a tensile load was applied in tensile condition on cohesive coupling.

#### Remarks:

1. Only used when Coulomb friction coefficients have been specified for a COUPLE entry.

**DYPARAM,COHESION**

Cohesion for Coulomb Friction - SOL 700

Allows friction and sticking during tensile conditions at the coupling surface.

**Format:**

DYPARAM,COHESION,MAXSTRS,FRIC,REFVEL

**Example:**

DYPARAM,COHESION,8.0e+10,8.0e+5,20.0

Describer	Meaning
MAXSTRS	Maximal normal stress. Allows tensile stresses at the coupling surface as long as the normal stress does not exceed MAXSTRS. (Real > 0)
FRIC	Friction stress under tensile conditions. (Real > 0)
REFVEL	Reference value for velocity. (Real > 0)

**Remarks:**

1. Only used when coulomb friction coefficients have been specified for a COUPLE entry.
2. During tension any relative tangential velocity between coupling surface and Eulerian material will yield a shear stress whose magnitude equals:

$$FRIC \cdot \min\left(1, \frac{V_{rel,tangential}}{REFVEL}\right)$$

This is a viscous-like friction law.

3. This shear force opposes the relative tangential movement along the coupling surface.



## DYPARAM,DMPOPT

Sets DMP option - SOL700

Sets DMP option. Used in SOL 700 only.

### Format:

PARAM,DMPOPT,value1,value2,value3,value4

### Example:

PARAM,DMPOPT,2,1

Describer	Meaning
Value1	Set dmp option. (Integer>0, default=1) 1 Only Euler dmp is activated, 2 Only Lagrangian dmp is activated. 3 Both, Euler and Lagrangian dmp are activated.
Value2	Set contact dmp type. (Integer>0, default=1). 1 Serial mode in contact 2 Parallel mode in contact (check value4)
Value3	Set number of printing summary for each item. (Integer>0, default=0). 0 Print all summaries (see Remark 2.)
Value4	Set parallel mode type in contact. (Integer>0, default=not used). -1 Use the same contact domain partitioning as the element domain partitioning and all primary surfaces must be made by MATRIG definition (see Remark 3.)

### Remarks:

1. To set partitioning manually, check DYPARAM,EULERPR for Euler partitioning and DYPARAM,LAGPR for Lagrangian partitioning.
2. Printing out too large summaries of each item can cause unexpected failure of job under certain conditions. This option can limit to certain number of summary information to avoid such failure.
3. When the same contact domain partitioning as the element domain partitioning is used, the dmp performance of contact may not be enhanced. However, when all deformable grid points are assigned in secondary body and they are involved to contact at the similar time, the dmp performance of contact will be enhanced as expected. For example, stamping of sheet metal may show a good performance.

## **DYPARAM, DUMSEGS**      Creating dummy segments for the auto coupling approach - SOL700

Specifies how holes in the coupling surface are meshed. It is only used for the auto coupling approach. This approach is activated by DYPARAM, AUTOCOUP.

### **Format and Example:**

DYPARAM,DUMSEGS,INTERNAL

Describer	Meaning
INTERNAL	Meshes internal holes with a triangular dummy segment mesh [YES,NO] Default NO

### **Remarks:**

1. With the auto coupling method, holes are partially or completely meshed with dummy segments. With option NO the standard method of creating dummy segment is used. For details see chapter 8 of the SOL700 User's Guide. In this standard method holes are partially meshed with dummy segments. For an example refer to Figure 20 of Chapter 8. For holes that rotate, have arbitrary form or show large deformations, the standard method of creating dummy segments can cause the simulation to terminate with messages like

```
%E-P4307807-P4_SUBELM_MESH_CURVED_HOLES, ,(13), DUMMY SURFACE
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION
PROBLEM.
```

```
%E-P4309907-V4_SUBELEM_CONNECT_EXTENDED_GPS, , DUMMY SURFACE
CANNOT BE TRIANGULATED THIS IS PROGRAM ERROR. PLEASE CONTACT
MSC. USING PARAM,DUMSEGS,ON CAN SOLVE THIS TRIANGULATION
PROBLEM.
```

In that case PARAM,DUMSEGS,ON can maintain a good dummy surface. The quality of the dummy surface can be checked by requesting a coupling surface archive. This is done by using the CPLSURFS command and by putting option DUMMY of DYPARAM, AUTOCOUP to ON. For all holes that are not internal holes, the standard method is used.

**DYPARAM,ELDLTH**

- SOL700

Print initial time step sizes for elements in the first cycle in SOL 700.

**Format:**

DYPARAM,ELDLTH,<value>

**Example:**

DYPARAM,ELDLTH,1

Describer	Meaning
value	Number of elements to be output. (Integer)
0	First 100 elements with the smallest time step sizes are printed.
1	The governing time step sized for all elements are printed. (Default)

**DYPARAM,EULERCB**

Euler Domain Decomposition Control - SOL700

Divides a Euler domain into several cubes. For SOL 700 only

**Format:**

DYPARAM,EULERCB,NELCUBE,NBX,NBY,NBZ

**Example:**

DYPARAM,EULERCB,2000,2,2,2

Describer	Meaning
NELCUBE	The number of elements per cube. (Integer; Default = 0)
NBX	The number of cubes in the x-direction. See Remark 2. (Integer; Default = 0)
NBY	The number of cubes in the y-direction. (Integer; Default = NBX)
NBZ	The number of cubes in the z-direction. (Integer; Default = NBX)

**Remarks:**

1. By setting NELCUBE equal to 2000 optimal use is made of memory caching during Euler computation. This may give a speedup of 1.5. When using adaptive meshing with dmp additional Euler cubes are created during the simulation. To keep the total number of cubes that are created within bounds the initial number of cubes should be limited to 100.
2. Defining NBX overrules the definition of NELCUBE.
3. Only supports Euler domains created by MESH,BOX. Limitations are:
  - No PEULERx/CHEXA's
  - No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.

## DYPARAM,EULERPR

## Euler Cube Partition Control - SOL700

Divides a Euler domain into several cubes. For SOL 700 only

### Format:

DYPARAM,EULERPR,PROCDIR,NPX,NPY,NPZ

### Example:

DYPARAM,EULERPR,USER,2,2,2

Describer	Meaning
PROCDIR	This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Character; Default = X) X            Partition in global X direction first. Y            Partition in global Y direction first. Z            Partition in global Z direction first. USER        Define user defined partitioning SIMPLE      Partition Euler cubes in a simple pattern. ORB         Partitions using orthogonal recursive bisection
NPX	The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer; Default = 0)
NPY	The number of cubes in the y-direction. (Integer; Default = NPX)
NPZ	The number of cubes in the z-direction. (Integer; Default = NPX)

### Remarks:

1. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.
2. When option PROCDIR=USER, the values for NBX, NBY and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option, NPX\*NPY\*NPZ must be equal to the number of processors used in the cluster.
3. For option PROCDIR=SIMPLE, the values NBX, NBY and NBZ on DYPARAM,EULERCUB must be such that NBX\*NBY\*NBZ is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4, NBX\*NBY\*NBZ must be equal to either 4 or 8 or 12, etc. Otherwise the calculation will terminate prematurely with an error message.
4. Only supports Euler domains created by MESH,BOX. Limitations are:
  - No MESH,ADAPT and no PEULERx/CHEXA's

- No FLOW or BARRIER that use the BCID option are allowed. All FLOW and BARRIER entries must use either the DIR option or the XMIN,YMIN,ZMIN, XMAX,YMAX,ZMAX option to define the Euler MESH boundaries.
5. With the orthogonal recursive bisection method the mesh is repeatedly cut in half. Each cut is along a coordinate direction and the direction of the cut is chosen to minimize the communication cost. To illustrate this method consider for example a mesh with 200,150 and 50 elements in respectively the x,y and z-direction. When split in x-direction the number of Euler faces at the split will be  $150 \cdot 50 = 7500$ . When splitting across the y-direction it will be  $200 \cdot 50 = 10000$  and for the z-direction it will be  $200 \cdot 150 = 30000$ . Since the x-split has the smallest number of faces at the split, the x-split has the smallest communication cost. Therefore the ORB scheme will select the x-split. At the x-split there are two mesh parts. Both have  $100 \cdot 150 \cdot 50$  elements. Now a split in y-direction will give the minimal communication costs. This process of bisecting is continued until the number of sub meshes equals the number of CPU's.

## DYPARAM,EULTRAN      Switch for Euler Transport Scheme of the Multi-Material Solver and the Single Material Strength Solver - SOL700

Sets the definition of the face velocity used in the transport scheme of the Multi-material solver and the single material strength solver. Used in SOL 700 only.

### Format:

DYPARAM,EULTRAN,option1,option2

### Example:

DYPARAM,EULTRAN,AVERAGE,FAIL

Describer	Meaning	
Option1	IMPULSE	The face velocity is impulse weighted. (Character; Default = IMPULSE)
	AVERAGE	The face velocity is a simple average.
Option2	NOFAIL	Failure is transported. See Remark 5. and 6. (Character; Default = NOFAIL)
	FAIL	Failure is transported. See Remark 5. and 6.

### Remarks:

1. The default value of IMPULSE is sufficient for most Euler problems. Especially problems where the reference density of the different materials varies widely (e.g., orders of magnitude) are required to use the default option.
2. In case the IMPULSE option (default) is used, the Euler transport scheme computes that the face velocity uses an impulse weighted average of the material velocity in the left and the right element adjacent to the face.
3. In case the AVERAGE option is used, the Euler transport scheme computes the face velocity as one-half times the sum of the material velocity in the left and the right element adjacent to the face.
4. Does not apply to the single material hydrodynamic solver and the Roe solver.
5. The option FAIL requires a failure model for at least one Eulerian material. In case of the default NOFAIL then failed Euler material can support shear stress again as soon as new material enters the Euler element. Thus the information that part of the material inside the Euler element has failed is lost. The option FAIL activates transport of fail fraction and thereby keeps track of material that has failed. In this way only the failed part of the element can no longer supports shear stresses. In more detail, the yield stress in the material is scaled by  $(1 - \text{failfrac})$ , where failfrac denotes the fail fraction of the material. The fail fraction of the first material in an element can be retrieved from Euler archive or time-history results files in the variable DAMAGE. The value of fail fraction DAMAGE is between zero and one.
6. Option FAIL cannot be combined with the Johnson-Cook failure model (FAILJC).

## DYPARAM,EUSUBMAX

- SOL700

Defines the maximum number of subcycles that can occur in the Euler solver. During a subcycle, the Euler computations are skipped. Used in SOL 700 only.

If coupling surface computations are more expensive than Euler computations than use of PARAM,COSUBMAX should be considered first. In that case the optimal setting of EUSUBMAX is LINKCS=BOTH and DFVUMAX is blank.

As with PARAM,COSUBMAX use of EUSUBMAX can lead to loss of accuracy for certain simulations. It is recommended to validate the use of EUSUBMAX by comparing the difference in results between using EUSUBMAX and not using EUSUBMAX for some typical target simulations.

### Format

DYPARAM,EUSUBMAX,NSUBMAX,DFVUMAX,LINKCS

### Example:

DYPARAM,EUSUBMAX,5

Describer	Meaning
NSUBMAX	NSUBMAX, The maximum number of time steps between updating Euler variables. (Integer $\geq 0$ ; Default = 0)
DFVUMAX	Maximum allowed increase in uncovered volume fraction between to subsequent Eulerian computations. (Real $> 0$ ; Required when COSUBMAX is present. Leave blank when COSUBMAX is not present.)
LINKCS	Specifies interactions between Euler subcycling and coupling surface subcycling. EUSUBMAX can be used with COSUBMAX. LINKCS specifies how the two subcycling processes influence each other. (Character; Default = BOTH)
BOTH	The Euler computations will not be skipped when the coupling surface computations has been done. On the other hand if an Euler computation is to be done, also a coupling surface computation is done. The number of time that the Euler computation I skipped equals the number of times that the coupling surface computation has been done.
COUPLE	If an Euler computation is to be done, also a coupling surface computation is done.
EULER	If a coupling surface computation is to be done, also a Euler computation is done.
NONE	The Euler subcycling and coupling surface subcycling are independent. DFVUMAX has to be defined.



**Remarks:**

1. Updating the coupling geometry takes a lot of CPU time. Subcycling gives substantial savings in CPU time for coupled calculations.
2. The smaller the value of DFBVUMAX, the greater the accuracy of the analysis and the greater the cost. Conversely, larger values offer significant CPU savings, but very large values give incorrect results.
3. When the Euler time step is considerably larger than the Lagrange time step it can be worthwhile to skip the Euler computation for several cycles. This can reduce computational costs considerable.
4. Skipping the Euler computation for several cycles, leads to postponed time steps. When the Euler computations are done these have to be taken into account by fluxing with an accumulated time step. Skipping the Euler computation is stopped as soon as the accumulated time step will become larger than the stable Euler time step. The total number of skipped cycles is limited by NSUBMAX. NSUBMAX is required input.
5. In skipping the Euler computation also the movement of the coupling surface has to be monitored. To estimate this movement the change in uncovered fraction of the elements is used. If the change in uncovered volume fraction from one cycle to the other is larger than DFBVUMAX than the Euler computation is not skipped that cycle. DFBVUMAX has to be left blank if coupling surface subcycling is used. Coupling surface subcycling already monitors the movement of the coupling surface. Choosing too large values for DFBVUMAX can make results inaccurate.
6. EUSUBMAX is only supported by the multi-material Euler solver and the standard single material Euler solver. It is not supported by the Roe solver.

## **DYPARAM,EUSUBCYC**

## Growth or Subcycling Interval in Euler Computations - SOL700

Controls the growth of the subcycling interval in Euler computations. Used in SOL 700 only.

### **Format:**

DYPARAM,EUSUBCYC,value

### **Example:**

DYPARAM,EUSUBCYC,100

<b>Describer</b>	<b>Meaning</b>
value	Maximum growth of the subcycling interval. (Integer > 0; default=1)

### **Remarks:**

1. The subcycling algorithm automatically estimates the number of subcycles to be used. This is updated throughout the calculation. This parameter controls how much the number of subcycles can grow. For example, EUSUBCYC is set to 1, and the current number of time steps between updates of the Euler variables. If Dytran estimates that the subcycling interval should be 7, the subcycling interval is increased by 1 until a value of 7 is reached.
2. There is no control on the amount by which the subcycling interval can decrease.

## DYPARAM,FAILDT

## Element Time-step Based Failure Model - SOL700

Defines the property of a failure model where element failure occurs when the element's time step falls below the specified limit. Used in SOL 700 only

### Format:

DYPARAM, FAILDT, value

### Example:

DYPARAM, FAILDT, 1.0E-3

Describer	Meaning
value	Minimum time-step. (Real > 0.0; default=1.0E-20)

### Remarks:

1. This failure model is valid for all Lagrangian solid (CHEXA) and shell (CQUAD4) elements.
2. All elements for which the time step falls below the specified value are removed from the computation.
3. Although it is not usually necessary to limit the element time-step for Lagrangian elements, there are occasions where specifying a minimum time-step can be advantageous for computational performance, for example, when adaptive contact is used.
4. Note that this parameter should be used with care as you may influence the results of the analysis when you set the time-step criterion to a too high value. You then run the risk that elements are removed from the analysis while they may still be relevant.

## **DYPARAM,FAILOUT**

Failed Element Output Parameter - SOL700

Defines whether failed elements are written to the output file (ARCHIVES). Used in SOL700 only.

### **Format:**

DYPARAM, FAILOUT, value

### **Example:**

DYPARAM, FAILOUT, NO

<b>Describer</b>	<b>Meaning</b>
value	Flag for FAILOUT. (Character; default=YES)
	NO            Failed elements are not written to the archive files.
	YES          Failed elements are written to the archive files.

### **Remarks:**

1. When the NO option is chosen, the archives are written out as one file per requested time step regardless of the number set in the SAVE command for the archive files that appear in the Case Control Section. Please check \$\$700.

**DYPARAM, FLOWMETH****Flow-Method Between Two Euler Domains Across Open Areas in Coupling Surfaces - SOL700**

Defines the method for simulating material flow between two Euler domains across open areas in coupling surfaces. Used in SOL 700 only.

**Format:**

DYPARAM, FLOWMETH, value

**Example:**

DYPARAM, FLOWMETH, POLPACK

Describer	Meaning
value	Flag for FLOWMETH. (Character; default=NO)
POLPACK	The facets in the coupling surfaces that represent an open area are subdivided into smaller facets, with each connecting exactly to one Euler element in the first Euler domain and to exactly one Euler element in the second Euler domain. Material flow takes place across these smaller, subdivided facets (POLPACKs). This is the most accurate method.
FACET	The facets in the coupling surfaces that represent an open area are not subdivided. Material flow takes place across the original facets. If these facets are too large in relation with the Euler elements, the method becomes inaccurate. Material flow across one facet can involve several Euler elements on both sides of the hole and averaging occurs.

**Remarks:**

- This parameter applies to simulations where:
  - Two coupling surfaces share a common set of facets.
  - Each coupling surface has own Euler domain.
  - Material flows from one Euler domain into the other through the open area represented by the common set of facets.

Flow only occurs if:

- The common facets are defined as 'open', using PORFCPL.
- The common facets open up due to failure of a shell structure, using COUP1INT.

Examples simulations are:

- Holes between air bag compartments.
- Holes between containers filled with gas or liquid.
- Open area between the top of a fuel-tank baffle and the fuel-tank skin.

- Open area in-between wide straps inside an air bag.
  - Failure of walls in between aircraft wing compartments.
  - Failure of tank armor by a blast wave.
  - Etc.
2. The following table summarizes what input cards support the simulation of material flow between two Euler domains across open areas in coupling surfaces:

Euler Solver	Material flow through a coupling surface	FLOW-METHOD=POLPACK	FLOW-METHOD=FACE	PORFLCPL (velocity based)	PORFLCPL (Pressure based)	COUP1INT/ COUP1FL (Failure of shell elements creates the opening)
HYDRO	YES	YES	NO	YES	YES	Only for flow method = polpack
HYDRO-Roe solver-1 <sup>st</sup> Order	YES	YES	YES	YES	YES	YES
HYDRO-Roe solver- 2 <sup>nd</sup> Order	YES	YES	YES	YES	YES	YES
MMHYDRO	YES	YES	NO	YES	NO	Only for flow method = polpack
STRENGTH	NO	-	-	-	-	-
MMSTREN	YES	YES	NO	YES	NO	Only for flow method = polpack

3. The Euler domains are shown below with the support types for each:

	FLOW-METHOD=POLACK	FLOW METHOD = FACET
<i>MESH</i> → <i>TYPE= ADAPT</i>	YES	No
<i>MESH</i> → <i>TYPE= BOX</i>	YES	YES
Modeling of CHEXA elements	No	YES

4. A Euler domain is associated with a coupling surface by specifying the MESHID or SET1ID on the COUPLE option.
5. FLOWMETH = POLPACK has the following limitations:

The entries NSTGP and NSTEL on all MESH entries should be left blank. It is not allowed to specify for any MESH entry the starting element number or starting grid point number.

- There are restrictions on output requests.

- Flow faces and wallets are not supported while FLOWDEF is supported.
- Viscosity is not supported

A case where these limitations require the use of FLOWMETH = FACET is when the Euler elements are generated in Patran, not using the MESH option, and one or more of the following options is used:

- FLOW boundaries are defined on some Euler faces.
- WALLET boundaries are defined on some Euler faces.
- Viscosity is defined.

**DYPARAM,GEOCHECK**

## Define Geometry Consistency Check - SOL700

This parameter forces a check of the geometry for consistent connectivity of the defined hexagonal elements and correction if needed. Used in SOL 700 only.

**Format:**

DYPARAM, GEOCHECK, value

**Example:**

DYPARAM, GEOCHECK, ON

Describer	Meaning
value	Flag for activating GEOCHECK. (Character; default=OFF)
ON	Geometry consistency check is performed.
OFF	No geometry consistency check is performed.

**Remarks:**

1. The defined geometry is checked for consistent connectivity of the hexagonal elements. If an inconsistency is detected, the connectivity is corrected. CFACE entries with references to elements that have been corrected are corrected as well.
2. If a hexagonal mesh is generated with other commercial preprocessors, this parameter can correct the connectivity of the hexagonal elements in case problems are encountered with face generation or volume computation.



## DYPARAM,FASTCOUP

## Fast Coupling Algorithm - SOL700

Defines the fast coupling algorithm. Used in SOL 700 only.

### Format:

DYPARAM,FASTCOUP,option1,option2,option3,option4,option5,option6,option7

### Example:

DYPARAM,FASTCOUP,INPLANE,FAIL

Describer	Meaning
Option1	NO Fast coupling interaction is turned off. (Character; Default = INPLANE)
	INPLANE Small offset for inplane coupling surface segments.
	NOFFSET No offset for inplane coupling surface segments
Option2	NOFAIL No failure of the coupling surface. (Character; Default = NOFAIL)
	FAIL Failure of the coupling surface will be taken into account.
Option3	Euler check algorithm. (Character; Default = NOCHKEUL)
	CHKEUL Checking whether all segments of the coupling surface are fully inside Euler elements.
	NOCHKEUL No checking
Option4	Euler update algorithm. (Character; Default = ALL)
	NEARONLY Do the full coupling surface only in the first cycle. In subsequent cycles only update elements and faces that are near the coupling surface.
	ALL Do the full coupling surface computation each cycle.
Option5	Controls coupling surface computations when there are multiple cubes defined with DYPARAM,EULERCB. (Character; Default = NONE)
	NONE All coupling surface computations are done for each cube.
	FACES The face cover fraction computation is done in one go for all cubes. The other coupling computations are still done per cube.
	POLPK The polpack computation efficiently handles multiple cubes. The other coupling computations are still done per cube.
	ALL All coupling surface computations efficiently handle cubes.

Describer	Meaning
Option6	<p>In the coupling surface computations it is determined which structural segments intersect the Euler mesh. This is done by looping across all structural segments. For dmp each cpu has only part of the Euler mesh. Therefore for many segments it is known beforehand that they will not intersect the Euler mesh of the cpu and can be skipped at an early stage of the computation. Option 6 activates this skipping. (Character; Default = ON)</p> <p>OFF            For dmp each CPU goes over all segments.</p> <p>ON             Each CPU skips all segments that do not intersect the Euler mesh of the CPU.</p>
Option7	<p>Each coupling surface segment has to be intersected with the Euler elements and Euler faces. For large Euler meshes a substantial amount of checking has to be done to find the intersecting elements and faces. By dividing the Euler mesh in search boxes the costs of this checking is kept to a minimum. Especially for large Euler meshes combined with large coupling surface this checking can become expensive and then the use of search boxes can significantly reduce the costs. For Euler meshes of less than 100000 elements the use of search boxes will have little effect. (Character; Default = ON)</p> <p>OFF            Search boxes are not used.</p> <p>ON             Search boxes are used.</p>

### Remarks:

1. The fast coupling algorithm is always turned on by default. In order to use general coupling, option1=NO should be used.
2. When option1 is set to INPLANE or when option 1 is blank, a small offset is given to coupling surface segments that are on top of a face of an Eulerian element. This is done because coupling surfaces segments on Eulerfaces make the Euler element volume computation invalid. Also boundary conditions on these segments are not correctly imposed. The net effect of these problems is unpredictable. The problem can either run correctly, or remain stable but give false results or become instable.
3. Option2 can only be used in combination with either PARAM,LIMITER,ROE or MMHYDRO or MMSTREN. The coupling surface must consist of CQUADs and/or CTRIAs and a failure model for the material of the surface must be defined.
4. This parameter can only be used when the Eulerian mesh is aligned with the basic coordinate system axes.

**DYPARAM, HYDROBOD**

## Hydro Body Force - SOL700

Defines a body force for single hydro material in Euler. Used in MSC Nasran SOL 700 only.

**Format:**

DYPARAM, HYDROBOD, XACC, YACC, ZACC

**Example:**

DYPARAM, HYDROBOD, -300., 0., 150.

Describer	Meaning
XACC	X-acceleration (Real; Default = 0.0)
YACC	Y-acceleration. (Real; Default = 0.0)
ZACC	Z-acceleration. (Real; Default = 0.0)

**Remark:**

1. This parameter defines a constant body force load in Euler for single hydro material only.

## **DYPARAM,HGCMEM**

## Shell Membrane Hourglass Damping Coefficient Parameters - SOL700

Defines the default membrane damping coefficient for shell elements. Used in SOL 700 only.

### **Format:**

DYPARAM, HGCMEM, value

### **Example:**

DYPARAM, HGCMEM,0.07

<b>Describer</b>	<b>Meaning</b>
value	Hourglass damping coefficient. ( $0.0 \leq \text{Real} \leq 0.15$ ; default=See remark <a href="#">3.</a> )

### **Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

## DYPARAM,HGCOEFF

## Hourglass Damping Coefficient - SOL700

Defines the global default hourglass damping coefficient. Used in SOL 700 only.

### Format:

DYPARAM, HGCOEFF, value

### Example:

DYPARAM, HGCOEFF, 0.14

Describer	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

### Remarks:

1. The default applies to all types of hourglass suppression methods and should be used unless there is good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficients are not explicitly defined on HGSUPPR entries or on a HGCMEM HGCWRP, HGCTWS, or HGCSOL entry.
3. If this entry is omitted, the default value of the hourglass damping coefficients is either equal to the default value of 0.1 or is equal to the value specified on a HGCMEM, HGCTWS, HGCWRP, or HGCSOL DYPARAM entry.
4. The value of the coefficients can be explicitly defined for each property by using an HGSUPPR entry.

**DYPARAM,HGCSOL**

## Solid Hourglass Damping Coefficient - SOL700

Define the default damping coefficient for solid elements. Used in SOL 700 only.

**Format:**

DYPARAM, HGCSOL, value

**Example:**

DYPARAM, HGCSOL, 0.11

Describer	Meaning
value	Hourglass damping coefficient. ( $0.0 \leq \text{Real} \leq 0.15$ ; default=See remark 3.)

**Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for solid elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

## DYPARAM,HGCTWS

## Shell Twisting Hourglass Damping Coefficient - SOL700

Defines the default twisting damping coefficient for shell elements. Used in SOL 700 only.

### Format:

DYPARAM, HGCTWS, value

### Example:

DYPARAM, HGCTWS, 0.02

Describer	Meaning
value	Hourglass damping coefficient. (0.0 ≤ Real ≤ 0.15; default=See remark 3.)

### Remarks:

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.

## **DYPARAM,HGCWRP**

## Shell Warping Hourglass Damping Coefficient - SOL700

Defines the default warping damping coefficient for shell elements. Used in SOL 700 only.

### **Format:**

DYPARAM, HGCWRP, value

### **Example:**

DYPARAM, HGCWRP, 0.0

<b>Describer</b>	<b>Meaning</b>
value	Hourglass damping coefficient. ( $0.0 \leq \text{Real} \leq 0.15$ ; default=See remark 3.)

### **Remarks:**

1. The default value applies to all types of hourglass suppression methods and should be used unless there is a good reason to define the hourglass damping coefficient by another means.
2. The value specified on this entry is used whenever the coefficient is not explicitly defined on the HGSUPPR entry.
3. If this entry is omitted, the default value of the coefficient used in the hourglass suppression method for shell elements is either equal to the default value of 0.1 or is equal to the default value defined on a DYPARAM,HGCOEFF entry.



## DYPARAM,HGSHELL

## Shell Hourglass Suppression Method - SOL700

Defines the default hourglass suppression method for shell elements. Used in SOL 700 only.

### Format:

DYPARAM, HGSHELL, value

### Example:

DYPARAM, HGSHELL, DYNA

Describer	Meaning
value	Type of HGSHELL. (Character; default=See remark 2.)
FBV	Flanagan-Belytschko viscous hourglass damping
DYNA	Viscous hourglass damping

### Remarks:

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for shell properties are left blank.
2. If this entry is omitted, the default suppression method used for shell elements is either FBV or the default method defined on the DYPARAM,HGTYPE entry.

## DYPARAM,HGSOLID

## Solid Hourglass Suppression Method - SOL700

Defines the default hourglass suppression method for solid elements. Used in SOL 700 only.

### Format:

DYPARAM, HGSOLID, value

### Example:

DYPARAM, HGSOLID, FBS

Describer	Meaning
value	Type of HGSOLID. (Character; default=See remark 2.)
FBS	Flanagan-Belytschko stiffness hourglass damping
DYNA	Viscous hourglass damping

### Remarks:

1. The type of hourglass suppression method defined on this entry is used as the default whenever the type fields on the HGSUPPR for solid properties are left blank.
2. If this entry is omitted, the default suppression method used for solid elements is either FBS or the default method defined on the DYPARAM,HGTYPE entry.

## DYPARAM,HGTYPE

## Hourglass Suppression Method - SOL700

Defines the default type of hourglass suppression method. Used in SOL 700 only.

### Format:

DYPARAM, HGTYPE, value

### Example:

DYPARAM, HGTYPE, FBS

Describer	Meaning
value	Type of HGTYPE. (Character; default=See remark 2.)
FBS	Flanagan-Belytschko stiffness hourglass damping.
FBV	Flanagan-Belytschko viscous hourglass damping.
DYNA	Viscous hourglass damping.

### Remarks:

1. The type of the hourglass suppression method defined on this entry is used as the default whenever the type fields in the HGSUPPR entries are left blank.
2. If this entry is omitted, the type can be defined on a DYPARAM,HGSHELL entry for shell elements, a DYPARAM,HGSOLID entry for solid elements, or on the HGSUPPR entries; otherwise the defaults apply. For shell elements the default is FBV; for solid elements, the default is FBS.

## **DYPARAM,HICGRAV**

Gravity Used by HIC Calculations - SOL700

Defines the value of the gravity to be used by the HIC calculations. Used in SOL 700 only.

### **Format:**

DYPARAM, HICGRAV, value

### **Example:**

DYPARAM, HICGRAV, 980.7

<b>Describer</b>	<b>Meaning</b>
value	Gravity used by HIC Calculations. (Real > 0.0; default=9.80665)

### **Remarks:**

1. The value set by this parameter will be used by all HIC output requests.
2. This parameter can only be set once in the input deck.

## DYPARAM,HVLFAIL

## Failure at Hydrodynamic Volume Limit - SOL700

Defines element failure on the hydrodynamic volume limit. Used in SOL 700 only.

### Format:

DYPARAM, HVLFAIL, value

### Example:

DYPARAM, HVLFAIL, YES

Descriptor	Meaning
value	Flag of HVLFAIL. (Character; default=NO)
YES	Element failure on hydrodynamic volume limit
NO	No element failure on hydrodynamic volume limit

### Remarks:

1. Lagrangian elements (CHEXA) that have a material model with a failure model fail when the hydrodynamic volume limit is reached and the parameter is set to YES.  
 The elements can fail only when the following items are presented in the input:
  - a. The material model has a hydrodynamic volume limit (HVL).
  - b. A failure model is defined.
  - c. DYPARAM, HVLFAIL, YES
2. The hydrodynamic volume limit by default allows for 10% expansion.

**DYPARAM,IMM**

## Initial Metric Method Formulation - SOL700

The option allows to specify the IMM method to be used. Used in SOL 700 only.

**Format:**

DYPARAM, IMM, OPTION1, OPTION2, VALUE1, VALUE2

**Example:**

DYPARAM, IMM, ZERO, YES, 1.0E-3, 1.0E-3

Describer	Meaning
OPTION1	Flag for element condition under compression. (Character; default=FULL)
	FULL While elements are under IMM condition, they will carry stresses when under compression.
	REDUCED While elements are under IMM condition they will carry a reduced stress when under compression. The relative area factor SMDFER is used to reduce the Young's modulus.
	ZERO While elements are under IMM condition they do not carry any compressive stresses. Use material damping to avoid excessive nodal velocities.
OPTION2	Flag for recalculation of IMM strain. (Character; default=See remark 2.)
	NO or OFF Do not recalculate IMM strains during the calculation.
	YES or ON Recalculate IMM strains during the calculation.
VALUE1	Start time of recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)
VALUE2	Times between recalculation of IMM strains. See remark 3.(Real > 0.0; Default is 1.0E-3)

**Remarks:**

- Method ZERO is best suitable when initially more than a couple of elements with zero or near zero area are present in the model.
- The default for Option2 depends on Option1.

Option1	Default for Option2
FULL	OFF
REDUCED	OFF
ZERO	ON

- When Option2 is OFF or NO, STREC and DTREC are neglected.

## DYPARAM,INFOBJ

## List the Generated BJOINS and Spotwelds - SOL700

Additional information about the BJOIN and spotweld connectivity will be listed in the output file. Used in SOL 700 only.

### Format:

DYPARAM, INFOBJ, value

### Example:

DYPARAM, INFOBJ, 0.14

Describer	Meaning
value	Flag of activation of writing BJOIN information. (Character; default=NO)
YES	Information is issued.
NO	Information is not issued.

### Remarks:

1. The information listed is:
  - Grid point pairs forming a BJOIN or a spotweld.
  - BJOINS and spotwelds initially connected.

## **DYPARAM,INISTEP**

Initial Time Step - SOL700

Defines the time step used at the start of the analysis. Used in SOL 700 only.

### **Format:**

DYPARAM, INISTEP, value

### **Example:**

DYPARAM, INISTEP, 1.0E-6

<b>Describer</b>	<b>Meaning</b>
value	Time step (in analysis time units) used for the first iteration. (Real > 0.0; required)

### **Remarks:**

1. This parameter is required to start an analysis.



## DYPARAM,LAGPR

## Lagrangian Partitioning Control - SOL700

Controls the partitions of the Lagrangian elements on the processors. Used in SOL 700 only.

### Format:

PARAM, LAGPR, IOPT, xvec, yvec, zvec, xcen, ycen, zcen

### Example:

PARAM, LAGPR, 6, 1.0, 0.0, 0.0, 0.0, -100.0, 0.0

Describer	Meaning
IOPT	Set partitioning type. See remark 1. (Integer > 0, default=1).
	1 Metis best partitioning
	2 Metis Decomposition on element based
	3 Metis Decomposition on node based
	4 Vector Decomposition with Specified User Direction. xvec, yvec and zvec are required.
	5 Radial Decomposition with User Direction. xvec, yvec, zvec, xcen, ycen and zcen are required.
	6 Angular Decomposition with User Direction. xvec, yvec, zvec, xcen, ycen and zcen are required.
	7 Recursive Coordinate Bisection Decomposition
xvec	X-component of directional vector. (Real, default=0.0)
yvec	Y-component of directional vector. (Real, default=0.0)
zvec	Z-component of directional vector. (Real, default=0.0)
xcen	X-coordinate of the origin of axis. (Real, default=0.0)
ycen	Y-coordinate of the origin of axis. (Real, default=0.0)
zcen	Z-coordinate of the origin of axis. (Real, default=0.0)

### Remark:

1. To use the option, check [DYPARAM,DMPOPT](#).

## **DYPARAM,LIMCUB**

## Contact Cube Sort Algorithm - SOL700

Defines the maximum number of cubes used to sort the grid points in a contact definition. Used in SOL 700 only.

### **Format:**

DYPARAM, LIMCUB, value

### **Example:**

DYPARAM, LIMCUB, 2300

<b>Describer</b>	<b>Meaning</b>
value	Maximum number of cubes. (Integer > 0; default=200)

### **Remarks:**

1. Each secondary node has to search for primary nodes that are close enough to have potential contact. It is too expensive to have each secondary node check each primary node. To limit the number of checks, the space in which the nodes reside is subdivided into cubes. This subdivision is done so that the secondary nodes have to check only the primary nodes in their own cube and those in the neighboring cubes. The maximum number of cubes used to subdivide the space is equal to the value of LIMCUB.

## DYPARAM,MATRMERG

## Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into a new FULLRIG assembly. Used in SOL 700 only.

### Format:

DYPARAM, MATRMERG, FR<id1>, MR<id2>, MR<id3>, FR<id4>, ...  
DYPARAM, MATRMERG, AUTO

### Example:

DYPARAM, MATRMERG, FR1, MR2, MR6, MR7, FR4, MR8

Describer	Meaning
FR<id1>	Name of the new FULLRIG assembly. (Character; required.)
MR<idi> or FR<idi>	Names of MATRIG and/or RBE2-FULLRIG rigid bodies merged into a new FULLRIG assembly with name FR<id1>. No names can be supplied for the AUTO option.
AUTO	Automatic merge. See remark 2. (Character; required.)

### Remarks:

1. FR<id1> must be a nonexisting RBE2-FULLRIG. The properties of FR<id1> (as mass, center of gravity, and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. Rigid body output can be asked for FR<id1>, and loads or rigid body constraints can be applied to FR<id1>. The other MATRIGs and RBE2-FULLRIGs mentioned on the MATRMERG entry disappear after they have been merged.
2. Instead of supplying rigid body names, the AUTO option can be used. After all the normal DYPARAM,MATRMERG and DYPARAM,MATRMERG1 entries have been applied, a DYPARAM,MATRMERG,AUTO merges all the resulting MATRIGs and RBE2-FULLRIGs which have common grid points into a new rigid assembly called FM<id>, where the id is a new FM number starting from 1. As it is not known at the start of an analysis how many FM assemblies will be created, no rigid body output can be asked for FM<id>, and no constraints or loads can be applied to FM<id>. The MATRIGs and RBE2-FULLRIGs, which have been merged by the AUTO option into a new FM<id> assembly, disappear.
3. To supply predefined properties for the merged assembly, DYPARAM,MATRMERG1 can be used, where the first rigid body mentioned on the entry must be an existing RBE2-FULLRIG or MATRIG.

**DYPARAM, MATRMRG1**

## Merges MATRIG and RBE2-FULLRIG Assemblies - SOL700

Merges MATRIG and/or RBE2-FULLRIG rigid bodies into one existing MATRIG or RBE2-FULLRIG assembly with predefined properties. Used in SOL 700 only.

**Format:**

DYPARAM, MATRMRG1, MR<id1>, MR<id2>, MR<id3>, FR<id4>, ...  
DYPARAM, MATRMRG1, FR<id1>, MR<id2>, MR<id3>, FR<id4>, ...

**Example:**

DYPARAM, MATRMRG1, MR1, MR2, MR6, MR7, FR4, MR8

Describer	Meaning
MR<id1> or FR<id1>	Name of the existing MATRIG or FULLRIG assembly (must be an existing one). (Character; required.)
MR<idi> or FR<idi>	Names of MATRIG and/or RBE2-FULLRIG rigid bodies, which are merged with the existing MR<id1> or FR<id1> into a new MATRIG assembly, with name MR<id1> or FR<id1>. (Character; required.)

**Remarks:**

1. MR<id1> or FR<id1> must be an existing MATRIG or RBE2-FULLRIG, respectively. For a FULLRIG, the properties of FR<id1> (as mass, center of gravity and moments of inertia) are computed by solver from the properties of each rigid body mentioned on the entry. For a MATRIG, the mass of MR<id1> is either the predefined mass on the MATRIG (id1) entry or the predefined density on the MATRIG (id1) entry times the total volume of all MATRIG members in the MATRMRG1 entry. The center of gravity and moments of inertia of MR<id1> are either predefined on the MATRIG (id1) entry, or are otherwise computed from the properties of each rigid body on the entry. The other MATRIGs and RBE2 FULLRIGs mentioned on the MATRMRG1 entry disappear after they have been merged.

## DYPARAM,MAXSTEP

## Maximum Time Step - SOL700

Defines the maximum allowable time step. Used in SOL 700 only.

### Format:

DYPARAM, MAXSTEP, value

### Example:

DYPARAM, MAXSTEP, 1 . E-3

Describer	Meaning
value	The maximum time step. (Real > 0.0; default=1.0E20)

### Remarks:

1. If the time step calculated by solver is greater than MAXSTEP, the time step is set to MAXSTEP.

## **DYPARAM,MESHPLN**

## Mesh Density for Covering Rigid Planes - SOL700

Defines Mesh density for covering rigid planes. Used in SOL 700 only.

### **Format:**

DYPARAM, MESHPLN, value

### **Example:**

DYPARAM, MESHPLN, 4

<b>Describer</b>	<b>Meaning</b>
value	Rigid planes will be meshed with MESHPLN times MESHPLN dummy quad elements. (Integer > 0; default=3)

### **Remarks:**

1. The default is sufficient in most cases.

**DYPARAM,MINSTEP****Minimum Time Step - SOL700**

Defines the minimum time step that causes the analysis to terminate. Used in SOL 700 only.

**Format:**

DYPARAM,MINSTEP,value

**Example:**

DYPARAM,MINSTEP,1.E-6

Describer	Meaning
value	When the time step is less than the MINSTEP value, the analysis terminates. (Real > 0.0; default=10% of DYPARAM,INISTEP)

**Remarks:**

1. When the elements become very distorted, in a poorly designed mesh for example, or when they have endured a very large distortion, the time step may drop dramatically. The analysis continues, however, and a lot of computer resources may be wasted. This option allows you to specify a minimum time step that causes the analysis to terminate.

## DYPARAM,MIXGAS

Controls Updating of Gas Fractions - SOL700

Specifies whether the gas constants of the Euler material or of gas bags are updated based on the gas composition and temperature. Used in SOL 700 only.

### Format:

DYPARAM, MIXGAS, value

### Example:

DYPARAM, MIXGAS, YES

Describer	Meaning
value	Flag of MIXGAS. (Character; default=NO)
YES	The gas constants for the Euler material and any gas bags are recalculated based on temperature and gas composition.
NO	Euler and gas bag gas constants are not recalculated.

### Remarks:

1. This parameter is only defined for use with GBAG gas bag definitions and/or the single-material Euler solver.
2. This parameter can be used in conjunction with INFLTR and INFLHB inflator definitions and with PORHOLE, PERMEAB, PORFGBG, and PERMGBG porosity definitions.
3. By default, DYPARAM,MIXGAS is set to YES if any INFLHB or INFLGAS entries are present.



## DYPARAM,NZEROVEL

## Auto Constrain Failed Nodes - SOL700

Set the velocity of a node to zero in case all attached elements have failed. Used in SOL 700 only.

### Format:

DYPARAM, NZEROVEL, value

### Example:

DYPARAM, NZEROVEL, YES

Describer	Meaning
value	Flag of NZEROVEL. (Character; default=NO)
	YES            Perform check and set the velocity to zero if all attached elements have failed.
	NO             Do not perform check.

### Remarks:

1. This parameter applies only to nodes of Lagrangian elements.
2. Specifying NO reduces the CPU overhead time.
3. When the velocity of a node is set to zero, effectively the node is constraint, like an SPC or SPC1.
4. Special attention is necessary for the contact definition. If the failed node is not taken out of the contact, it behaves as a rigid boundary constraint.

## **DYPARAM,OLDLAGT**

Use Collapsed Hexahedron Scheme for CTETRA - SOL700

Activate the collapsed hexahedron scheme as default for lagrangian CTETRA elements. Used in SOL 700 only.

### **Format:**

DYPARAM, OLDLAGT, value

### **Example:**

DYPARAM, OLDLAGT, 1

<b>Describer</b>	<b>Meaning</b>
value	Flag of OLDLAGT. (Integer $\geq 0$ ; default=See remark 1.)
	0     Deactivate collapsed hexahedron scheme
	1     Activate

### **Remarks:**

1. The current default integration scheme for Lagrangian CTETRA elements use linear tetrahedron FE one. It is more consistent (in terms of accuracy) and efficient (both in memory and CPU time) compared with the collapsed hexahedron scheme. The old scheme based on collapsed hexahedron with reduced integration is deactivated. If the old scheme is activated, it is used as default. But, it is still possible to use the new scheme for CTETRA by using separate PSOLID with IN = 1 and ISOP = 1 combination.

## DYPARAM,JWLDET

## Hydro Body Force - SOL700

Specifies whether the blast wave of one explosive can ignite another explosive. Here it assumed that the explosives are modeled by a combination of EOSJWL and DETSPH entries.

### Format:

PARAMJWLDET,OPTION

### Example:

PARAM,JWLDET,NOLINK

Describer	Meaning
LINK	Multiple denotations with EOSJWL are LINKED. The detonation wave of one explosive can ignite another explosive.
NOLINK	Multiple denotations with EOSJWL are NOT LINKED. The detonation wave of one explosive cannot ignite another explosive.
Default	NOLINK

### Remark:

1. Option NOLINK: TDET is set to -1 for elements that have no JWL material. The "NOLINK" option is only valid with true JWL materials - not valid for the "Static Detonation/Ideal Gas" model.
2. Setting this parameter as "NOLINK" will prevent "sympathetic ignition". Each charge will ignite at the specified "TDET" in its own DETSPH card.

**DYPARAM,LIMITER**

## Original Roe-Type Euler Solver Scheme - SOL700

Defines the type and the spatial accuracy of scheme used in the Euler solver based on the ideas of Prof. Philip Roe. Used in SOL 700 only.

**Format:**

DYPARAM,LIMITER,TYPE,OPTION

**Example:**

DYPARAM,LIMITER,ROE

Describer	Meaning
TYPE	Type of scheme. (Character; Default = ROE)
	ROE           Roe solver for single hydrodynamic materials.
OPTION	blank        Second order in space. (Character; Default = blank)
	NONE        First order in space.

**Remarks:**

1. By default, when the parameter is not set, the solver is used that is defined on the PEULER or the PEULER1 entry. In case “2ndOrder” or “1stOrder” was defined on the PEULER or PEULER1 entry, the parameter setting has no effect.
2. By default, second order spatial accuracy is used. The temporal accuracy is automatically defined according to the spatial accuracy that you select.
3. Note that 2<sup>nd</sup> order spatial accuracy in the Roe solver involves the internal flow field only. We recommend that you use the full 2<sup>nd</sup> order improved fluid-and gas Euler solver. You can activate the improved solver by putting the “2ndOrder” field on the PEULER or PEULER1 entry.
4. When type ROE is defined, no void elements are allowed and it cannot be used in combination with EOSJWL. Also options concerning air-bag analyses are not supported.
5. For more details on fluid- and gas Euler solvers, refer to the Getting Started and the Theory Manuals.

## DYPARAM,PARALLEL

## Parallel Execution Information - SOL700

The option allows you to gather information on the parallel section. Used in SOL 700 only.

### Format:

DYPARAM, PARALLEL, INFPAR, value

### Example:

DYPARAM, PARALLEL, INFPAR, ON

Describer	Meaning
INFPAR	A report is written on the actual amount of work done at the reported parallel levels. (Character; required)
Value	Flag for INFPAR. (Character; default=OFF)
	ON            Activate writing information.
	OFF           Deactivate writing information.

### Remarks:

1. A summary on the parallel operation when using the shared-memory mode can be requested by including a DYPARAM,PARALLEL,INFPAR,ON entry in the input file.
2. Currently, the information on the parallel sections is available for the shell solver only.

**DYPARAM,PLCOVCUT**

Pressure Cut Off Time - SOL700

Defines time when PLCOVER is cut off. Used in SOL 700 only.

**Format:**

DYPARAM, PLCOVCUT, value

**Example:**

DYPARAM, PLCOVCUT, 3.E-3

Describer	Meaning
value	<p>If there are one or more COUPLE definitions with a PLCOVER specified on the COUOPT entry, a cut off is applied to the PLCOVER until time = PLCOVCUT. From time = 0 to time = PLCOVCUT, the PLCOVER is cut off to the pressure in the intersected Eulerian element.</p> <p>For times greater than PLCOVCUT, the full PLCOVER is applied to the coupling surface.</p> <p>This parameter is useful in air-bag analyses, where PLCOVER is used to model the environment pressure. During the early stages of the deployment of the air bag, the pressure inside the bag may drop. Applying the full PLCOVER may lead to an unstable deployment of the air bag. (Real <math>\geq 0.0</math>; default=0).</p>

**Remarks:**

1. See also the COUPLE and COUOPT Bulk Data entries.

## DYPARAM,PMINFAIL

## Switches Failure at Spall Limit - SOL700

Defines Lagrangian solid element failure on reaching the spall limit. Used in SOL 700 only.

### Format:

DYPARAM, PMINFAIL, value

### Example:

DYPARAM, PMINFAIL, YES

Describer	Meaning
VALUE	Flag of PMINFAIL. (Character; default=NO)
YES	Element failure on spall limit
NO	No element failure on spall limit

### Remarks:

1. Lagrangian elements (CHEXA) that have a material definition with a failure model will fail when the parameter is set to YES and the spall limit (minimum pressure) is reached, even when the other failure criterion is not yet reached.
2. The spall limit is set on the PMINC entry.

## **DYPARAM,RBE2INFO**

Lists MATRIG and RBE2 Grid Points - SOL700

The grid points attached to MATRIG and RBE2assemblies are listed to the output file. Used in SOL 700 only.

### **Format:**

DYPARAM, RBE2INFO, value

### **Example:**

DYPARAM, RBE2INFO, GRIDON

<b>Describer</b>	<b>Meaning</b>
VALUE	Flag of PBE2INFO. (Character; default=GRIDOFF)
GRIDON	Information is issued C
GRIDOFF	No information is issued



## DYPARAM,RHOCUT

## Global Density Cutoff Value - SOL700

Defines the minimum density for all Eulerian elements. Used in SOL 700 only.

### Format:

DYPARAM, RHOCUT, value

### Example:

DYPARAM, RHOCUT, 1.E-10

Describer	Meaning
value	Density cutoff. (Real > 0.0; default=See remark 4.)

### Remarks:

1. Any Eulerian element with a density less than RHOCUT is considered to be empty. All of its variables are set to zero, and the equation of state is bypassed.
2. In the Eulerian transport calculation, if the material is flowing from element A to element B, and
  - a. If the density of element B after transport is less than RHOCUT, then no transport is done.
  - b. If the density of element A after transport is less than RHOCUT, then all of the mass is transported to element B.
3. A reasonable value of RHOCUT is 1.E-5 times the initial density.
4. If only RHOCUT is defined, all Eulerian elements use the RHOCUT value as cutoff density. If RHOCUT is omitted, all Eulerian elements use a cutoff density automatically set to 1.E-5 times a characteristic density. For single-material Eulerian elements, this characteristic density is the reference density.

**DYPARAM,RJSTIFF**

## Rigid-joint Stiffness - SOL700

Defines the stiffness of a rigid joint. Used in SOL 700 only.

**Format:**

DYPARAM, RJSTIFF, value

**Example:**

DYPARAM, RJSTIFF, 100 .

Describer	Meaning
value	Multiplication factor for the stiffness of all rigid joints. (Real > 0.0; default=1.0)

**Remarks:**

1. The absolute stiffness of rigid joints is calculated automatically by solver. The stiffness of joints is taken so that a stable solution is guaranteed. The stiffness calculation takes into account the fact that a rigid body can be constrained by more than one joint.
2. This parameter can be used to increase or decrease the stiffness of the joints. Care must be taken because too high a value may lead to an unstable calculation.

## DYPARAM,SCALEMAS

## Mass Scaling Definition - SOL 700

Defines the activation of mass scaling.

### Format:

DYPARAM, SCALEMAS, DTMIN, MXPERC, STEPS.

### Example:

DYPARAM, SCALEMAS, 1.0E-6, 100.0, 1

Descriptor	Meaning
DTMIN	Minimum allowable time step. (Real > 0.0; required)
MXPERC	Maximum percentage of added numerical mass with respect to original mass. (Real > 0.0; required)
STEPS	Number of steps. (Integer > 0; required)

### Remarks:

1. Numerical mass is added to all Lagrangian solid, triangular, quadrilateral, rod, bar and beam elements such that its time step never becomes less than:

$$dt = STEPFACT * DTMIN.$$

Where,

dt = time step of calculation

STEPFACT = time step safety factor (see PARAM, STEPFACT)

DTMIN = value specified on the PARAM, SCALEMAS entry

If the added mass of a certain element exceeds the maximum percentage (MXPERC) of its original mass, no more mass will be added, and subsequently, the time step may decrease again.

2. The value of STEPS determines the checking frequency against the mass scaling criterion; the check is done for every defined number of STEPS. STEPS = 1 is recommended.
3. By requesting MSMASS in an ELOUT output request, the ratio of scaled mass to original mass of the elements can be retrieved. By making fringe plots of this parameter, a check can be made if mass has not been added in a critical area.
4. See *MSC Nastran Explicit Nonlinear (SOL 700) User's Guide*, Chapter 4: Special Modeling Techniques, [Mass Scaling](#) Definition for instructions on how to use this entry

**DYPARAM,SHELLFRM**

Sets the Default of the Shell Formulation - SOL700

Sets the default for the shell formulation for quadrilateral elements. Used in SOL 700 only.

**Format:**

DYPARAM, SHELLFORM, value

**Example:**

DYPARAM, SHELLFORM, BLT

Describer	Meaning
value	Flag for default of shell formulation. (Character; default=KEYHOFF)
BLT	The shell-formulation default is BLT.
KEYHOFF	The shell-formulation default if KEYHOFF.

**Remarks:**

1. The DYPARAM,SHELLFORM changes the default formulation for quadrilateral shell elements. All shell properties entries that do not explicitly define the formulation, use the default as specified on the DYPARAM entry.
2. Triangular shell elements have only one formulation (C0-TRIA). Therefore, the DYPARAM is ignored for triangular elements.

## DYPARAM,SHELMSYS

## Shell Element System Definition - SOL700

Defines the shell element system for the BLT shells. Used in SOL 700 only.

### Format:

DYPARAM, SHELMSYS, value

### Example:

DYPARAM, SHELMSYS, SIDE21

Describer	Meaning
value	Type of shell element system. (Character; default=MIDSIDES)
	SIDE21      x-axis along side21.
	MIDSIDES    x-axis connecting midpoints.

### Remarks:

1. SIDE21 puts the x-axis along side21 of the element, whereas MIDSIDES puts the x-axis along the vector connecting the midpoints of the side14 and side32.
2. Using the SIDE21 option for the BLT shell will result in the same Belytschko-Lin-Tsay implementation as BELY.

**DYPARAM,SHPLAST**

## Type of Plane-Stress Plasticity for Shells - SOL700

Specifies the type of calculation used to determine the plane-stress plasticity method for shells. Used in SOL 700 only.

**Format:**

DYPARAM, SHPLAST, value

**Example:**

DYPARAM, SHPLAST, VECT

Describer	Meaning
value	Type of Plain-stress plasticity for shells. (Character; default=ITER)
RADIAL	Noniterative, approximate radial return.
VECT	Iterative, vectorized with three iterations.
ITER	Nonvectorized iterations.

**Remarks:**

1. The RADIAL approach does not require iterations and, therefore, is the most efficient. It is, however, an approximation.
2. The other two approaches iterate to find the solution. ITER is the best since it takes as many iterations as are necessary. On vector machines, such as CRAY, this is inefficient since it cannot be vectorized. VECT always performs three vectorized iterations, which is more efficient. However, three iterations may not be enough, and inaccuracies could occur.

## DYPARAM,SHSTRDEF

## Composite Shell Stress and Strain Output Definition - SOL700

Specifies the default coordinate system for the stress and strain output of composite shells. Used in SOL 700 only.

### Format:

DYPARAM, SHSTRDEF, value

### Example:

DYPARAM, SHSTRDEF, ELEM

Describer	Meaning
value	Type of output. (Character; default=FIBER)
FIBER	Stresses and strains are output in the fiber and matrix directions.
ELEM	Stresses and strains are output in the element coordinate system.

### Remarks:

1. The default setting can be overruled per property on a PCOMPA entry on the STRDEF field.

**DYPARAM,SHTHICK**

## Shell-Thickness Modification Option - SOL700

Specifies whether or not the thickness of the shell changes with membrane straining. Used in SOL 700 only.

**Format:**

DYPARAM, SHTHICK, value

**Example:**

DYPARAM, SHTHICK, YES

Describer	Meaning
value	Flag of shell thickness modification. (Character; default=YES)
	YES Shell thickness is modified according to the membrane strain.
	NO Shell thickness is constant.

**Remarks:**

1. The YES option gives a true large-strain shell but requires some extra computation.
2. The NO option should give adequate results as long as the membrane strains are not very large (i.e., not more than 5–10%).
3. This option applies to all the formulations of the shell elements, except for the PCOMP. The thickness of PCOMP shell elements will always remain constant.



## DYPARAM,SLELM

## Store Shell Sublayer Variables - SOL700

Defines whether shell sublayer variables are to be stored in the element arrays. Used in SOL 700 only.

### Format:

DYPARAM, SLELM, value

### Example:

DYPARAM, SLELM, NO

Describer	Meaning
value	Flag of storing shell sublayer variables. (Character; default=YES)
	YES      Store as an element variable.
	NO      Do not store as an element variable.

### Remarks:

1. This parameter applies only to shell elements.
2. The shell sublayer variables are primarily stored in sublayer arrays. They can be copied into the element arrays only for specific output purposes.
3. Specifying NO reduces the CPU overhead time.
4. Irrespective of the entry on this parameter, sublayer variables are accessible in the sublayer arrays. For example, requesting TXX1 retrieves the stress from the element array, whereas TXX01 retrieves it from the sublayer arrays.

## DYPARAM,SMP,BATCHSIZ

## SMP Batchsize and CPULOOOP Definition - SOL700

Define batch size and number of CPU loops, per entity type. Used in SOL 700 only.

### Format:

DYPARAM, SMP, <entity>, BATCHSIZ, <value1>, CPULOOPS, <value2>

### Example:

DYPARAM, SMP, BEAMS, BATCHSIZ, AUTOMAT  
 DYPARAM, SMP, BEAMS, BATCHSIZ, 256

Describer	Meaning
ENTITY	Define the batchsize for the following entities. (Character; required) BEAMS      Beam elements SHELLS     Shell elements SOLIDS      Solid elements CONTACT    Contacts
VALUE1	The batchsize for this entity (Integer number) or AUTOMAT (character). See Remarks 1.and 3. (Character or Integer >0; default = 63)
VALUE2	If value1 is defined as AUTOMATIC, the user may define the number of loops per CPU See Remark 2.(Integer > 0; default=2)

### Remarks:

- The batchsize may be defined per entity type. The SMP parts uses the batchsize to assign a certain number of batches to each CPU. Example:
  - Batchsize = 63 (default)
  - Number of Beam elements = 580
  - Number of CPUs = 4

As a results,

- Needs to process 10 batches (9x63 + 1x13)
- To spread this evenly over four CPUs, a solver increases this to 12 batches because then each CPU can process 3 batches.
- As a result, the batch size is recalculated for 12 batches to: batchsize = 49
- This means that:

CPU# 1,2,3 process three batches of 49 elements each (3x3x49 = 441 elements)

CPU# 4 processes two batches of 49 elements and one batch of 41

It is clear that the user-defined batchsize may not be used ultimately. The reason for this is that each CPU is allocated the same number of batches in order to optimize the work done by each processor.

2. When the batchsize is set to AUTOMAT, a solver computes an initial batchsize based on the number of batches processed by each CPU. By default, processes two batches per CPU.

In the example above, this means that based on:

- Batchsize = AUTOMAT
- Number of batches per CPU = 2 (default)
- Number of Beam elements = 580
- Number of CPUs = 4

As a result,

- Computes an initial batchsize of 72 ( $=580/(4*2)$ )
- Increases this batchsize to a higher number to make sure the number of batches fits in the fixed number of CPU's:  $\text{batchsize} = 72 + (580 - 4*2*72) = 72 + 4 = 76$  elements
- Based on this initial batchsize, the number of batches would be:  $1 + (580-1)/76 = 8$  batches
- This fits nicely on four CPU's as expected
- The CPUs process:

CPU# 1,2,3 processes two batches of 76 elements each ( $3*2*76 = 456$  elements)

CPU# 4 processes one batch of 76 elements and 1 batch of 48 elements.

3. By default, a solver does not use the AUTOMAT batchsize algorithm. Testing has shown that an initial batchsize of 63 yields the fastest results. This may vary depending on the problem simulated and the number of elements and may be worth adjusting.

The reason for this may be found in the fact, that with many elements in the model, the number of elements per batch increases significantly when the AUTOMAT batchsize algorithm is used. Each CPU will have to process two batches (by default) with many elements. Based on the cache memory, this is usually not optimal for a CPU processor. Shorter batchsizes ( $< 128$ ) will do a better job.

**DYPARAM,SMP,CPUINFO**

## SMP CPU Information - SOL700

Define the CPU information per entity type. Used in SOL 700 only.

**Format:**

DYPARAM, SMP, <entity>, CPUINFO, <value1>

**Example:**

DYPARAM, SMP, CONTACT, CPUINFO, ON

Describer	Meaning
ENTITY	Define the CPUINFO for the following entities. (Character; required)
BEAMS	Beam elements
SHELLS	Shell elements
SOLIDS	Solid elements
CONTACT	Contacts
MEMBR	Membrane elements
EULT9	Eulerian Roe Solver
FSCOUP	Fast coupling algorithm
ALL	Output for all of the above
VALUE1	Flag for writing CPU information. (Character; required)
ON	Ask for parallel CPU time information C
OFF	No CPU time information (default)

**Remarks:**

1. By default, a solver does not give any SMP CPU timings per entity. In order to judge the efficiency of the different algorithms, this parameter may be used to judge the speedup acquired by processing on multiple CPUs.

## DYPARAM,SNDLIM

## Sound Speed Minimum Value - SOL700

Defines the minimum value for the speed of sound. Used in SOL 700 only.

### Format:

DYPARAM, SNDLIM, value

### Example:

DYPARAM, SNDLIM, 1 . E-6

Describer	Meaning
value	Minimum value of speed of sound. (Real > 0.0; default=1.0E-3)

### Remarks:

1. This parameter is used to avoid the possibility of division by zero in the time step calculation.
2. SNDLIM has the units of velocity.

**DYPARAM,SPREMAP**

## 1-D Spherical Symmetric Euler Archive Remap - SOL700

Allows a 1-D spherical symmetric Euler archive importation to a 3-D simulation. Used in SOL 700 only.

**Format:**

DYPARAM,SPREMAP,X0,Y0,Z0,range

**Example:**

DYPARAM,SPREMAP,0.0,0.5,0.5

Descriptor	Meaning
X0,Y0,Z0	X, Y, Z coordinate of the point at which the 1-D spherical symmetric mesh is remapped. (Real; default=0.0)
RANGE	Only material whose distance to (X0,Y0,Z0) is smaller than "range" will be initialized with the 1-D spherical symmetric Euler archive. (Real > 0.0; default = 1.0E20)

**Remarks:**

1. Since 1-D spherical symmetric simulations run much faster than 3-D simulation, it can save much CPU time to do the first part of the simulation with a 1-D axial symmetric mesh. Afterwards, the 1-D spherical symmetric Euler archive is imported into the 3-D simulation. By default, the 1-D spherical symmetric archive will not be expanded in 3-D. To enable this expansion, DYPARAM,SPREMAP has to be used. It is useful for blast wave simulations. The 1-D spherical symmetric simulation has to be terminated before the blast wave approaches any 3-D structure.
2. This import of Euler archives is done by means of the EID option in the pth file.
3. To generate axial symmetric meshes DYPARAM,SPHERSYM can be used.
4. The remap of 1-D spherical symmetric to axial symmetric Euler meshes is supported.

## DYPARAM,SPHERSYM

## Spherical Symmetric Analyses - SOL700

Enables an efficient and accurate 1D spherical symmetric solution for Eulerian materials. A much larger time step becomes possible by basing the time step only on the mesh-size in radial direction. Used in SOL 700 only.

### Format:

DYPARAM, SPHERSYM, MESHTYPE, RADAXIS, PHI

### Example:

PARAM, SPHERSYM, RECT, X, 2.0

Descriptor	Meaning
MESHTYPE	Two types of Euler meshes are supported. (Character; required) SPHERIC     Axial symmetric meshes RECT         Rectangular meshes
RADAXIS	Radial Axis. (Character; required.) X             Global x direction Y             Global y direction Z             Global z direction
PHI	Only used for MESHTYPE = RECT. Used to creates a 1d Spherical mesh with angles +PHI/2 and -PHI/2. (Real ≥ 0; default=0.0)

### Remarks:

1. Only available for Eulerian elements and does not support Lagrange elements. The effect of this parameter is not limited to the solvers. Also Euler archives will reflect the modified Euler mesh geometry.
2. The Euler mesh can already be symmetric but also a rectangular mesh comprising of one row of elements can be used. Using the angle specified by PHI this Euler mesh is mapped into a 1D spherical symmetric mesh.
3. The Euler mesh has to consist of one row of elements.
4. In the time step computation only the mesh-size in radial direction will be taken into account.

## **DYPARAM,STRNOUT**

## Shell Sublayer Strain Output - SOL700

Saves the total strains and equivalent effective stress (von Mises stress) at shell sublayers for output. Used in SOL 700 only.

### **Format:**

DYPARAM, STRNOUT, value

### **Example:**

DYPARAM, STRNOUT, YES

<b>Describer</b>	<b>Meaning</b>
value	Flag for writing strain results. (Character; default=YES)
	YES      Save
	NO      Do not save.

### **Remarks:**

1. A limited output set saves memory.
2. Perfectly elastic materials only have the limited output set.
3. Total strain output for shell composite materials can be requested from the PCOMPA Bulk Data entry.



## DYPARAM,TOLCHK

## Tolerance for Fast Coupling - SOL700

Fast coupling requires that Euler elements are orthogonal. This means that normals of Euler element faces have to be in a coordinate direction. In practice there can be small errors in the geometry of the Euler element and Euler face normals do not always exactly point into a coordinate direction. Small deviations from the coordinate direction do not give problems and are allowed by fast coupling.

To check the direction of an Euler face, the face normal vector is projected onto the closest coordinate direction. If this projection is 1, the normal is exactly in the coordinate direction. When this projection is within a sufficient small tolerance of 1, the face can be handled by fast coupling. The tolerance used is TOLCHK. If the projection is smaller than 1-TOLCHK, the face cannot be handled by fast coupling and the analysis terminates. Options are then to slightly increase TOLCHK, write out double precision format in PATRAN, use general coupling, or use the MESH entry. Increasing TOLCHK too much can make the coupling surface computation less accurate. To keep the computation accurate, the maximal allowed value of TOLCHK is 1e-6. If DYPARAM,TOLCHK is not used, the tolerance used is 1e-14. Used in SOL 700 only.

### Format:

DYPARAM, TOLCHK, value

### Example:

DYPARAM, TOLCHK, 1e-10

Describer	Meaning
value	Tolerance in accepting faces by fast coupling. (Real > 0.0; required.)

## DYPARAM,VDAMP

## Dynamic Relaxation Parameter - SOL700

Controls the global damping in the dynamic relaxation. Used in SOL 700 only.

### Format:

DYPARAM, VDAMP, value

### Example:

DYPARAM, VDAMP, 0.001

Describer	Meaning
value	Dynamic relaxation parameter. (Real $\geq 0.0$ ; default=0.0)
0.0	No dynamic relaxation.

### Remarks:

- The dynamic relaxation parameter is connected to the system natural frequency,  $\omega$ , as  $\omega = s\omega_0 t$ , where  $s$  denotes a percentage of critical damping. The damping occurs by factoring the velocities every time step as follows:

$$F_1 = (1 - \gamma) / (1 + \gamma)$$

$$F_2 = 1 / (1 + \gamma)$$

$$v^{n+1/2} = F_1 v^{n-1/2} + F_2 a^n \Delta t^n$$

where  $v$  is the velocity,  $a$  is the acceleration, and  $\gamma$  is the dynamic relaxation parameter.

## DYPARAM,VELMAX

## Maximum Velocity - SOL700

Defines the maximum velocity in Eulerian and Lagrangian meshes. Used in SOL 700 only.

### Format:

DYPARAM, VELMAX, value, option

### Example:

DYPARAM, VELMAX, 1 . E 6

Describer	Meaning				
VALUE	Maximum velocity. (Real > 0.0; default=1.0E10)				
OPTION	Flag for removing Euler elements. (Character; default=YES)				
	<table border="0"> <tr> <td>YES</td> <td>Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.</td> </tr> <tr> <td>NO</td> <td>Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.</td> </tr> </table>	YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.	NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.
YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.				
NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.				

### Remarks:

1. For Eulerian elements, VELMAX is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAX. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAX.
2. Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
3. Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
4. VELMAX must be greater than the minimum velocity specified by PARAM,VELCUT.

**DYPARAM,VELMAX1**

## Maximum Translational and Angular Velocity - SOL700

Defines the maximum translational and angular velocity in Eulerian and Lagrangian meshes. Used in SOL 700 only.

**Format:**

DYPARAM, VELMAX1, VELMAXT, option, VELMAXA

**Example:**

DYPARAM, VELMAX1, 1.E6

Describer	Meaning				
VELMAXT	Maximum translational velocity. (Real > 0.0; default=1.0E20)				
OPTION	Flag for removing Euler elements. (Character; default=YES)				
	<table border="0"> <tr> <td>YES</td> <td>Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.</td> </tr> <tr> <td>NO</td> <td>Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.</td> </tr> </table>	YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.	NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.
YES	Remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.				
NO	Do not remove the mass in Eulerian elements in which the velocity exceeds the maximum specified velocity.				
VELMAXA	Maximum angular velocity. (Real > 0.0; default=1.0E20)				

**Remarks:**

1. For Eulerian elements, VELMAXT is applied to the velocity components separately. Therefore, the magnitude of the velocity vector is limited to VELMAXT. For Lagrangian elements, all components are scaled down by the same factor to ensure that the maximal magnitude of the velocity vector does not exceed VELMAXT.
2. VELMAXA is used to limit the angular velocity of grid points. By default, the angular velocities are not limited. In general, there is no need to limit angular grid point velocities.
3. Although it is not usually necessary to limit the velocity in Eulerian meshes, there are occasions in regions of near-vacuous flow for example, where specifying a maximum velocity can be advantageous. The same applies to Lagrangian meshes in contact regions for example. This parameter should be used with care.
4. Because very high velocities occur mostly in Eulerian elements with very small mass, the mass in these elements can be removed to keep the analysis stable. This option is not available for Lagrangian solid elements.
5. VELMAXT must be greater than the minimum velocity specified by PARAM, VELCUT.
6. DYPARAM, VELMAX, and DYPARAM, VELMAX1 can be used in the same input deck. The values set by DYPARAM, VELMAX1 will overrule the values set by DYPARAM, VELMAX.

For example, when using

DYPARAM, VELMAX, 1000, YES

```
DYPARAM,VELMAX1,,YES,1e+10
```

In the OUT file shows:

```
%I-INIT 02-p3_set_velmaxat, , , (13),  
SUMMARY VELMAX SETTINGS:  
MAXIMAL VALUE TRANSLATIONAL VELOCITY = 0.100000E+04  
MAXIMAL VALUE ANGULAR VELOCITY = 0.100000E+11  
REMOVE EULERIAN MASS WHEN VELOCUIITY EXCEEDS LIMIT: YES
```

## DYPARAM,VISCPPLAS

## Use Overstress Formula to Update Strain-rate Dependent Plasticity - SOL700

Activate the overstress formula to update strain-rate dependent plasticity. This formula is normally used for viscous-plastic material. Used in SOL 700 only.

### Format:

DYPARAM,VISCOPLAS,value

### Example:

DYPARAM,VISCOPLAS,1

Describer	Meaning
VALUE	Flag of updating overstress formula. (Integer $\geq 0$ ; default=0)
	0 Use scaling-up scheme
	1 Use overstress formula

### Remarks:

1. The strain rate dependent plasticity is normally calculated by scaling up the basic yield stress without strain rate effect. Then the trial stresses are mapped back to the scaled-up yield surface. This algorithm may lead to premature instability. Another technique is to calculate the so-called viscous-plastic strain rate using “overstress” formula. And then the stresses are updated based on this viscous-plastic strain. This technique seems to be more stable than the previous one. For shell elements, this option works when combined with DYPARAM,SHPLAST,RADIAL. Only MATEP and Johnson-Cook models are supported. For solid elements, this option works only for MATEP.
2. For shell elements, DYPARAM,VISCOPLAS,1, in combination with DYPARAM,SHPLAST,RADIAL, will use consistent plane stress plasticity algorithm both for strain rate dependent and independent plasticity. This new algorithm is more accurate than 3-D approach.

## ECHOOFF Deactivate Printed Echo

Marks the point or points in the input file to deactivate printed echo of the input file.

### Format:

1	2	3	4	5	6	7	8	9	10
ECHOOFF									

### Example:

ECHOOFF									
---------	--	--	--	--	--	--	--	--	--

### Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\)](#).
2. The companion to this entry is the ECHOON entry.
3. The ECHOOFF command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended:  
ECHO=UNSORT.

## ECHOON

### Activate Printed Echo

Marks the point or points in the input file to activate printed echo of the input file.

#### Format:

1	2	3	4	5	6	7	8	9	10
ECHOON									

#### Example:

ECHOON									
--------	--	--	--	--	--	--	--	--	--

#### Remarks:

1. This entry may also be used in the Executive Control and Case Control Sections. It is described in the Case Control command, [ECHO \(Case\)](#).
2. The companion to this entry is the ECHOOFF entry.
3. The ECHOON command is BEGIN BULK or BEGIN SUPER = n dependent. Its last time used does carry over between changes in BEGIN BULK or BEGIN SUPER = n commands. Also, the Case Control ECHO command should be above subcase level and the following is recommended:  
ECHO=UNSORT.



## EIGB Buckling Analysis Set

Defines data needed to perform buckling analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
EIGB	SID	METHOD	L1	L2	NEP	NDP	NDN		
	NORM	G	C						

### Example:

EIGB	13	INV	0.1	2.5	2	1	1		
	MAX								

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character: "INV" for inverse power method or "SINV" for enhanced inverse power method.)
L1, L2	Eigenvalue range of interest. (Real, L1 < L2)
NEP	Estimate of number of roots in positive range not used for METHOD = "SINV". (Integer > 0)
NDP, NDN	Desired number of positive and negative roots. (Integer>0; Default = 3*NEP)
NORM	Method for normalizing eigenvectors. (Character: "MAX" or "POINT"; Default = "MAX")
	MAX            Normalize eigenvectors to the unit value of the largest component in the analysis set. (Default).
	POINT        Normalize eigenvectors to the unit value of the component defined in G and C fields. The value for NORM defaults to MAX if the defined component is zero.
G	Grid or scalar point identification number. Required only if NORM="POINT". (Integer > 0)
C	Component number. Required only if NORM="POINT" and G is a geometric grid point. (1 ≤ Integer ≤ 6)

### Remarks:

1. The EIGB entry must be selected with the Case Control command METHOD = SID.
2. Each eigenvalue is the factor by which the pre-buckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.

3. The continuation entry is optional. If the continuation is not specified, than NORM = “MAX” normalization is performed.
4. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
5. The SINV method is an enhanced version of the INV method. It uses Sturm sequence techniques to ensure that all roots in the range have been found. It is generally more reliable and more efficient than the INV method.
6. Convergence is achieved at  $10^{-6}$ .
7. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the L1 and L2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If NDP and NDN are set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.

## EIGC Complex Eigenvalue Extraction Data

Defines data needed to perform complex eigenvalue analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
EIGC	SID	METHOD	NORM	G	C	E	ND0		

The following continuation is repeated for each desired search region. (J = 1 to n, where n is the number of search regions.

	ALPHAAJ	OMEGA AJ	ALPHABJ	OMEGABJ	LJ	NEJ	NDJ		
--	---------	----------	---------	---------	----	-----	-----	--	--

### Alternate Format for Continuation Entry for Block Complex Lanczos:

	ALPHAAJ	OMEGA AJ	MBLKSZ	IBLKSZ	KSTEPS		NJi		
--	---------	----------	--------	--------	--------	--	-----	--	--

### Examples:

EIGC	14	CLAN							
		+5.6					4		
EIGC	15	HESS					6		

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of complex eigenvalue extraction. (Character: "INV", "HESS", or "CLAN")
INV	Inverse power.
HESS	Upper Hessenberg. For linear perturbation of ANALYSIS= DCEIG with large displacement, please don't use HESS. See Remarks 2. and 3.
CLAN	Complex Lanczos. For linear perturbation of ANALYSIS= DCEIG with large displacement, CLAN is recommended. See Remark 8.
IRAM	Implicitly Restarted Arnoldi Method. See Remark 13.
NORM	Method for normalizing eigenvectors. See Remark 7. (Character: "MAX" or "POINT"; Default = "MAX")
MAX	Normalize the component having the largest magnitude to a unit value for the real part and a zero value for the imaginary part.
POINT	Normalize the component defined in fields 5 and 6 to a unit value for the real part and a zero value for the imaginary part. The value for NORM defaults to "MAX" if the magnitude of the defined component is zero.
G	Grid or scalar point identification number. Required if and only if NORM = "POINT". (Integer > 0)

Descriptor	Meaning
C	Component number. Required if and only if NORM = "POINT" and G is a geometric grid point. ( $0 \leq \text{Integer} \leq 6$ )
E	Convergence criterion. (Real $\geq 0.0$ . Default values are: $10^{-4}$ for METHOD = "INV", $10^{-15}$ for METHOD = "HESS", E is machine dependent for METHOD = "CLAN".)
MBLKSZ	Maximum block size. See Remark 11. (Default = 7; Real $\geq 0.0$ )
IBLKSZ	Initial block size. See Remark 11. (Default = 2; Real $\geq 0.0$ )
KSTEPS	Frequency of solve. (Default = 5; Integer $> 0$ )

Table 9-11 Relationship Between METHOD Field and Other Fields

Field	METHOD Field		
	HESS	INV	CLAN
NDj (Integer $> 0$ )	Desired number of eigenvalues and eigenvectors. (No Default)	Desired number of roots and eigenvectors in j-th search region. (Default = $3 \cdot \text{NE}_j$ )	Desired number of roots and eigenvectors to be extracted at j-th shift point. (No Default)
ALPHAAj OMEGAAj Real and imaginary parts of $A_j$ in radians per unit time. (Real).	Not used	End point $A_j$ of j-th search region in complex plane. (Default = 0.0)	j-th shift point. (Default = 0.0)
ALPHABj OMEGABj Real and imaginary parts of $B_j$ in radians per unit time. (Real).	Not used	End point $B_j$ of j-th search region in complex plane. (Default = 0.0)	See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.
Lj (Real $> 0.0$ )	Not used	Width of j-th search region. (Default = 1.0)	See alternate definitions in fields MBLKSZ and IBLKSZ below; see also Remark 11.
NEj (Integer $> 0$ )	Not used	Estimated number of roots in j-th search region.	Not used
MBLKSZ For block CLAN only			Maximum Block Size Default = 7
IBLKSZ For block CLAN only			Initial Block Size Default = 2

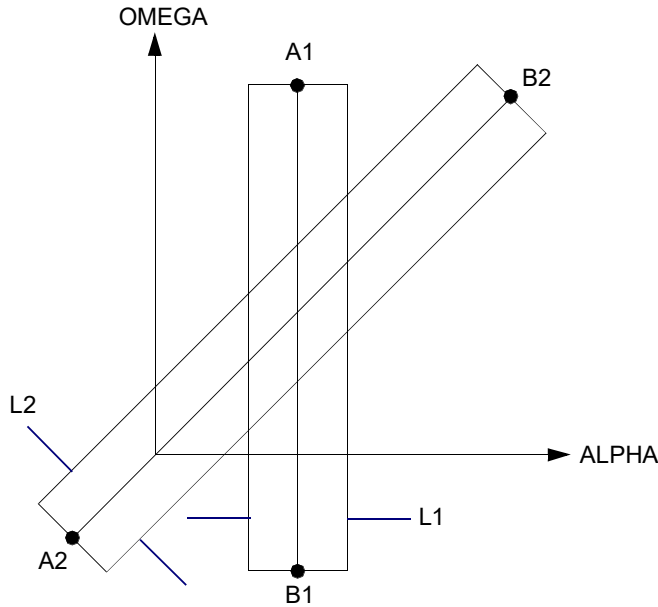


Figure 9-90 Sample Search Regions

**Remarks:**

1. The EIGC entry must be selected in the Case Control Section with the command CMETHOD = SID. Methods of solution are also controlled by SYSTEM(108) on the NASTRAN statements.

SYSTEM(108)	Specification
0 (Default)	QZ HESS method or CLAN block Lanczos, as selected by EIGC entry or equivalent PARAM input. Default value.
1	Force Householder QR (Hessenberg with spill, mass matrix must be nonsingular)
2	Force old single vector complex Lanczos
4	Force new block complex Lanczos
8	Debugging output for Lanczos methods
16	Turn off block reduction in block complex Lanczos
32	Turn off block augmentation in block complex Lanczos
64	Turn of full orthogonality in block complex Lanczos

SYSTEM(108)	Specification
128	Turn off preprocessing of initial vectors in block complex Lanczos
256	Force LR method (Hessenberg, no spill, mass matrix must be invertible)
512	Force QZ method
65536	Use semi-algebraic sort on imaginary part of roots (pre-V70.6 sort)

The word “force” above implies that the method selected by the system cell will be used even when another method has been selected on an EIGC entry. Sums of these values will produce two or more actions at once, when feasible. As the numbers get larger, the function is more developer-oriented than user-oriented.

- When using METHOD = “HESS”, the following should be noted:  
The “HESS” method is generally more reliable and economical for small and moderate-size problems. It computes ND eigenvalues and eigenvectors.  
For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, HESS may yield incorrect results. Therefore, please avoid to use HESS in linear perturbation analysis with large displacement.
- The EIGC entry may or may not require continuations as noted below.
  - For the “HESS” method, continuations are not required; and their contents are ignored when present, except for ND1. However, it is recommended that continuations are not used.
  - For the “CLAN” method when the continuation entry is not used a shift is calculated automatically. When a shift is input on the first continuation entry it is used as the initial shift. Only one shift is used. Data on other continuation entries is ignored.
  - For METHOD = “INV”, each continuation defines a rectangular search region. Any number of regions may be used and they may overlap. Roots in overlapping regions will not be extracted more than once.
  - For all methods, if no continuation is present, then ND0 must be specified on the first entry. If a continuation is present, then NDj must be specified on the continuation and not on the first entry.
- The units of ALPHA AJ, OMEGA AJ, ALPHA BJ, and OMEGA BJ are radians per unit time.
- DIAG 12 prints diagnostics for the inverse power method, the complex Lanczos method and the QZ HESS method.
- If METHOD = “HESS” and the LR or QR methods (non-default methods) are selected by system cell 108 the mass matrix must be nonsingular. The default QZ method does not require a nonsingular mass matrix.
- The normalized eigenvectors may be output with the SDISPLACEMENT and/or DISPLACEMENT Case Control commands.

8. When using METHOD = CLAN, the following should be noted. The modern CLAN method (default for METHOD entry of CLAN) has been enhanced to include a block complex Lanczos approach. This method is more reliable and will not accept inaccurate roots which the old method had a tendency to do. Thus, given the same input, the new method may often accept fewer roots. For continuity the old method has been maintained and may be selected by setting SYSTEM(108). For linear perturbation of ANALYSIS= DCEIG with large displacement (param, lgdisp, 1) in SOL 400, CLAN is recommended.
9. The SVD method is provided for DMAP applications. If used in solution 107 or 110, and mass or damping terms are present, a user fatal exit is taken. See the MSC Web site for the Flight Loads Product examples on the use of the SVD method. The SVD operation decomposes the input stiffness matrix K into the factors U, S, and V. U and V are collections of vectors of orthogonal functions. S is a rectangular matrix with terms on the diagonal of its left partition. The factors satisfy the equation  $K = U*S*V'$ , where " ' " implies complex conjugate transpose. The ND1 value has a meaning for the SVD functions which differs from eigensolution.

ND1	OUTPUT
>0	All vectors of U and V are output.
=0	U and V are returned in a purged state.
<0	S is returned as a square matrix whose number of columns is equal to the minimum number of rows or columns of the input matrix. U and V are truncated to be commensurate with S. This is a method to reduce the costs of solving very rectangular input matrices by providing a partial solution for the most interesting vectors.

10. For DMAP applications there are input parameters, not present in the solution sequences, that may be used to replace the function of the EIGC and CMETHOD entries. See the MSC website for details.
11. The MBLKSZ and IBKLSZ parameters are integers in concept, but must be input at real numbers (that is, with a decimal point.) They represent maximum sizes, and may be reduced internally for small size problems.
12. The IRAM method is an out-of-core implementation of the public domain ARPACK complex eigenvalue solver. To use IRAM, the mass matrix has to be symmetric or Hermitian, and the number of modes desired can be at most the order of the generalized eigenproblem minus one. IRAM does not compute the left eigenvectors and thus is not suitable for SOL200 design optimization analysis.
13. For the "IRAM" method, only the fields that contain EIGC, SID, METHOD, NORM, ND0, NDJ are meaningful. The user selects the method by specifying IRAM in the METHOD field, the eigenvector normalization scheme in the NORM field, and the number of desired modes in the ND0 or the NDj field. The other fields are irrelevant.
14. If Modules are present then this entry may only be specified in the main Bulk Data section.

**EIGP** Poles in Complex Plane

Defines poles that are used in complex eigenvalue extraction by the Determinant method.

**Format:**

1	2	3	4	5	6	7	8	9	10
EIGP	SID	ALPHA1	OMEGA1	M1	ALPHA2	OMEGA2	M2		

**Example:**

EIGP	15	-5.2	0.0	2	6.3	5.5	3		
------	----	------	-----	---	-----	-----	---	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
ALPHA <sub>i</sub> ,OMEGA <sub>i</sub>	Coordinates of point in complex plane. (Real)
M <sub>i</sub>	Multiplicity of complex root at pole defined by point at ALPHA <sub>i</sub> and OMEGA <sub>i</sub> . (Integer > 0)

**Remarks:**

1. The EIGP entry defines poles in the complex plane that are used with an associated EIGC entry having the same set number.
2. The units of ALPHA<sub>i</sub> and OMEGA<sub>i</sub> are radians per unit time.
3. Poles are used only in the determinant method. (METHOD = "DET" on the EIGC entry).
4. One or two poles may be defined on a single entry.
5. See *The NASTRAN Theoretical Manual*, Section 10.3.4, for details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.



## EIGR Real Eigenvalue Extraction Data

Defines data needed to perform real eigenvalue analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
EIGR	SID	METHOD	F1	F2	NE	ND			
	NORM	G	C						

### Example:

EIGR	13	LAN				12			
------	----	-----	--	--	--	----	--	--	--

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
METHOD	Method of eigenvalue extraction. (Character) Modern Methods: LAN        Lanczos Method AHOU      Automatic selection of HOU or MHOu method. See Remark 13. Obsolete Methods: INV        Inverse Power method. SINV      Inverse Power method with enhancements. GIV        Givens method of tridiagonalization. MGIV      Modified Givens method. HOU        Householder method of tridiagonalization. MHOu      Modified Householder method. AGIV      Automatic selection of METHOD = "GIV" or "MGIV". See Remark 13.
NORM	Method for normalizing eigenvectors. (Character: "MASS," "MAX," or "POINT"; Default = "MASS") MASS      Normalize to unit value of the generalized mass. (Default) MAX        Normalize to unit value of the largest component in the analysis set. POINT     Normalize to a positive or negative unit value of the component defined in fields 3 and 4. The POINT option is not supported for METHOD=LAN. (Defaults to "MASS" if defined component is zero.)

Describer	Meaning
G	Grid or scalar point identification number. Required only if NORM = "POINT". (Integer > 0)
C	Component number. Required only if NORM = "POINT" and G is a geometric grid point. (1 ≤ Integer ≤ 6)

Table 9-12 Relationship Between METHOD Field and Other Fields for Obsolete Methods

Field	METHOD Field	
	INV or SINV	GIV, MGIV, HOU, or MHO
F1, F2	Frequency range of interest. F1 must be input. If METHOD = "SINV" and ND, is blank, then F2 must be input. See also Remark 21. (Real ≥ 0.0)	Frequency range of interest. If ND is not blank, F1 and F2 are ignored. If ND is blank, eigenvectors are found with natural frequencies that lie in the range between F1 and F2. (Real ≥ 0.0; F1 < F2)
NE	Estimate of number of roots in range (Required for METHOD = "INV"). Not used by "SINV" method. (Integer > 0)	Not used.
ND	Desired number of roots. If this field is blank and METHOD = "SINV", then all roots between F1 and F2 are searched and the limit is 600 roots. (Integer > 0, Default is 3 · NE for METHOD = "INV" only.)	Desired number of eigenvectors. If ND is zero, the number of eigenvectors is determined from F1 and F2. If all three are blank, then ND is automatically set to one more than the number of degrees-of-freedom listed on SUPORTi entries. (Integer ≥ 0; Default = 0)

**Remarks:**

1. The EIGR entry must be selected with the Case Control command METHOD = SID.
2. See [Real Eigenvalue Analysis](#) in the *MSC Nastran Dynamic Analysis Guide* for a discussion of method selection.
3. The units of F1 and F2 are cycles per unit time.
4. The continuation entry is optional. If the continuation entry is not specified, then mass normalization is performed.
5. The contemporary methods are LAN and AHOU. The other methods are in a maintenance-only status, with no enhancements planned for them. They may be eliminated in a future release of Nastran.

6. The LAN method is the most general-purpose method, and may be used on both small- and large-size problems. It takes advantage of sparsity of input matrices, leading to greater efficiency on large-size problems. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHOU. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry. The NE, G, and C fields are ignored for the LAN method. The NORM field may be set to MASS (the default value) or MAX. The conventions used when both the Fi and ND fields are specified are described in [Table 9-13](#) of the EIGRL entry description. The EIGRL entry is an alternate method to select the LAN method. It has several other input options for special cases. When both and EIGRL and EIGR have the same SID and that SID is selected by a METHOD command the EIGRL entry takes precedence.
7. The AHOU method is competitive with the LAN method when there are small, dense matrices and many eigenvectors are required. This most commonly occurs when static or dynamic reduction is performed. The AHOU method does not take advantage of matrix sparsity, so that computation cost rises with the cube of the number of DOFs. The AHOU method responds to all permitted values for all the other fields except NE, which is ignored.
8. All methods require a positive semi-definite (psd) mass matrix for stable solutions. The mass matrix may be tested for this condition for all methods of solution by setting SYSTEM(303). A value of “-4” should be sufficient to identify problem matrices. A fatal error exit is taken when it is not met. All Nastran metric elements are designed to produce psd mass matrices. CMASSi elements, DMIG matrices selected by the M2GG command, and matrices input via INPUTT4 are special methods that allow addition of non-psd terms by use of non-metric element input. If none of this type of special input is present and the fatal error exit is taken you may have encountered an error in a metric element. Contact your local MSC technical representative for corrective action in this case.
9. The LAN and AHOU methods allow singular but positive semi-definite mass matrices.
10. The tridiagonal methods include the xGIV and xHOU methods, where “x” is described in the following comments. All tridiagonal methods compute all eigenvalues, and the number of eigenvectors specified by the Fi and Nd fields, as described in [Table 9-12](#).
11. If “x” is blank (for example, the HOU method is selected) the mass matrix must be non-singular.
12. If “x” is M (for example, the MHOU method is selected) the mass matrix may be singular. A modified, shifted problem is solved in an inverse basis with this method. Some precision in the solution and longer computation time is exchanged for a more stable solution.
13. If “x” is A (for example, the AHOU method is selected) an attempt is made to solve the problem without shifting, in the interest of cost reduction and maximum precision. If the mass matrix is determined to be poorly conditioned for inversion the basis is automatically shifted with the modified method.
14. If NORM = “MAX”, components that are not in the analysis set may have values larger than unity.
15. If NORM = “POINT”, the selected component should be in the analysis set (a-set). (The program uses NORM = “MAX” when it is not in the analysis set.) The displacement value at the selected component will be positive or negative unity.

16. The “SINV” method is an enhanced version of the “INV” method. It uses Sturm sequence number techniques to make it more likely that all roots in the range have been found. It is generally more reliable and more efficient than the “INV” method.
17. For the “INV” and “SINV” methods, convergence is achieved at  $10^{-6}$ . Convergence is tested by other criteria for the other methods.
18. For the “SINV” method only, if F2 is blank, the first shift will be made at F1, and only one eigensolution above F1 will be calculated. If there are no modes below F1, it is likely that the first mode will be calculated. If there are modes below F1 (including rigid body modes defined by SUPORT entries), a mode higher than the first mode above F1 may be calculated.
19. When F1, F2, and ND are all zero or blank, ND is reset to 1. A User Warning Message is produced for this condition, which is interpreted as likely to be due to an inadvertent omission by the user.
20. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the F1 and F2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
21. F2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.
22. For large sized problems, Lanczos is the most efficient and is the recommended method for large problems.
23. It is not recommended to use AHOU, HOU, MHOU if RIGID=LAGRAN case control.
24. If Modules are present then this entry may only be specified in the main Bulk Data section.

## EIGRL Real Eigenvalue Extraction Data, Lanczos Method

Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method.

### Format:

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCL	NORM	
option_1 = value_1    option_2 = value_2,    etc.									

### Example:

EIGRL	1	0.1	3.2	10					
NORM=MAX    NUMS=2									

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
V1, V2	For vibration analysis: frequency range of interest. For buckling analysis: eigenvalue range of interest. See Remark 4. (Real or blank, $-5 \times 10^{16} \leq V1 < V2 \leq 5. \times 10^{16}$ )
ND	Number of roots desired. See Remark 4. (Integer > 0 or blank)
MSGLVL	Diagnostic level. ( $0 \leq$ Integer $\leq 4$ ; Default = 0)
MAXSET	Number of vectors in block or set. See Remark 13.
SHFSCL	Estimate of the first flexible mode natural frequency. See Remark 9. (Real or blank)
NORM	Method for normalizing eigenvectors (Character: "MASS" or "MAX")
	<div style="display: flex; align-items: flex-start;"> <div style="width: 100px; padding-right: 10px;">MASS</div> <div>Normalize to unit value of the generalized mass. Not available for buckling analysis. (Default for normal modes analysis.)</div> </div>
	<div style="display: flex; align-items: flex-start;"> <div style="width: 100px; padding-right: 10px;">MAX</div> <div>Normalize to unit value of the largest displacement in the analysis set. Displacements not in the analysis set may be larger than unity. (Default for buckling analysis.)</div> </div>
ALPH	Specifies a constant for the calculation of frequencies (Fi) at the upper boundary segments for the parallel method based on the following formula. See Remark 12. (Real > 0.0; Default = 1.0):
	$\text{ALPH} < 1.0, \quad Fi = V1 + (V2 - V1) \frac{1 - \text{ALPH}^i}{1 - \text{ALPH}^{\text{NUMS}}}$
	$\text{ALPH} = 1.0, \quad Fi = V1 + (V2 - V1) \frac{i}{\text{NUMS}}$
NUMS	Number of frequency segments for the parallel method. (Integer > 0; Default = 1)

Descriptor	Meaning
Fi	Frequency at the upper boundary of the i-th segment. See Remark 12. (Real or blank; $V1 < F1 < F2 < \dots < F15 < V2$ )
option_i= value_i	Assigns a value to the fields above except for SID. ALPH, NUMS, and Fi must be specified in this format. V1, V2, ND, MSGVLV, MAXSET, SHFSCL, and NORM may be specified in this format as long as their corresponding field is blank in the parent entry.

**Remarks:**

1. Real eigenvalue extraction data sets must be selected with the Case Control command METHOD = SID.
2. The units of V1 and V2 are cycles per unit time in vibration analysis, and are eigenvalues in buckling analysis. Each eigenvalue is the factor by which the prebuckling state of stress is multiplied to produce buckling in the shape defined by the corresponding eigenvector.
3. NORM = "MASS" is ignored in buckling analysis and NORM = "MAX" will be applied.
4. The roots are found in order of increasing magnitude; that is, those closest to zero are found first. The number and type of roots to be found can be determined from Table 9-13.

Table 9-13 Number and Type of Roots Found with EIGRL Entry

V1	V2	ND	Number and Type of Roots Found
V1	V2	ND	Lowest ND or all in range, whichever is smaller.
V1	V2	blank	All in range
V1	blank	ND	Lowest ND in range $[V1, +\infty]$
V1	blank	blank	Lowest root in range $[V1, +\infty]$
blank	blank	ND	Lowest ND roots in $[-\infty, +\infty]$
blank	blank	blank	Lowest root. See Remark 10.
blank	V2	ND	Lowest ND roots below V2
blank	V2	blank	All below V2

5. In vibration analysis, if  $V1 < 0.0$ , the negative eigenvalue range will be searched. (Eigenvalues are proportional to  $V_i$  squared; therefore, the negative sign would be lost.) This is a means for diagnosing improbable models. In buckling analysis, negative V1 and/or V2 require no special logic.
6. Eigenvalues are sorted on order of magnitude for output. An eigenvector is found for each eigenvalue.
7. MSGVLV controls the amount of diagnostic output during the eigenvalue extraction process. The default value of zero suppresses all diagnostic output. A value of one prints eigenvalues accepted at each shift. Higher values result in increasing levels of diagnostic output.

8. In vibration analysis, if V1 is blank, all roots less than zero are calculated. Small negative roots are usually computational zeroes which indicate rigid body modes. Finite negative roots are an indication of modeling problems. If V1 is set to zero, negative eigenvalues are not calculated.
9. A specification for SHFSCL may improve performance, especially when large mass techniques are used in enforced motion analysis. Large mass techniques can cause a large gap between the rigid body and flexible frequencies. If this field is blank, a value for SHFSCL is estimated automatically.
10. On occasion, it may be necessary to compute more roots than requested to ensure that all roots in the range have been found. However, this method will not output the additional roots.
11. NASTRAN SYSTEM(146) (also known as FBSMEM) provides options for I/O reduction during FBS operations according to the following table.

Table 16 SYSTEM(146) Options

SYSTEM(146) (default=0)	Description
-1 or 0	Use available memory to store as much of the factor matrix as possible during FBS operations. System Information Message 4199 is printed in the F04 file.
1	Disable all special memory operations for FBS.
2	Increase memory reserved for sparse method by approximately 200%.
3	Increase memory reserved for sparse method by approximately 300%.
4	Increase memory reserved for sparse method by approximately 400%.

12. For the distributed parallel method, the frequency range between V1 and V2 may be subdivided into segments that can then be analyzed in parallel. V1 and V2 must be specified for the parallel method. NUMS must be specified greater than 1 to take advantage of the parallel method. NUMS may also be specified on the NUMSEG keyword of the NASTRAN statement. Currently, NUMSEG must equal the number of processors and by default NUMSEG is set to the number of processors requested by the DMP keyword. If both are specified, then NUMS takes precedence.

The upper frequencies of each segment may be generated automatically by ALPH or specified directly in Fi. If both are specified, then Fi takes precedence over ALPH as long as they are consistent. ALPH if multiplied by 100 may also be specified on FRQSEQ keyword of the NASTRAN statement.

13. Increasing MAXSET may improve performance for large problems where a large number of eigenvalues are being found. The default is 7 on all machines. SYSTEM(263) may be set in an rcfile to effectively modify the default; however the setting on the EIGRL entry always takes precedence.

The optimal MAXSET, also known as Lanczos block size, should be at least one more than the number of repeated eigenvalues. For example, for a model that has six rigid body modes (six zero-frequency eigenvalues), the recommended block size is 7. Additionally, the MAXSET value can also be increased to help with numerical orthogonalization during the Lanczos recurrence, for models with

many closely spaced roots. Note that there must be sufficient memory to keep 'MAXSET' vectors in memory; otherwise, the MAXSET (Lanczos block size) will be reduced automatically. Maximum value for MAXSET is 30. Activate Lanczos message level 1 (EIGRL field 6) to see Lanczos diagnostic messages. Note also that DIAG 16 is equivalent to Lanczos message level 1.

14. SYSTEM(196), keyword SCRSAVE, controls reuse of scratch files when segment logic is invoked. SYSTEM(196) is useful only when multiple frequency segments are requested on a Lanczos run. (Multiple frequency segments can be requested via the NUMS field in the EIGRL entry and by SYSTEM(197).) Each frequency segment requires a minimum of three scratch files. When multiple frequency segments are used on a single processor computer then each frequency segment is solved serially. In this case, it makes sense to let segment #2 use the scratch files which were used by segment #1 since work for segment #1 has been completed (otherwise it wouldn't be working on #2). Similarly, when work for segment #2 is finished, segment #3 should be able to use #2's scratch files. SYSTEM(196)=1 allows such file reuse and is considered a safe default on Version 70 and later systems.
15. The new buckling shift logic in Version 70.5 tends to shift to 1.0 first. The logic may have difficulty finding the lowest ND roots if a problem requests a small number of roots (ND) when there are thousands of roots below 1. In this case either the loading should be scaled, SHFSCL specified, or a smaller frequency range requested.
16. Because Lanczos performance is tuned for medium to large problems, this has caused difficulties with very small problems. Thus, by default, on problems with fewer than 20 degrees-of-freedom when the LAN method is selected, the method is switched to AHO. The criteria for automatic switching is controlled by SYSTEM(359) on the NASTRAN entry.
17. V2 must be specified if DOMAINSOLVER ACMS or DOMAINSOLVER MODES is also specified in the Executive Control Section.
18. For buckling solutions, the Lanczos algorithm is most reliable if it is allowed to compute the lowest mode first, then the remainder in increasing order of magnitude. The lowest mode is usually the mode of most interest. If the V1 and V2 fields are left blank, an estimate of the first mode eigenvalue is found by a robust, efficient inverse iteration. If ND is set to 1, there is a high probability that the first mode will be computed reliably. If more modes are needed it is easier to fill out the other fields correctly once the location of the lowest mode is known.
19. For buckling solutions, if V1 is explicitly set to 0.0, negative eigenvalues will be calculated based on a negative shift value. This is done to avoid singularities resulting from a shift at 0.0. If V1 is set to a small positive value, then negative eigenvalues will not be calculated. This behavior was changed with the MSC Nastran 2010 release whereas before setting V1 to 0.0 yielded no negative eigenvalues. The best approach, however, is to follow the suggestion from Remark 19.
20. If Modules are present then this entry may only be specified in the main Bulk Data section.



**ELEMUDS****Element Property User Defined Service or Subroutine**

Allows the user to provide element property routines for use with specified Nastran elements.

**Format:**

1	2	3	4	5	6	7	8	9	10
ELEMUDS	PID	PTYPE	GROUP	UNAME	DEPEN	NMLSPD			
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	“CHAR”	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

**Examples:**

*In FMS Section of Nastran input stream:*

```
CONNECT SERVICE ELEMENT 'SCA.MDSolver.Util.Ums'
```

*In Bulk Data:*

ELEMUDS	17	PBUSH2D	ELEMENT	THPAD	FREQ				
---------	----	---------	---------	-------	------	--	--	--	--

**Describer****Meaning**

PID	Element property identification number that matches the identification number on a PBUSH2D element property entry. (Integer > 0)
PTYPE	The name of the element entry. Currently supported element property entry is: PBUSH2D. (Character; no Default)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. See Remark 6.(Character)
DEPEN	Specifies if element is frequency dependent. DEPEN=NOM or blank; user element is not frequency dependent. DEPEN=FREQ; user element is frequency dependent. (Character default=NOM)
NMLSPD	Nominal speed (cycles per second) used by the CBUSH2DA to calculate nominal stiffness, mass and damping terms. Nominal terms will be used during reduction and real eigenvalue solution. In a frequency-dependent problem, updated values will be calculated at each frequency.
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified property entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)

RDATAi	Additional user supplied Real data not already existing on the specified property entry. (Real; no Default)
“CHAR”	Keyword indicating that the following data is Character. (Character)
CDAi	Additional user supplied Character data not already existing on the specified property entry. (Character; no Default)

**Remarks:**

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing PID.
4. A CDAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
5. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
6. UNAME must be truncated to 8 characters in the bulk data field
7. If the ROMAC interface is being used then UNAME=THPAD is required. See the THPAD Bulk Data entry (p. 3240) for a description of ROMAC service. (<http://www.virginia.edu/romac/>)

**ELEMUDS****Element Property User Defined Service or Subroutine - SOL 600**

Allows the user to provide element property routines for use with enhanced nonlinear elements in SOL 600.

**Format:**

1	2	3	4	5	6	7	8	9	10
ELEMUDS	PID	PType	GROUP	UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	“CHAR”	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

**Examples:**

ELEMUDS	17	PSHELL	UELEM	RTN1					
ELEMUDS	17	PSHELL	UELEM1	RTN3					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

<b>Describer</b>	<b>Meaning</b>
PID	Element property identification number that matches the identification number on a PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD element property entry. (Integer > 0)
PType	The name of the element entry. Currently supported element property entries include: PSHELL, PSOLID, PCOMP, PCOMPG, PBAR, PBEAM, PSHEAR, or PROD. (Character; no Default)
GROUP	The group name used for the FMS section CONNECT SERVICE entry. (Character; no Default)
UNAME	A primary or secondary name to identify the user subroutine. See Remark 6. (Character; Default blank)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATA <sub>i</sub>	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

“CHAR”	Keyword indicating that the following data is Character. See Remark 7. (Character)
CDATAi	Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

**Remarks:**

1. This entry triggers the call to a user element property subroutine for advanced nonlinear materials.
2. PID must match an existing PID.
3. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”.
4. Certain user subroutines may require integer, real, or character data input as specified in the User Defined Services manual.
5. The ELEMUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. EVAL will be stored as a character\*16 name in common block /userch/.
6. UNAME identifies the user subroutine name to be called.
7. Character fields CDATAi are ignored in SOL 600 are not passed to the user subroutine.

## ELIST Element List

Defines a list of CQUAD4 and CTRIA3 structural elements for virtual fluid mass.

### Format:

1	2	3	4	5	6	7	8	9	10
ELIST	LID	E1	E2	E3	E4	E5	E6	E7	
	E8	E9	E10	-etc.-					

### Example:

ELIST	3	51	-62	68	THRU	102	122		
-------	---	----	-----	----	------	-----	-----	--	--

Describer	Meaning
LID	Identification number of list. (Integer > 0)
Ei	Identification number of a structural element. See Remark 1. for the meaning of the negative sign. The string "THRU" may be used to indicate that all existing elements between those referenced in the preceding and succeeding fields are in the list. (Integer ≠ 0 or "THRU")

### Remarks:

1. If the ELIST entry is referenced by field 6 of an MFLUID entry, the wetted side of the element is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal as determined by applying the right-hand rule to the sequence of its corner points. If the "THRU" option is used, then immediately preceding and succeeding elements must have the same sign.
2. Large open "THRUs" should be avoided.
3. The word "THRUs" must not appear in field 2 or 9 on the parent entry or on any continuations.
4. If any ELIST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, ELIST entry changes or additions are not recommended on restart.

## ENDDATA

### Bulk Data Delimiter

Designates the end of the Bulk Data Section.

**Format:**

ENDDATA

**Remark:**

1. ENDDATA is optional.

**ENDDYNA**

Defines the End of Direct Text to Dytran

All entries between TODAYNA and ENDDYNA will be passed directly by SOL 700 to Dytran. Used in SOL 700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
ENDDYNA									

**Example:**

TODYNA									
MAT1	345	29.0E6	0.285	0.0004					
ENDDYNA									

Describer	Meaning
TODYNA	
MAT1	
ENDDYNA	

**Remarks:**

See [TODYNA](#) for details of how this entry is used.

## ENTUDS User Defined Logic at Entry Point

Calls user defined logic within a SCA service at the point specified within the solution sequence.

### Format:

1	2	3	4	5	6	7	8	9	10
ENTUDS	ENTID	ENTPNT	GROUP						
	"INT"	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	"REAL"	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	"CHAR"	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

### Example:

ENTUDS	1	RCENT1	MY_FUNC						
	INT	2	17						
	REAL	.5	.25						

Describer	Meaning
ENTID	Entry point identification number (Integer > 0)
ENTPNT	The point of entry in the solution sequence. Acceptable values for now: "RCENT1", "RCENT2", "RCOUT1", "RCEXEC1". (Character; Required)
GROUP	The SCA group name used to identify the service (Character; Required)
"INT"	Key word indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
"REAL"	Key word indicating that the following data is real. (Character)
RDATA <sub>i</sub>	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)
"CHAR"	Key word indicating that the following data is Character. (Character)
CDATA <sub>i</sub>	Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

### Remarks:

1. This entry is for RC Network solver only.
2. In SINDA input file, there are 4 entry points for users to input customized logics.
  - a. RCENT1 ---- Variable 1 block



- b. RCENT2 ---- Variable 2 block
- c. RCOUT1 ---- Output block
- d. RCEXEC1 ---- Execution block

## EOSDEF

Deflagration - SOL 700 only

EOSDEF defines the properties of the deflagration equation of state, and the reaction rate to model the burning of solid propellants. The burning of the solid propellant produces hot gas.

### Format

1	2	3	4	5	6	7	8	9	10
EOSDEF	MID	GAMMA	B	R	$C_v$	$C_p$	E	RHOS	
	RHOF	W	BETA	SAVR	X	Y			

### Example

1	2	3	4	5	6	7	8	9	10
EOSDEF	3	1.123	0.001	304			IE+6	1600	
	0.1	4E-8	0.85	6000	0.6	0.0			

Field	Contents
MID	Unique material number. (Integer > 0; Required).
GAMMA	Constant $\gamma$ . (Real>0; Required)
B	Gas co-volume. (Real $\geq$ 0; Required)
R	Gas constant. (Real>0; Refer to remark 2.)
CV	Specific heat at constant volume. (Real>0; Refer to remark 3.)
CP	Specific heat at constant pressure. (Real>0; Refer to remark 3.)
E	Chemical energy per unit burned mass. (Real>0.0; Required)
RHOS	Reference density of propellant. (Real>0; Required).
RHOF	$\frac{\text{Density of powder}}{\text{reference density of propellant}}$ (Real>0; Refer to remark 5.)
W	Burning rate coefficient (Real $\geq$ 0; Required).
BETA	Burning rate exponent. (Real $\geq$ 0; Required).
SAVR	Initial surface area divided by volume (Real>0; Required).
X	Parameter form function(Real>0; Required).
Y	Parameter form function(Real>0; Required).

## Remarks

1. EOSDEF – The deflagration Equation of State is explained in *Dytran Theory Manual*, Chapter 4: Models.
2. This equation of state is used with Eulerian elements.
3. The temperature of gas is calculated when one of the constants  $R$ ,  $C_v$  or  $C_p$  are specified. When temperature is not mentioned in an output request, omit the he constants .
4. The pressure in the reaction products is defined by the Noble-Abel equation of state as follows:

$$\rho = (Y - 1) \frac{P}{1 - bP} e \text{ for reacted product,}$$

$$T = \frac{(Y - 1)e}{R} \text{ where } Y, b \text{ are constants and } R \text{ is the gas constant.}$$

The chemical reaction rate for conversion of un-reacted explosive to reaction products is mentioned the relation below:

$$\xi = wSAVR \quad \text{vivacity}$$

$$\phi = (1 - F)^X + YF \quad \text{form function}$$

$$\frac{dF}{dt} = \xi \phi P^\beta \quad \text{time derivative of burn fraction}$$

where;

$$w = \quad \text{burning rate coefficient}$$

$$\beta = \quad \text{burning rate exponent}$$

$$SAVR = \quad \text{initial surface area divided by volume}$$

$$Y = \quad \text{parameter form function}$$

$$S = \quad \text{parameter form function}$$

5. RHOF allows taking into account the air between gunpowder grains. Also, it allows taking into account air inside the combustion chamber. For example If the chamber is filled with 10% real propellant and the rest is filled with air then RHOF = 0.1.
6. To indicate what Eulerian regions can burn, the variable DEFMAT can be used. In regions that can burn DEFMAT=1. DEFMAT can be set on the TICVAL card:  
 TICVAL,14,,DENSITY,1.14,SIE,300000.2,DEFMAT,1.0
7. Ignition of the propellant approached by setting SIE to the flame temperature in specific areas:  
 TICVAL,15,DENSITY,1.14,SIE,4231908.591,DEFMAT,1.0

Where  $SIE_{\text{ignition}} = \text{Flame Temp} \times C_v$

Flame Temp can be extracted from Chemical Energy per unit burned mass and Gas constant:  $T_{\text{flame}}$   
 $= E / R .$

**EOSGAM****Gamma Law Gas Equation of State - SOL 700 only**

Defines the properties of a Gamma Law equation of state where the pressure  $p$  is defined as:

$$p = (\gamma - 1)\rho e$$

where:

$e$  = specific internal energy per unit mass

$\rho$  = overall material density

$\gamma$  = A constant

Used in SOL 700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
EOSGAM	EID	GAMMA	R	CV	CP	VISC			

**Example:**

EOSGAM	2	1.4	287.0						
--------	---	-----	-------	--	--	--	--	--	--

Describer	Meaning
EID	Unique equation of state number. (Integer > 0, Required)
GAMMA	Constant $\gamma$ . (Real $\geq 0$ , default=blank)
R	Gas constant. See Remarks 1. and 3. (Real > 0, default=blank)
CV	Specific heat at constant volume. See Remarks 1. and 3. (Real > 0, default=blank)
CP	Specific heat at constant pressure. See Remarks 1. and 3. (Real > 0, default=blank)
VISC	Viscosity coefficient. If zero, no viscosity. See Remarks 4., 5., and 6. (Real $\geq 0$ , default=blank)

**Remarks:**

1. The temperature of the gas will be calculated when one of the gas constants,  $R$ ,  $C_v$  or  $C_p$  is specified.
2. The Euler variable name for temperature is TEMPTURE.
3.  $\gamma$ ,  $R$ ,  $C_v$  and  $C_p$  have the following relationships:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

4. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient. The SI-unit of viscosity is

$$Pa \cdot s = \frac{N_s}{M^2} = \frac{kg}{ms}$$

5. If possible, use (in coupled analysis) the FASTCOUP coupling algorithm because viscous fluxes are computed more accurately for fast coupling than for general coupling.
6. For the single mat solver, viscous stresses can be requested by the use of TXX through TZX. Also, EFFSTS is available. For the multi-material solver, viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZX-VIS. This viscous stresses depend only on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stress. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.

## EOSIG

## Ignition and Growth Equation of State - SOL 700 only

Defines the properties of Ignition and Growth equation of state and the reaction rate equation used to model high explosives. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
EOSIG	EID	UNITDEF	DBEXPL	UNITCNV					+
+		AE	BE	R1E	R2E	OMGE	I	G	+
+	A	AP	BP	R1P	R2P	OMGP	X	Y	+
+	Z	R	ECHEM	PRSTOL	ITRMAX				

### Example:

EOSIG	10		LX17	SI					
-------	----	--	------	----	--	--	--	--	--

Describer	Meaning
EID	Unique equation of state number referenced from MATDEOL. (Integer > 0; Required)
UNITDEF	User-defined default unit for the inputs: See Remarks 2. and 3. (Character)
	CCGMS      m/g/μs units
	SI            International System units
	METRIC     Metric units
	IMPER       imperial units
	MMMG       mm/mg/μs units
DBEXPL	Use explosive material from the database (See Remarks 4. and 6.) (Character, NO). The following detonation materials are available in the data base:
	NO            The database is not used. See Remark 5.
	P94A         PBX-9404 (a) explosive
	TATB         TATB explosive
	PENT         PETN explosive
	CTNT         Cast TNT explosive
	LCOMPB     LANL COMP B explosive
	MCOMPB     Military COMP B explosive
	P94B         PBX-9404 (b) explosive
	LX17         LX-17 explosive
UNITCNV	User defined conversion units: (Character; see Remarks 2. and 3.)
	CGMS        cm/g/μs units



Describer	Meaning
	SI International System units
	METRIC Metric units
	IMPER Imperial units
	MMMGS mm/mg/ $\mu s$ units
AE	Constant $A_e$ for un-reacted explosive. See Remark 5. (Real; Required)
BE	Constant $B_e$ for un-reacted explosive. See Remark 5. (Real; Required)
R1E	Constant $R_{1e}$ for un-reacted explosive. See Remark 5. (Real; Required)
R2E	Constant $R_{2e}$ for un-reacted explosive. See Remark 5. (Real; Required)
OMGE	Constant $\omega_e$ for un-reacted explosive
I	First ignition coefficient. See Remark 5. (Real; Required)
G	Second ignition coefficient. See Remark 5. (Real; Required)
A	Density ignition coefficient. See Remark 5. (Real; Required)
AP	Constant $A_p$ for reacted product. See Remark 5. (Real; Required)
BP	Constant $B_p$ for reacted product. See Remark 5. (Real; Required)
R1P	Constant $R_{1p}$ for reacted product. See Remark 5. (Real; Required)
R2P	Constant $R_{2p}$ for reacted product. See Remark 5. (Real; Required)
OMGP	Constant $\omega_p$ for reacted product. See Remark 5. (Real; Required)
X	Surface burning exponent. See Remark 5. (Real, 2./9.)
Y	Surface burning exponent. See Remark 5. (Real, 2./3.)
Z	Pressure exponent. See Remark 5. (Real; Required)
R	Relative density exponent. See Remark 5. (Real, .4)
ECHEM	Chemical energy of high explosive per unit mass. See Remark 5. (Real; Required)
PRSTOL	Tolerance for pressure equilibrium iterations in mixed phase elements. (Real > 0, 1.E-6)
ITRMAX	Maximum number of iterations in pressure equilibrium iterations. (Integer > 0, 16)

**Remarks:**

1. This equation of state can only be used with solid Eulerian elements.
2. The definition of the unit system in which the input values are defined is required information only in case you wish to have an automatic conversion to a different unit system as defined by the UNITCNV field. In case you are using the conversion mechanism, note that the density RHO in the corresponding MATDEUL entry will be interpreted in the unit system defined here. [Table 9-14](#) defines sets of units available:



**Table 9-14      Sets of Units Used in the IG Model**

Quantity	CG $\mu$ s	SI	Metric	Imperial	MMMG $\mu$ s
Length	Centimeter (cm)	Meter (m)	Centimeter (cm)	Inch (in)	Millimeter (mm)
Time	Microsecond ( $\mu$ s)	Second (s)	Second (s)	Second (s)	Microsecond ( $\mu$ s)
Mass	Gram (g)	Kilogram (kg)	Gram (g)	Slug (lbf-s <sup>2</sup> /in)	Milligram (mg)
Force	Teradyne	Newton (N)	Dyne	Pound force (lbf)	kN
Density	g/cm <sup>3</sup>	Kg/m <sup>3</sup>	g/cm <sup>3</sup>	lbf-s <sup>2</sup> /in <sup>4</sup>	mg/ mm <sup>3</sup>
Stress	Mbar	Pascal (Pa)	$\mu$ bar	Lbf/in <sup>2</sup>	GPa
Energy	1012 erg (Mbars-cm <sup>3</sup> )	Joule (J)	Erg	Lbf-in	J
Temperature	Kelvin (K)	Kelvin (K)	Kelvin (K)	Kelvin (K)	Kelvin (K)

3. The UNITCNV field defines the unit system to which the material parameters are converted. In case you are not using one of the database material models, you also have to define the default unit system (UNITDEF) in which you supplied the data.
4. You can use the database containing several detonation materials to start the analysis. The material data are taken from Lee/Tarver (Ref. 1) and Murphy/Lee (Ref. 2) papers in the Theory Manual. The equations of state parameters are given in the [Table 9-15](#).
5. The default setting for DBEXPL is NO, which means you should define the values in the input fields (fields 12 to 36). If the database material name is defined, all values in the input fields will be overridden. The reference density RHO defined on the corresponding MATDEUL entry will be set to the value from the database.
6. The default unit system for the material database parameters is the CGMS unit system. If you wish to use the material base data in a different unit system, you can specify this by defining the target unit system in the UNITCNV field.
7. You can define the shear property and yield model of the material with respectively SHXXX and YLDXX entry. Note that the unit system of data required in these entries should be consistent with the unit system defined in the UNITCNV field.
8. The IG equation of state cannot be used in combination with a spallation model.
9. The following JWL equation of state is used to calculate the pressure of the un-reacted explosive (in "solid" state):

$$p_e = A_e \left( 1 - \frac{\omega_e \eta_e}{R_{1e}} \right) e^{\frac{-R_{1e}}{\eta_e}} + B_e \left( 1 - \frac{\omega_e \eta_e}{R_{2e}} \right) e^{\frac{-R_{2e}}{\eta_e}} + \omega_e \eta_e \rho_e E_e$$

where:

$\eta_e = \rho_e / \rho_0$	the relative density of the unreacted explosive.
$E_e$	the specified internal energy per unit mass of the unreacted explosive
$\rho_0$	the initial density of the explosive
$A_e, B_e, \omega_e, R_{1e}, R_{2e}$	the input constants of the unreacted explosive

Similarly, the pressure in the reaction products (in “gas” state) is defined by another JWL form as follows:

$$p_p^1 = A_p \left( 1 - \frac{\omega_p \eta_p}{R_{1p}} \right) e^{\frac{-R_{1p}}{\eta_p}} + B_p \left( 1 - \frac{\omega_p \eta_p}{R_{2p}} \right) e^{\frac{-R_{2p}}{\eta_p}} + \omega_p \eta_p \rho_p E_p$$

where:

$\eta_p = \rho_p / \rho_0$	=	the relative density of the unreacted explosive
$E_p$	=	the specified internal energy per unit mass of the unreacted explosive
$A_p, B_p, \omega_p, R_{1p}, R_{2p}$		the input constants of the reaction product. The chemical reaction rate for conversion of un-reacted explosive to reaction products is described by the following reaction rate equation:

$$\frac{\partial F}{\partial t} = I(1 - F)^x (\eta_e - 1 - a)^r + G(1 - F)^x F^y (P)^z$$

here  $F$  denotes the burn fraction that is defined as the fraction of the explosive that has already reacted. For more details concerning the implementation of this equation of state, please refer to the Theory Manual

10. You can access the results of the un-reacted explosive and reaction products for IG elements. These EOSIG specific output variables are

Keyword	Description
SIE-E	Specific internal energy per unit mass of un-reacted explosive part
SIE-P	Specific internal energy per unit mass of reaction products part
FMAT	Volume fraction
RHO-E	Density of un-reacted explosive part
RHO-P	Density of reaction products part
MASS-E	Mass of un-reacted explosive part
MASS-P	Mass of reaction products part

The output variables for the burn fraction are

Keyword	Type of Elements	Description
FBURN	Solid Lagrangian Elements	Burn fraction of EOSIG material
	Euler Elements	Not applicable for EOSIG materials. Burn fraction for EOSJWL material
IGBURN	Solid Lagrangian Elements	Not available
	Euler Elements	Burn fraction of EOSIG MATERIAL

11. The ignition of IG material can be initiated by:
- a. Compression of the IG material in a small region, where the compression originates from outside that region. This is the most physical method to initiate ignition. Examples are a shock wave entering the region, a flow boundary that supplies mass to the region and a plate or other structural part that compresses the region. In all these cases the IG material should be initialized with zero pressures. This can be achieved by not specifying the specific energy on the TICVAL entry that prescribes the initial state of the IG material. The specific energy will be computed such that the initial pressure is zero.
  - b. Compression of the IG material in a small region where the compression originates within that region. This can be done by specifying either a density that exceeds the compression limit or a specific energy that gives rise to a sufficiently large pressure.

Table 9-15 Coefficients for the IG Model of Several Explosions in the Database.

Explosive	PBX-9404 (a)	TATB	PETN	Case TNT	LANL COMP B	Military COMP B	PBX-9404 (b)	LX-17
Unreacted Equation of State and Constitutive Values:								
RHO (g/cm <sup>3</sup> )	1.842	1.90	1.842	1.61	1.712	1.630	1.842	1.903
AE (Mbar)	69.69	108.2	37.42	17.98	778.1	1479.	9522.	778.1
BE (Mbar)	-1.727	-2.406	-1.313	-0.931	-0.05031	-0.05261	-0.5944	-0.05031
R1E	7.8	8.2	7.2	6.2	11.3	12.	14.1	11.3
R2E	3.9	4.1	3.6	3.1	1.13	1.2	1.41	1.13
OMGE	0.8578	1.251	1.173	0.8926	0.8938	0.9120	0.8867	0.8938
Reacted Product Equation of State Values:								
AP (Mbar)	8.524	6.5467	6.17	3.712	5.242	5.5748	8.524	6.5467
BP (Mbar)	0.1802	0.071236	0.16926	0.032306	0.07678	0.0783	0.1802	0.071236
R1P	4.6	4.45	4.4	4.15	4.2	4.5	4.6	4.45
R2P	1.3	1.2	1.2	0.95	1.1	1.2	1.3	1.2
OMGP	0.38	0.35	0.25	0.30	0.34	0.34	0.38	0.35
ECHEM (Mbar-cm <sup>3</sup> /g)	0.0554	0.0363	0.0548	0.0433	0.0496	0.04969	0.0554	0.03626
Reaction Rate Parameters:								
I (μ s <sup>-1</sup> )	44.0	50.0	20.0	50.0	44.0	44.0	44.0	50.0
G (Mbar-z μ s <sup>-1</sup> )	200.0	125.0	400.0	40.0	414.0	514.0	850.0	500.0
A	0.01	0.01	0.01	0.01	0.01	0.01	0.01	0.0
Z	1.6	2.0	1.4	1.2	2.0	2.0	2.0	3.0
X	2/9	2/9	2/9	2/9	2/9	2/9	2/9	2/9
Y	2/3	2/3	2/3	2/3	2/3	2/3	2/3	2/3
R	4.0	4.0	4.0	4.0	4.0	4.0	4.0	4.0

12. EID must unique among all EOSxx entries in one model.

## EOSJWL

## JWL Explosive Equation of State - SOL700

Defines the properties of a JWL equation of state commonly used to calculate the pressure  $p$  of the detonation products of high explosives

$$p = p_0 + A \left(1 - \frac{\omega\eta}{R_1}\right) e^{\frac{-R_1}{\eta}} + B \left(1 - \frac{\omega\eta}{R_2}\right) e^{\frac{-R_2}{\eta}} + \omega\eta\rho_0 e + \frac{\omega\rho\lambda Q}{\rho_0}$$

$$\frac{d\lambda}{dt} = a(1 - \lambda)^m p^n$$

$e$  = specific internal energy per unit mass

$\rho_0$  = reference density

$\rho$  = overall material density

$\eta$  =  $\rho/\rho_0$

$p_0$  = initial pressure

$\lambda$  = fraction that describes the afterburning. It ranges from 0 to 1

$A, B, R_1, R_2, a, m, n$  and  $Q$  are constants.

The last term:  $\frac{\omega\rho\lambda Q}{\rho_0}$  models afterburning.

**Format:**

1	2	3	4	5	6	7	8	9	10
EOSJWL	MID	A	B	R1	R2	OMEGA	P0	Q	
	a	m	N						

**Example:**

EOSJWL	37	5.2E11	0.77E11	4.1	1.1	0.34		0.0	
	0.0	0.0	0.0						

### Format and Example

```
EOSJWL  MID  A      B      R1    R2    OMEGA  P0    Q    +
EOSJWL  37   5.2E11  0.77E11  4.1   1.1   0.34      0.0  +

+      a      m      n
+      0.0    0.0    0.0
```

Describer	Meaning
EID	Unique equation of state number referenced from MATDEUL. (Integer > 0; Required)
A	Constant A. (Real, 0.0)
B	Constant B. (Real, 0.0)
R1	Constant R1. (Real, 0.0)
P2	Constant R2. (Real, 0.0)
OMEGA	Constant $\omega$ . (Real, 0.0)
P0	Initial pressure. See Remark 3. (Real, 0.0)
Q	Initial pressure. (Real, $\geq 0.0$ ). see Remark 6.
a	Initial pressure (Real, 0.0). see Remark 6.
m	Initial pressure (Real, 0.0). see Remark 6.
n	Initial pressure (Real, 0.0). see Remark 6.

### Remarks:

1. This equation of state can be used only with Eulerian elements.
2. A [DETSPL](#) entry must be used to specify the detonation model.
3. This equation of state is discussed in *Dytran Theory Manual*, Chapter 4: Models, [EOSJWL – JWL Equation of State](#).
4. In simulations with ideal gases, the JWL material needs to have an initial pressure to counter balance the pressure of the ideal gas. Similarly, in case of under calculations where an explosive is located at a certain depth, P0 should be preset to equal the initial hydrostatic pressure.
5. The [DETSPL](#) definition of one EOSJWL material also applies to all other EOSJWL materials. Therefore, the blast wave of one explosive can ignite other explosives. To prevent this linked ignition, [PARAM](#), [JWLDET](#), [NOLINK](#) can be used.
6. By default the constants  $a, m, n$  and  $Q$  are zero, and afterburning is not taken into account. To model afterburning all the variables  $Q, a, m$  and  $n$  have to be set to positive constants. The extent of the afterburning can be checked by requesting the Euler element variable AFTERBURN. This variable equals  $\lambda$ . If the afterburning in an Euler element is complete then AFTERBURN = 1.0.

**EOSMG****Mie-Gruneisen Equation of State - SOL 700 only**

Defines the properties of a Mie-Gruneisen equation of state commonly used to calculate the pressure  $p$  in high strain rate processes. Used in SOL 700 only.

$$p = \frac{\rho_0 c^2 n}{(1 - s\eta)^2} \left(1 - \frac{\Gamma_0 \eta}{2}\right) + \Gamma_0 \rho_0 e$$

$$\eta = 1 - \frac{\rho_0}{\rho_1}$$

$$\rho_1 = \min(\rho, \text{RM})$$

where

$e$  = specific internal energy per unit mass. For material at zero pressure,  $e$  has to be initialized as zero.

$\rho_0$  = reference density

$\rho$  = overall material density

$\Gamma_0$  = Gruneisen parameter at reference density.

$s$  = defined by  $U_s = c_0 + sU_p$ , where  $U_s$  and  $U_p$  are respectively the linear shock velocity and particle velocity as obtained from shock data.

$c$  = sound speed at reference density

RM = Cut-off value for density.

**Format:**

1	2	3	4	5	6	7	8	9	10
EOSMG	EID	c	S	R1	RM				

**Example:**

EOSMG	37	2000	1.5	2.0	3000				
-------	----	------	-----	-----	------	--	--	--	--

Descriptor	Meaning
EID	Unique equation of state number referenced from MATDEUL. (Integer > 0; Required)
c	Sound speed at reference density. (Real; Required)
s	Constant $s$ . (Real; Required)
$\Gamma_0$	Gruneisen gamma. (Real; Required)
RM	Cut off value for density. (Real; Required)

**Remarks:**

1. This equation of state can be used only with Eulerian elements.
2. This equation of state is discussed in Equations of State.
3. The cut off value RM is only used for limiting the pressure. To prevent division by zero RM should be less than  $s/s - 1 \rho_{ref}$ . RM can be set slightly below this value. In case the simulation gets instable because of too large pressures RM can be decreased.
4. EID must unique among all EOSxx entries in one model.



## EOSNA

## Noble-Abel equation of state - SOL700

Defines the properties of Noble-Abel equation of state where the pressure  $p$  is defined as:

$$p = (\gamma - 1) \frac{\rho}{1 - b\rho} e$$

where:

- $e$  = specific internal energy per unit mass
- $\rho$  = overall material density
- $\gamma$  = a constant
- $b$  = Gas co-volume

### Format and Example

1	2	3	4	5	6	7	8	9	10
EOSNA	EID	GAMMA	B	R	CV	CP			
EOSNA	35	1.2363	0.001	314.46					

Field	Contents	Type	Default
EID	Unique equation of state number.	$I > 0$	Required
GAMMA	Constant $\gamma$ .	$R \geq 0.0$	Required
B	Gas co-volume .	$R \geq 0.0$	0.0 Refer to Remark 3.
R	Gas constant.	$R > 0.0$	Refer to remark 4.
CV	Specific heat at constant volume.	$R > 0.0$	Refer to remark 4.
CP	Specific heat at constant pressure	$R > 0.0$	Refer to remark 4.

## Remarks

1. Refer to *Dytran Theory Manual*: Chapter 4-Models, for EOSNA – Noble-Abel Equation of State.
2. EOSNA – Noble-Abel Equation of State is used with Eulerian elements only.
3. The parameter B models the interactions between gas particles.
4. The temperature of the gas is calculated when one of the gas constants,  $R$ ,  $C_v$ , and  $C_p$  is mentioned.
5. The Euler variable name for temperature is TEMPTURE.
6. The relation between Gamma,  $R$ ,  $C_v$ , and  $C_p$  is shown below:

$$\gamma = \frac{C_p}{C_v} \quad R = C_p - C_v$$

**EOSPOL****Polynomial Equation of State for Solids - SOL 700 only**

Defines the properties of a polynomial equation of state where the pressure  $p$  is defined in SOL 700 as follows:

In compression ( $\mu > 0$ ),

$$p = a_1\mu + a_2\mu^2 + a_3\mu^3 + (b_0 + b_1 + b_2\mu^2 + b_3\mu^3)\rho_0 e$$

In tension ( $i < 0$ ),

$$p = a_1\mu + (b_0 + b_1\mu)\rho_0 e$$

Where

$$\mu = \eta - 1$$

$$\eta = \rho/\rho_0$$

$p$  = overall material density

$p_0$  = reference density

$e$  = specific internal energy per unit mass

Used in SOL 700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
EOSPOL	EID	A1	A2	A3	B0	B1	B2	B3	
	HVL	VISC							

**Example:**

EOSPOL	100	80.E6							
	1.1								

Describer	Meaning
EID	Unique equation of state member. (Integer > 0; Required)
A1	Coefficient a1 or Bulk Modulus. (Real; Default = 0.0)
A2	Coefficient a2. (Real; Default = 0.0)
A3	Coefficient a3. (Real; Default = 0.0)
B0	Coefficient b0. (Real; Default = 0.0)
B1	Coefficient b1. (Real; Default = 0.0)

Describer	Meaning
B2	Coefficient b2. (Real; Default = 0.0)
B3	Coefficient b3. (Real; Default = 0.0)
HVL	Hydrodynamic volume limit. (Real > 1.0; Default = 1.1)
VISC	Viscosity coefficient.(Real; Default = 0.0)

**Remarks:**

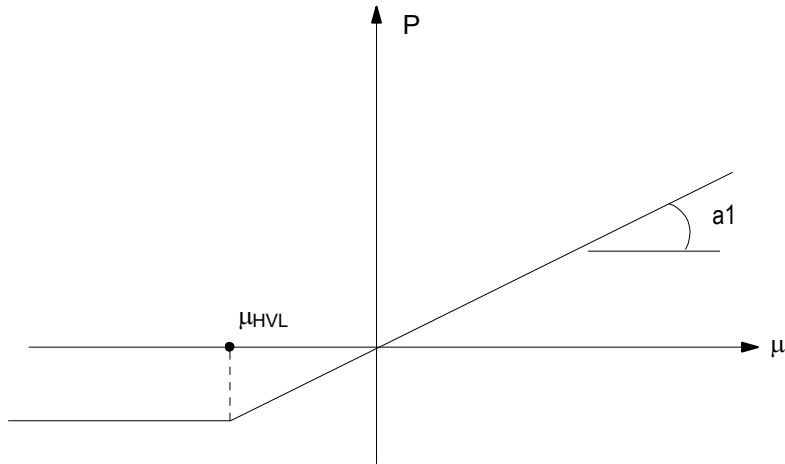
1. When the relative volume ( $\rho_0/\rho$ ) exceeds HVL, the pressure is cut off to

$$P_{HVL} = f(\mu_{HVL})$$

with

$$\mu_{HVL} = \frac{1}{HVL} - 1$$

e.g., for  $p = a1 \cdot \mu$ , the pressure behavior is as follows:



2. When the PARAM,HVLFail is set to YES, the elements where the relative volume ( $\rho_0/\rho$ ) exceeds HVL fail completely. Their stress state is zero.

**EOSUDS**

## User-defined Equation of State - SOL 700 only

Defines the frequency dependent properties for an isotropic poroelastic material. Use in SOL700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
EOSUDS	EID	GROUP	UNAME	VISC					

**Example:**

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE water 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

1	2	3	4	5	6	7	8	9	10
EOSUDS	12	WATER	EXEOS	.01					

Describer	Meaning
EID	Unique equation of state number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXEOS)
VISC	Viscosity coefficient $R > 0$ No viscosity. See Remarks 4.and 5.

**Remarks:**

1. The EID must be referenced by a MATDEUL or MAT1 entry.
2. The equation of state name is passed to method usrEOS of the SCAIMDSolver700 interface and can be used to identify the equation of state.
3. The viscosity coefficient is the dynamic viscosity. It is the ratio between shear stress and velocity gradient.
4. For the single mat solver viscous stresses will be stored in the output files by the use of TXX through TZX. Also, EFFSTS is available. For the multi-material solver viscous stresses are stored in TXX-VIS, TYY-VIS, TZZ-VIS, TXY-VIS, TYZ-VIS, TZX-VIS. These viscous stresses only depend on the current velocity gradients. The stresses like TXX are elastic-plastic stresses and depend on past stresses. The total stress tensor in the element is given by the average of the viscous stress and elastic-plastic stress. The weight factors are the material fraction of viscous fluid and the remaining materials.
5. UNAME can be:

Subroutine Name	Function
EXEOS	Standard user defined Equation of State

**EPOINT**

## Extra Point List

Defines extra points for use in dynamics problems.

**Format:**

1	2	3	4	5	6	7	8	9	10
EPOINT	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	

**Example:**

EPOINT	3	18	1	4	16	2			
--------	---	----	---	---	----	---	--	--	--

**Alternate Format and Example:**

EPOINT	ID1	“THRU”	ID2						
EPOINT	17	THRU	43						

Describer	Meaning
IDI	Extra point identification number. (100,000,000 > Integer > 0; for “THRU” option, ID1 < ID2).

**Remarks:**

1. All extra point identification numbers must be unique with respect to all other structural, scalar, and fluid points for direct methods of solution. For modal methods, they must be larger than the number of eigenvectors retained for analysis.
2. EPOINT is used to define coordinates used in transfer function definitions (see the TF and DMIG entries).
3. If the alternate format is used, extra points ID1 through ID2 are also defined to be extra points.
4. See the *MSC Nastran Dynamic Analysis User's Guide* for a discussion of extra points.

**ERPPNL****Equivalent Radiated Power Definition**

Defines one or more panels by referencing sets of elements or properties.

**Format:**

1	2	3	4	5	6	7	8	9	10
ERPPNL	NAME1	SETID1	NAME2	SETID2	NAME3	SETID3	NAME4	SETID4	
	NAME5	SETID5							

**Example:**

ERPPNL	ROOF	1	DOORLF	16					
--------	------	---	--------	----	--	--	--	--	--

Describer	Meaning
NAMEi	Panel label. (CHAR)
SETIDi	Identification number of a SET3 Bulk Data entry that lists the panel property entries or the panel elements. (Integer > 0)

**Remarks:**

1. The SET3 entries can only refer to 2D and/or 3D structural element types, such as QUAD4, TRIA3, QUADR, CTRIAR, QUAD8, TRIA6, HEXA, PENTA, PYRAM and TETRA, or PSHELL, PCOMP, PCOMPG and PSOLID entry associated with those element types.
2. NAMEi are used in a Case Control SET definition defining *setp* to select the panels in the Case Control command ERP.
3. Duplicate Element IDs are removed if they occur on the referencing SET3 entries prior to the ERP calculation.
4. User does not need to specify the outer face(s) of 3D elements. The outer face(s) or exposed face(s) of 3D elements will be determined automatically. Interior 3D element(s) which has no exposed face(s) will not be included for ERP computation.

## EULFOR

## Body Force Loading on Euler Elements - SOL 700 only

Defines a body force loading (acceleration) on Euler elements per unit mass. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
EULFOR	BID	CID	VALUE	TID	N1	N2	N3		

### Example:

EULFOR	100	5		13	1.	0.	0.		
--------	-----	---	--	----	----	----	----	--	--

Describer	Meaning
BID	Unique body force number. (Integer > 0; Required)
CID	ID of a coordinate system. (Integer ≥ 0; Default = 0)
VALUE	Scale factor for the vector. See Remark 1. (Real ≥ 0.0)
TID	TABLED1 ID describing the scale factor for the load as function of time. See Remark 1. (Integer)
N1, N2, N3	Components of a vector giving the load (acceleration) direction defined in coordinate system CID. At least one must be nonzero. See Remark 2. (Real; Default = 0.0)

### Remarks:

1. Either VALUE or TID must be nonzero.
2. By default the components are zero, but at least one of them should be nonzero.



**EULFOR1****Body force for Eulerian Regions - SOL700**

Alternative way to define an acceleration within a geometric region of the Euler model regions are defined by geometric shapes which are defined by EULFREG entries.

**Format:**

EULFOR1	SID	ESID							
---------	-----	------	--	--	--	--	--	--	--

**Example:**

EULFOR1	300	1							
---------	-----	---	--	--	--	--	--	--	--

Describer	Meaning
SID	Unique EULFOR1 number referenced from a PEULER1 entry .(Integer >0, required)
ESID	Group of geometric region EULFREG ID. (Integer > 0; Required)

**Remarks:**

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.
4. To increase the accuracy of the region definition parameter MICRO can be used.

## EULFREG

## Body force for Eulerian Regions - SOL700

Defines the acceleration field for sets of Eulerian regions, The Eulerian regions are defined by geometric shapes. For each coordinate direction a time-dependend acceleration can be defined.

### Format:

EULFREG	ERID	ESID	TYPE1	VID1	MID1	ACCX	ACCY	ACCZ	
	LEVEL								

### Example:

EULFREG	300	1	BOX	400	100	100	200	300	
	0.0								

Describer	Meaning
ERID	Unique ERID number. (Integer > 0; Required)
ESID	ID of group of Euler regions referenced from the EULFOR1 entry. (Integer > 0; Required)
TYPEi	The type of Eulerian region. (Character; Required)
	SURF            Region inside or outside a multifaceted surface.
	SPHERE        Region inside a sphere.
	CYLINDER     Region inside a cylinder.
	BOX            Region inside a box.
	ELEM           Region defined by element list.
VIDi	Number of a geometric entity, a SET1 number, or number of a SURFINI entry. (Integer > 0; Required)
MIDi	Number of a MATDEUL entry to which the acceleration field will be applied. (Integer > 0; Required)
ACCi	Unique table number that defines the variation of acceleration in time. ACCX, ACCY and ACCZ respectively denote the acceleration in the x-, y- and z-direction. (Integer > 0; Required)
LEVELi	Level indicator for this material and initial values. (Real; default=0.0)

### Remarks:

1. EULFOR1 and EULFREG are only available for the multi-material Euler solver.
2. The combination of Eulerian region and material ID determines where the acceleration field is applied.
3. It is allowed to cover only part of the Euler domain with EULFOR1 definitions.

4. All level indicators  $LEVEL_i$  must have different values. The level indicator can be negative.
5. To increase the accuracy of the region definition parameter  $MICRO$  can be used.

## EXCLUDE

Ignore Bulk Data entries from primary to secondary Module

Specifies Bulk Data entries in the primary Module to be ignored in the secondary (or copied) Module.

### Format:

1	2	3	4	5	6	7	8	9	10
EXCLUDE	ENTRY1	ENTRY2	ENTRY3	-etc.,					

### Example:

EXCLUDE	SPC	FORCE							
---------	-----	-------	--	--	--	--	--	--	--

Describer	Meaning
ENTRYi	Name of a Bulk Data entry to be ignored.

### Remarks:

1. A secondary Module may have its own BEGIN MODULE section but it is not required as long as the user is willing to accept all modeling data from the reference Module because, by default, all entries in the reference Module will be copied to the secondary Module. However, if the user wishes to exclude and/or replace some of the entries from the reference Module in the copy then he simply specifies at least one instance of the entry to be ignored. For example, if the user wants to ignore all FORCE entries from the reference Module then specify a single FORCE entry in the secondary Module's Bulk Data section. Alternatively, the user may specify a list of Bulk Data entries to ignore on the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
2. EXCLUDE must be specified in the secondary (or copied) Module's Bulk Data section.
3. GRID entries cannot be ignored or replaced and EXCLUDE,GRID will cause a fatal error.
4. See [MDBULK](#) for the definition of secondary (or copied) Modules.

## EXTRN Partitioned External Superelement Connection

Defines a boundary connection for an external superelement.

### Format:

1	2	3	4	5	6	7	8	9	10
EXTRN	GID1	C1	GID2	C2	GID3	C3	GID4	C4	
	-etc.-		GID6	"THRU"	GID7	C6	-etc.-		

### Example:

EXTRN	1001	123	1120	123456	1201	123			
-------	------	-----	------	--------	------	-----	--	--	--

Describer	Meaning
GID <sub>i</sub>	Grid identification number to which the exterior superelement matrices will be connected.
C <sub>i</sub>	Component numbers. (Integer 0, blank, or 1 for scalar points; Integers 1 through 6 with no embedded blanks for grids.)

### Remarks:

1. EXTRN can only be specified in partitioned Bulk Data Sections and is ignored in the main Bulk Data Section.
2. Connection grids must be specified in the partitioned Bulk Data Section following BEGIN SUPER = SEID.
3. "THRU" may be specified only in fields 3, 5, or 7.
4. Pairs of blank fields may be entered to allow easier modification of the EXTRN entry.
5. The order of the GID<sub>i</sub> and C<sub>i</sub> pairs must be in the internal order of the grids set in the creation run. But since Nastran no longer re-sequences grids then the pairs are specified in ascending GRID ID order. However, if PARAM,OLDSEQ or the SEQGP entry is specified then the proper order may not be ascending. To determine the internal grid order, add PARAM,USETPRT,0 and PARAM,USETSTR1,A.

## Entries F - L

### FAILJC

Johnson-Cook Failure Model - SOL 700 only

Defines the properties of the Johnson-Cook failure model. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILJC	FID	D1	D2	D3	D4	D5	.0 $\epsilon_{pl}$	TROOM	
	TMELT	CP	MTH						

Example:

FAILJC	1	.05	3.44	-2.12	0.002	0.16	1.0	297.0	
	1495	450	CONT						

Descriptor	Meaning
FID	Unique failure model number. Referenced from MATDEUL. (Integer > 0; Required)
D1...D5	Parameters. See Remark 4. (Real; Default = 0.0)
.0 $\epsilon_{pl}$	Reference plastic strain rate. (Real; Default = 1.0)
TROOM	Room temperature. (Real; 0.0)
TMELT	Melt temperature. (Real; 1.E+20)
CP	Heat capacity. (Real; 1.E+20)
MTH	Specifies how failure is applied. (Character; CONT)
	CONT    Continuous failure
	DISC    Discrete failure
	NOFAIL    Damage is not used for failure

Remarks:

1. This failure model is only available for Eulerian materials using the multi-material solver with strength.
2. The use of coupling surfaces is not supported.
3. The variable D can be visualized by adding DAMAGE to the Output request for Euler elements.
4. Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{time} \frac{\Delta \epsilon_p}{\epsilon^{frac}}$$

$$\epsilon^{frac} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left( 1 + D_4 \ln \frac{\dot{\epsilon}_{pl}}{\dot{\epsilon}_{pl}^0} \right) (1 + D_5 T^*)$$

$$\sigma^* = \frac{\sigma_m}{\sigma}$$

$$T^* = \frac{T - T_{room}}{T_{melt} - T_{room}}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature,  $\sigma_m$  is the mean stress,  $\sigma$  is the effective stress and  $\epsilon^{frac}$  is the fracture strain. The fracture strain depends on a non-dimensional plastic strain rate  $\dot{\epsilon}_{pl} / \dot{\epsilon}_{pl}^0$ . If D exceeds one it set equal to one.

The damage variable D is transported along with the Eulerian material.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

## FAILMPS

## Maximum Plastic Strain Failure Model - SOL 700 only

Defines the properties of a failure model where failure occurs when the equivalent plastic strain exceeds the specified value. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILMPS	FID	MPS							

Example:

FAILMPS	1	.15							
---------	---	-----	--	--	--	--	--	--	--

Describer	Meaning
FID	Unique failure model number referenced from MATDEUL. (Integer > 0; Required)
MPS	Maximum plastic strain that causes failure. (Real; Required)



## FAILUDS

## User Defined Failure For Nonlinear Explicit Analysis - SOL 700 only

User defined simple failure of Eulerian materials. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FAILUDS	FID	GROUP	UNAME						

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE usrfail 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

1	2	3	4	5	6	7	8	9	10
FAILUDS	200	usrfail	EXFAIL						

Describer	Meaning
FID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXFAIL)

Remarks:

1. Only can be used for SOL 700.
2. FID has to be referenced by a MATDEUL, MAT1 or MATORT entry.
3. UNAME can be:

Subroutine Name	Function
EXFAIL	Standard user defined failure
EXFAIL1	Alternative 1 user defined failure
EXFAIL2	Alternative 2 user defined failure

4. For option UNAME=EXFAIL2, for each material and for each Euler element a variable will be created that monitors the degree of failure of the material. This variable is denoted by DAMAGE and is between 0 and 1. The EXFAIL2 routine allows updating this damage variable due to the plastic strain increment of the current cycle.
5. For option UNAME=EXFAIL2 there are three ways in which this damage variable can model failure. These are:

- Continuous failure: The yield stress is reduced by a factor  $(1-D)$ . When  $D$  exceeds 1.0, the yield stress equals zero and the element fails.
  - Discrete failure: the element fails when  $D$  equals one.
  - No failure: positive damage values will not lead to failure. This is useful if the failure modeling is done by an YLDUDS routine. Then the yield stress can be reduced depending on the magnitude of the damage variable.
6. `UNAME=EXFAIL2` is only supported by the multi-material Euler solver with strength.
  7. `UNAME=EXFAIL1` is only supported with `MATORT`.
  8. For Lagrangian materials, `UNAME=EXFAIL` or `EXFAIL1` are only available.

## FBADLAY      Dynamic Load Time Delay for FRF Based Assembly (FBA)

Defines the time delay term  $\tau$  in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBADLAY	SID	COMPID/ COMPNAME	PNTID	C	DELAY				

Examples:

FBADLAY	15	BODY	10	3	0.5				
---------	----	------	----	---	-----	--	--	--	--

FBADLAY	25	30	5	2	0.1				
---------	----	----	---	---	-----	--	--	--	--

Describer	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
DELAY	Time delay term $\tau$ . See Remark 4. (Real)

Remarks:

- SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
- This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
- The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
- Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the time delay term  $\tau$  in frequency response analysis.
- All FBADLAY entries specified in an FBA process are automatically converted internally by the program to equivalent DELAY entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

## FBALOAD

## Load Scale Factor Specification for FRF Based Assembly (FBA) Process

Defines the scale factor for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBALOAD	SID	COMPID/ COMPNAME	PNTID	C	A				

Examples:

FBALOAD	10	WING	20	3	2.5				
---------	----	------	----	---	-----	--	--	--	--

FBALOAD	20	30	25	1	1.5				
---------	----	----	----	---	-----	--	--	--	--

Describer	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Optional name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed).
PNTID	Grid or scalar point identification number. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integer 1 through 6 for grid point; blank or 0 for scalar point)
A	Scale factor. See Remark 4. (Real)

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the scale factor A in frequency response analysis.
5. All FBALOAD entries specified in an FBA process are automatically converted internally by the program to equivalent DAREA entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

**FBAPHAS**

## Dynamic Load Phase Lead for FRF Based Assembly (FBA)

Defines the phase lead term  $\theta$  in the equations of motion of the dynamic loading function for frequency dependent loads in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FBAPHAS	SID	COMPID/ COMPNAME	PNTID	C	PHASE				

Examples:

FBAPHAS	25	FRAME	40	1	10.0				
FBAPHAS	30	50	10	2	5.0				

Descriptor	Meaning
SID	Identification number. See Remark 1. (Integer > 0)
COMPID	Identification number of the FRF component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the FRF component whose FRFs have been generated in a previous Nastran execution. (Up to 8 characters; no blank allowed)
PNTID <sub>i</sub>	Grid or scalar point identification numbers. See Remark 3. (Integer > 0)
C	Component number. See Remark 3. (Integers 1 through 6 for grid points; blank or 0 for scalar points.)
PHASE	Phase lead term $\theta$ . See Remark 4. (Real)

Remarks:

1. SID is referenced by RLOAD1, RLOAD2 and ACSRCE entries.
2. This entry is ignored if the specified COMPID/COMPNAME is not part of the FBA process. A user warning message is issued in this case.
3. The component C of the point PNTID specified in this entry must be among the excitation degrees of freedom of the FBA process. If not, the program terminates the job with a user fatal message.
4. Refer to RLOAD1, RLOAD2 and ACSRCE entries for the formulas that define the phase lead term  $\theta$  in frequency response analysis.
5. All FBAPHAS entries specified in an FBA process are automatically converted internally by the program to equivalent DPHASE entries by replacing the grid/scalar point IDs referenced in these entries by equivalent internal point IDs.

## FBODYLD      Equilibrated Free-Body Applied Load Case Definition

Defines an equilibrated free-body applied load case.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYLD	NAMEL	FBODYSB							
LABEL									

Example:

FBODYLD	WINGLD	WINGSB							
LOAD ON THE RIGHT WING									

Describer	Meaning
NAMEL	User defined name identifying the load case. (Character; Required)
FBODYSB	Name of a FBODYSB Bulk Data entry that defines the subsystem for this load. (Character; Required)
LABEL	A string comprising no more than 64 characters (fields 2 through 9) that identifies and labels the load case. (Character; optional)

Remarks:

1. NAMEL must be unique.
2. The Label is optional.

## FBODYSB

## Equilibrated Free-Body Subsystems Definition

Defines an equilibrated free-body subsystem.

Format:

1	2	3	4	5	6	7	8	9	10
FBODYSB	NAMES	GRIDSET	ELEMSET	XFLAG					
LABEL									

Example:

FBODYSB	WING	1	1	ADM					
RIGHT WING									

Descriptor	Meaning
NAMES	User defined name identifying the submodel. (Character; Required)
GRIDSET	Identification number of a SET1 entry that has a list of Grid Point Force grids to include in defining the subsystem. (Integer > 0)
ELEMSET	Identification number of a SET1 entry that has a list of elements to include in the system (Integer > 0 or blank)
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types. Default            Blank (no type excluded) S                    SPC forces M                    MPC forces A, L, or P        Applied loads D                    DMIG's (and any other type not described above)
Label	An optional string of up to 64 characters (fields 2 through 9) that identifies the subsystem.

Remarks:

1. Only those Grid Point Forces which have both an included grid point and element (or Grid Point Force type) will be taken into account.
2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
3. Fictitious grids or elements do not produce error or warning messages.
4. The XFLAG data can be any combination of the letters S,M,A,L,P and D (e.g., MAD).
5. The continuation is optional.

## FFCONTR

## Closed Volume Intended for Fluid Filled Containers

Defines the pressure within a closed volume. Intended for the use in (partially) filled containers, where dynamic fluid effects are negligible, e.g. top loading and hot filling. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FFCONTR	FFID	SID	FVOL	PATM	TEMPTAB	DENSTAB	TACTIVE		

Example:

FFCONTR	1	2	1.50E-03	0.1E6	10	20			
---------	---	---	----------	-------	----	----	--	--	--

Describer	Meaning
FFID	Unique FFCONTR identification number. (Integer > 0; Required)
SID	Number of a BSURE, BCBOX, BCPROP, BCMATL or BCSEG entry defining the closed surface. (Integer > 0; Required)
FVOL	Fluid volume in the container. (Real ≥ 0.0; Required)
PATM	Atmospheric pressure. Used for determination of the constant $C$ for $p \cdot V = C$ . (Real > 0.0; Required)
TEMPTAB	A reference to a TABLED1 ID that specifies how temperature of the container changes in time. (Integer ≥ 0; Blank)
DENSTAB	A reference to a TABLED1 ID that specifies how density of the container changes with temperature. (Integer ≥ 0; Blank)
TACTIVE	Time at which the pressure computation of the bottle is started. Until this time the pressure inside the bottle equals the ambient pressure. The volume of the bottle at TACTIVE will be used for the initial pressure computation of the gas in the bottle.

Remarks:

- If TEMPTAB is not set, the gas above the fluid is assumed to be an ideal, iso-thermal gas:  
 $p \cdot V = C$ , where  $C$  is a constant. If TEMPTAB is set, the temperature is applied to both the fluid as well as the gas. Then the gas satisfies  $p \cdot V/T = C$ , where  $T$  is the temperature of the fluid.
- The fluid is assumed incompressible.
- The pressure is based on the uniform pressure gasbag algorithm, where the pressure is uniform in the volume, but variable in time.
- Output for the fluid-filled container is available through a SURFOUT definition. The available variables are: PRESSURE, VOLUME, TEMPTURE, VOLGAS, VOLFLUID, GAUGEPRES and RHOFLUID.
- The normals of the surface referenced by SID are reversed automatically if required.



6. Modeling guidelines are described in the “Getting Started” Section.
7. If DENSTAB is set then volume of the fluid changes according to

$$V^{Fluid} = \frac{\rho(T_0)V_0^{Fluid}}{\rho(T)}$$

Here  $T_0$  and  $V_0^{Fluid}$  are initial values for temperature and fluid volume,  $\rho$  is the fluid density and  $T$  denotes the current temperature. If TEMPTAB is not set the DENSTAB entry will not be used.

8. At time=TACTIVE the gas is assumed to be in contact with the ambient pressure for the last time. This means that at Time = TACTIVE the pressure in the bottle equals the ambient pressure. After TACTIVE the bottle has been closed and there is no longer contact between ambient and gas inside the bottle. Any change in volume of the bottle or temperature or fluid will result in change of pressure of the gas inside the bottle.

## FLFACT Aerodynamic Physical Data

Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLFACT	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Example:

FLFACT	97	.3	.7	3.5					
--------	----	----	----	-----	--	--	--	--	--

Alternate Format and Example:

FLFACT	SID	F1	"THRU"	FNF	NF	FMID			
FLFACT	201	.200	THRU	.100	11	.133333			

Describer	Meaning
SID	Set identification number. (Unique Integer > 0)
F <sub>i</sub>	Aerodynamic factor. (Real)
FNF	Final aerodynamic factor. (Real)
NF	Number of aerodynamic factors. (Integer > 0)
FMID	Intermediate aerodynamic factors. See Remark 4. (Real)

Remarks:

1. Only the factors selected by a FLUTTER entry will be used.
2. Embedded blank fields are not allowed in the first format above.
3. The factors must be specified in the order in which they are to be used within the looping of flutter analysis.
4. *FMID* must lie between *F1* and *FNF*; otherwise, *FMID* will be set to  $(F1 + FNF)/2$ .

Then

$$F_i = \frac{F1(FNF - FMID)(NF - i) + FNF(FMID - F1)(i-1)}{(FNF - FMID)(NF - i) + (FMID - F1)(i-1)}$$

where  $i = 1, 2, \dots, NF$

The use of FMID (middle factor selection) allows unequal spacing of the factors.

$$FMID = \frac{2 \cdot F1 \cdot FNF}{F1 + FNF} \text{ gives equal values to increments of the reciprocal}$$

of  $F_i$ .

5. If method = PK and this entry specifies velocities, then the velocities must be non-zero. Input of negative values produces eigenvector results at a velocity equal to the positive value of the input. Input of positive values provide eigenvalues results without eigenvectors.

## FLOW

## Flow Boundary Condition

Defines the properties of a material for the boundaries of an Eulerian mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOW	LID	BCID	MESH	DIR					
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			
	TYPE1	VALUE1	TYPE2	VALUE2	TYPE3	VALUE3	TYPE4	VALUE4	
	TYPE5	VALUE5							

Example:

FLOW	120	122							
	XVEL	100.0							

Describer	Meaning
LID	Number of a set of flow boundary conditions. (Integer > 0; Required)
BCID	Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 5. (Integer ≥ 0)
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 6. (Integer ≥ 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 7. (Character)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 8. (Real)
TYPEi	The flow boundary property being defined. (Character)
	Material                      The MATDEUL ID number.
	XVEL                            The material velocity in the x-direction.
	YVEL                            The material velocity in the y-direction.
	ZVEL                            The material velocity in the z-direction.
	PRESSURE                    The pressure of the material at the boundary.
	DENSITY                      The density of the material at inflow.
	SIE                              The specific internal energy at inflow
	FLOW                          The type of flow boundary required.

Describer	Meaning
VALUEi	<p>HYDSTAT                    A Hydrostatic pressure profile using a HYDSTAT entry.</p> <p>The value for the property defined. (Real; Integer or Character; Required)</p> <p>For TYPEi set to FLOW, the value is a character entry being either IN, OUT, BOTH or SYM defining that the flow boundary is defined as an inflow, outflow, possibly an in- or outflow or symmetry boundary. The default is BOTH.</p> <p>VALUEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary.</p> <p>For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used.</p>

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. Any material properties not specifically defined have the same value as the element with the flow boundary condition.
3. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
4. In the case of material flow into a multi-material Euler mesh, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively -1E+20 and 1E+20 for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
9. Prescribing both pressure and velocity may lead to the instabilities.
10. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

## FLOWC

## Cyclic Flow Boundary Condition

Defines the properties of a material for the boundaries of a Eulerian mesh.

Inflow values can be taken from another boundary condition. This allows cyclic or periodic boundary conditions. Likewise, the outflow of material goes into the other boundary condition.

FLOWC entries have to be defined in pairs. The FID on one entry has to be equal to FID2 of the other entry. For example

```
FLOWC, 10, 30, 20
```

```
FLOWC, 20, 40, 10
```

For FLOWC boundaries, mass flow summaries can be created as time history. Used in SOL700 only

Format:

1	2	3	4	5	6	7	8	9	10
FLOWC	FID	BCID	TYPE	MESH	DIR	FID2			+
+	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			+
+	METHOD	TID							

Example:

FLOWC	1	122	2						
+									
+	2	1							

Field	Contents
FID	Unique number of a FLOWC entry. (Integer > 0; Required)
BCID	Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer > 0)

Field	Contents
TYPE	<p>Flow boundary types. See Remarks 2.and 3. (Character; Required)</p> <p>IN: Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sic.</p> <p>OUT: Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</p> <p>BOTH: Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</p>
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer > 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character; no default)
FID2	Referenced FLOWC id from which inflow and outflow values will be taken from. (Integer > 0; Required)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real)
METHOD	<p>Method describing how material properties and pressure are mapped going from one boundary to the other. (Integer &gt; 0; Default=1)</p> <ol style="list-style-type: none"> <li>1. Map both velocity, material flow properties, and pressure loads one-to-one.</li> <li>2. Map velocity, material flow one-to-one. Pressure is given by element pressures.</li> <li>3. Inflow is taken as the average of outflow properties. This applies to both velocity and material properties. Pressure is given by element pressures.</li> </ol>
TID	TABLED1 ID that specifies a time dependent scale factor by which the amount of inflow is multiplied. This will not conserve total mass. (Integer > 0. Default = not used).

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.
3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.

5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively  $-1E+20$  and  $1E+20$  for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. FLOWC can be used to specify flow boundaries for CHEXA's and also for Euler element created by the MESH,BOX option.
8. FLOWC is only supported by the multi-material Euler solver.
9. To enable using the outflow values of FID2 as the inflow values for FID, the boundary faces of FID and FID2 are mapped onto each other. For this mapping, only translations and rotations around coordinate axes are permitted.
10. Consider a cubic Euler mesh and that material flows from the left to the right. At the right side, the boundary condition is imposed

FLOWC,4,30,,5

and at the left side

FLOWC,5,40,,4

These definitions cause all material that flows out of the right side boundary into the left side boundary. Moreover, the Euler element pressures on the right side are put on the Euler elements of the left boundary condition.

In practice, it may be useful to skip the coupling between the two boundaries with regard to pressure. This can be done by setting METHOD equal to 2.

With these definitions, material from boundary 4 still flows into boundary 5 but pressure boundaries are transmitting.

11. TID is useful if several objects have identical outflow that is used as inflow by another object. Then only one object has to be modeled. To account for the other objects when defining inflow, the scale factor can be used. It can also be used to turn off in and outflow. When TID is set, either METHOD=2 or METHOD=3 are recommended and METHOD=1 should not be used.
12. It is allowed that the definition of the FLOWC entry overlaps with FLOW definitions. In that case, the FLOWC definition overrules the other ones



## FLOWDEF

## Default Flow Boundary

Definition of default Eulerian flow boundary condition. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWDEF	FID		TYPEM						
	TYPE1	VALUE1	TYPE2	-etc.-					

Example:

FLOWDEF	25		HYDRO						
	DENSITY	1000							

Describer	Meaning
FID	Unique FLOWDEF number. (Integer > 0; Required)
TYPEM	HYDRO, STRENGTH, MMHYDRO, or MMSTREN. (Character, HYDRO)
TYPEi	The flow boundary property being defined. (Character)
Material	The MATDEUL ID number.
XVEL	The material velocity in the x-direction.
YVEL	The material velocity in the y-direction.
ZVEL	The material velocity in the z-direction.
PRESSURE	The pressure of the material at the boundary.
DENSITY	The density of the material at inflow.
SIE	The specific internal energy at inflow
FLOW	The type of flow boundary required.
HYDSTAT	A Hydrostatic pressure profile using a HYDSTAT entry.
VALUEi	The value for the property defined. (Real; Integer or Character; Required)
	For TYPEi set to flow the value is a character entry being either IN, OUT, or BOTH, defining that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH.
	VALUEi is required data only if one or more of the TYPEi entries are defined. The TYPEi entries are not required. Thus, a flow boundary by default allows for in- or outflow of the material adjacent to the boundary.
	For TYPE = HYDSTAT, the value is an integer entry denoting the HYDSTAT entry to be used.

Remark:

1. If this entry is not specified, a default wall boundary condition is applied to all Eulerian free faces.
2. For TYPE = HYDSTAT, the pressure is set using HYDSTAT, the velocity equals the element velocity. In case of inflow the density follows from the hydrostatic pressure by using the equation of state.

## FLOWT

## Time Dependent Flow Boundary

Defines the material properties for the in- or outflow of material through the boundary of an Euler mesh. Inflow velocity and material properties can be chosen time dependent. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWT	FID	BCID	TYPE	MESH	DIR				
	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			
	VELTYPE	VELOCITY	PRESTYP	PRES					
	MID	DENSTYP	DENSITY	SIETYPE	SIE				

Example:

FLOWT	2	122	IN						
	TABLE	101	TABLE	102					
	91	TABLE	104	TABLE	107				

Describer	Meaning
FID	Unique number of a FLOWT entry. (Integer > 0; Required)
BCID	Number of a set of segments specified by the BCSEG entries where the flow boundary is located. See Remark 3. (Integer ≥ 0)
TYPE	<p>IN                    Inflow boundary. See Remarks 2.and 3. (Character; Required) Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.</p> <p>OUT                    Only outflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use material mixture as present in the adjacent Euler element.</p> <p>BOTH                    Material is allowed to flow in or out. In or outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be optionally defined. If not given the pressure in the adjacent Euler element will be taken.</p>
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 4. (Integer ≥ 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 5. (Character)

Describer	Meaning
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 6. (Real)
VELTYPE	Type of velocity definition. (Character, Element) ELEMENT Value of Euler Element CONSTANT Value is constant in time TABLE Value varies in time
VELOCITY	Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real)
PRESTYP	Type of pressure definition (Character, Element) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time
PRES	Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1 ID. (Integer or Real)
MID	MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and sie for the inflowing material. (Integer)
DESTYP	Type of density definition. (Character, Default is required when MID is given.) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time
DENSITY	Value of density. If DENSTYP = TABLE it refers to a TABLED1 ID. (Character; Default is required when MID is given.)
SIETYPE	Type of density definition. (Character; Default is required when MID is given.) ELEMENT Value of Euler element CONSTANT Value is constant in time TABLE Value varies in time.
SIE	Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1 ID. (Integer or Real; Default is required when MID is given.)

## Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. TLOAD entries referencing FLOW entries must have the TID field blank or zero.

3. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
4. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
5. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
6. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively  $-1E+20$  and  $1E+20$  for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the Min value.
7. Any material properties not specifically defined have the same value as the element that with the boundary conditions.
8. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
9. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be instable. However, for supersonic inflow one can specify both pressure and velocities there are no outgoing waves at a supersonic inflow boundary.

## FLOWUDS

## User Defined Flow Boundary.

Defines a flow boundary on an Eulerian mesh specified by a user subroutine. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FLOWUDS	LID	GROUP	UNAME	BCID	MESH	DIR			
+	XMIN	XMAX	YMIN	YMAX	ZMIN	ZMAX			

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myflow 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

*In Bulk Data:*

1	2	3	4	5	6	7	8	9	10
FLOWUDS	12	MYFLOW	EXFLOW	300					

Descriptor	Meaning
LID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXFLOW)
BCID	Number of a set of segments, specified by BCSEG entries, where the flow boundary is located. See Remark 6. (Integer > 0)
MESH	Denotes the ID of the Euler mesh to which the boundary condition has to be applied. See Remark 7. (Integer > 0)
DIR	Allowed values are: NEGX, POSX, NEGY, POSY, NEGZ and POSZ. See Remark 8. (Character)
XMIN-ZMAX	Defines a square by specifying the ranges of the x,y,z coordinates. For a square in for example the x-plane it is required that either XMIN = XMAX or that XMAX is left blank. See Remark 9. (Real)

Remarks:

1. Only can be used for SOL 700.
2. UNAME can be:

Subroutine Name	Function
EXFLOW	Standard user defined flow boundary on the Euler mesh

3. For multi material models EXFLOW allows for the definition of any material to flow into the Eulerian mesh. The outflow can only be of materials present in the mesh.
4. TLOAD1 entries referencing FLOWUDS entries must have the TID field blank or zero.
5. BCID is optional. If used, all other inputs are ignored. If not used, the flow boundary can be defined by either using DIR or by using XMIN, XMAX, YMIN, etc.
6. The MESH-ID is only used when multiple Euler domains have been defined and when BCID is blank. If multiple Euler domains have been defined but if the MESH-ID is blank all Euler domains will be considered in assigning the boundary condition.
7. DIR is optional. It will only be used when BCID is blank. When DIR is used XMIN, XMAX, YMIN etc. are ignored.
8. XMIN, XMAX, YMIN, etc. are only used when both BCID and DIR are blank. If neither the MIN nor MAX value has been set the default value is respectively  $-1E+20$  and  $1E+20$  for the MIN and MAX value. If the MIN value has been set the default value of the MAX value is the MIN value.

## FLSYM Axisymmetric Symmetry Control

Defines the relationship between the axisymmetric fluid and a structural boundary having symmetric constraints. The purpose is to allow fluid boundary matrices to conform to structural symmetry definitions.

Format:

1	2	3	4	5	6	7	8	9	10
FLSYM	M	S1	S2						

Example:

FLSYM	12	S	A						
-------	----	---	---	--	--	--	--	--	--

Describer	Meaning
M	Number of symmetric sections of structural boundary around the circumference of the fluid being modeled by the set of structural elements. (Even Integers $\geq 2$ )
S1, S2	Description of boundary constraints used on the structure at the first and second planes of symmetry. (Character: "S" means symmetric, "A" means antisymmetric.)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. Only one FLSYM entry is allowed.
3. This entry is not required if there are no planes of symmetry.
4. First plane of symmetry is assumed to be at  $\phi = 0$ . Second plane of symmetry is assumed to be at  $\phi = 360^\circ/M$ .
5. Symmetric and antisymmetric constraints for the structure must, in addition, be provided by the user.
6. The solution is performed for those harmonic indices listed on the AXIF entry that are compatible with the symmetry conditions.
7. For example, if FLSYM is used to model a quarter section of structure at the boundary,  $M = 4$ . If the boundary constraints are "SS", the compatible cosine harmonics are 0, 2, 4, ..., etc. If "SA" is used, the compatible cosine harmonics are 1, 3, 5, ..., etc.



## FLUTTER

## Aerodynamic Flutter Data

Defines data needed to perform flutter analysis.

Format:

1	2	3	4	5	6	7	8	9	10
FLUTTER	SID	METHOD	DENS	MACH	RFREQ	IMETH	NVALUE/ OMAX	EPS	

Example:

FLUTTER	19	K	119	219	319	S	5	1.-4	
---------	----	---	-----	-----	-----	---	---	------	--

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
METHOD	Flutter analysis method. (Character: “K” for K method, “PK” for PK method, “PKNL” for PK method with no looping, “PKS” for PK sweep method, “PKNLS” for PK sweep method with no looping, “KE” for the K method restricted for efficiency.) See Remark 9.
DENS	Identification number of an FLFACT entry specifying density ratios to be used in flutter analysis. (Integer > 0)
MACH	Identification number of an FLFACT entry specifying Mach numbers ( $m$ ) to be used in flutter analysis. (Integer > 0)
RFREQ (or VEL)	Identification number of an FLFACT entry specifying reduced frequencies ( $k$ ) to be used in flutter analysis; for the “PKx” methods, the velocities FLFACT entry is specified in this field. (Integer > 0)
IMETH	Choice of interpolation method for aerodynamic matrix interpolation. See Remark 6. (Character: “L” = linear, “S” = surface; “TCUB” = termwise cubic; Default = “L”.)
NVALUE	Number of eigenvalues beginning with the first eigenvalue for output and plots. (Integer > 0; Default is the number of modal degrees-of-freedom ( $u_h$ ).)
OMAX	For the PKS and PKNLS methods, OMAX specifies the maximum frequency, in Hz., to be used in the flutter sweep. (Real > 0.0; Default = maximum normal mode eigenfrequency)
EPS	Convergence parameter for $k$ . Used in the PK and PKNL methods only. See Remark 4. (Real > 0.0; Default = $10^{-3}$ .) See Remark 9. for the meaning of EPS when the PKS or PKNLS methods are being used.

Remarks:

1. The FLUTTER entry must be selected with the Case Control command FMETHOD = SID.

2. The density is given by  $DENS \cdot RHOREF$ , where RHOREF is the reference value specified on the AERO entry and DENS is the density ratio specified on the FLFACT entry.
3. The reduced frequency is given by  $k = (REFC \cdot \omega / 2 \cdot V)$ , where REFC is given on the AERO entry,  $\omega$  is the circular frequency, and  $V$  is the velocity. If  $k = 0.0$ , as specified on the FLFACT entry, then only the K method may be specified and the Inverse Power method of eigenvalue extraction (INV on the EIGC entry) must be used. Aeroelastic divergence analysis is more appropriately performed using one of the “PKx” methods.
4. For the PK and PKNL methods, an eigenvalue is accepted when:

$$|k - k_{estimate}| < EPS \quad \text{for } k_{estimate} < 1.0$$

$$|k - k_{estimate}| < EPS \cdot k_{estimate} \quad \text{for } k_{estimate} \geq 1.0$$

5. When one of the “PKx” methods is selected, physical displacements will only be generated for the velocities on the FLFACT that are specified as negative values of the requested velocity. Also, structural damping as specified on the GE field of MATi entries is ignored.
  6. If IMETH = “L”, a linear interpolation is performed on reduced frequencies at the Mach numbers specified on the FLFACT entry using the MKAEROi entry Mach number that is closest to the FLFACT entry Mach number. For IMETH = “S”, a surface interpolation is performed across Mach numbers and reduced frequencies. For IMETH = “TCUB” a termwise cubic interpolation on reduced frequency is used. IMETH = “S” is only available for the “K” and “KE” flutter methods. IMETH = “TCUB” is only available for the “PKx” methods. For the “PKx” methods, IMETH = “S” or “L” or blank provides linear interpolation while “TCUB” provides a termwise cubic interpolation.
  7. For the “K”, “KE”, “PK”, and “PKS” methods, all combinations of the FLFACT entry are analyzed. For the “PKNL” and “PKNLS” methods, only ordered pairs are analyzed; i.e.,  $(\rho_1, M_1, V_1), (\rho_2, M_2, V_2) \dots (\rho_n, M_n, V_n)$ . For the PKNL and PKNLS methods, equal number of densities, Mach numbers and velocities must be specified.
  8. “K” and “KE” methods are not supported for design sensitivity and optimization.
  9. The PKS and PKNLS methods determine flutter eigenvalues by performing a sweep of equally spaced reduced frequencies ranging from  $k_{est} = 0.0$  through  $k_{est} = \pi \cdot REFC \cdot OMAX / \text{Velocity}$ . The number of intervals is calculated using  $NINT = INT(1.0/EPS)$ .
- 10.
- The PK method uses only real matrix terms for computing the flutter solution. This means any imaginary terms in any of the matrices, M, B, or K are ignored with the PK method and the imaginary part of the aerodynamic matrix is added as a real matrix to the viscous damping matrix B.
  - For the KE method, the B matrix is ignored while complex stiffness forms of structural damping are supported. Modal viscous damping (TABDMP1 input) will be included when the KDAMP parameter is set to -1.
  - The K method supports all forms of damping.

11. METHOD=K requires a CMETHOD in Case Control pointing to an EIGC Bulk Data entry. It is recommended that the number of roots requested on the EIGC entry be twice the number of normal modes used in the modal flutter analysis.

## FORCE

## Static Force

Defines a static concentrated force at a grid point by specifying a vector.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE	SID	G	CID	F	N1	N2	N3		

Example:

FORCE	2	5	6	2.9	0.0	1.0	0.0		
-------	---	---	---	-----	-----	-----	-----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer $\geq$ 0; Default = 0)
F	Scale factor. (Real)
Ni	Components of a vector measured in coordinate system defined by CID. (Real; at least one $N_i \neq 0.0$ , unless F is zero)

Remarks:

1. The static force applied to grid point G is given by

$$\vec{f} = F\vec{N}$$

where  $\vec{N}$  is the vector defined in fields 6, 7 and 8. The magnitude of  $\vec{f}$  is equal to F times the magnitude of  $\vec{N}$ .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank (the default) references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body: the MID of a rigid material MATRIG or the EID of a RBE2. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2 and a rigid material are used with these TYPEs. SOL 700 only.
6. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

## FORCE1

## Follower Force, Alternate Form 1

Defines a concentrated force at a grid point by specification of a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE1	SID	G	F	G1	G2				

Example:

FORCE1	6	13	-2.93	16	13				
--------	---	----	-------	----	----	--	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
G1, G2	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident.)

Remarks:

1. The force applied to grid point G is given by

$$\vec{f} = F\vec{n}$$

where  $\vec{n}$  is a unit vector parallel to a vector from G1 to G2.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

## FORCE2

## Follower Force, Alternate Form 2

Defines a concentrated force at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
FORCE2	SID	G	F	G1	G2	G3	G4		

Example:

FORCE2	6	13	-2.93	16	13	17	13		
--------	---	----	-------	----	----	----	----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number. (Integer > 0)
F	Magnitude of the force. (Real)
Gi	Grid point identification numbers. (Integer > 0; G1 and G2 may not be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The direction of the force is parallel to the cross product of vectors from G1 to G2 and G3 to G4.
2. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
4. For axisymmetric elements, the point loads to be entered on the card should be obtained by integrating over 1 radian of the circumference.

## FORCEAX

### Conical Shell Static Force

Defines a static concentrated force on a conical shell ring.

Format:

1	2	3	4	5	6	7	8	9	10
FORCEAX	SID	RID	HID	S	FR	FP	FZ		

Example:

FORCEAX	1	2	3	2.0	0.1	0.2	0.3		
---------	---	---	---	-----	-----	-----	-----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
RID	RINGAX entry identification number. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark 5. (Integer ≥ 0)
S	Scale factor for the force. (Real)
FR, FP, FZ	Force components in $r$ , $\phi$ , $z$ directions. (Real)

Remarks:

1. FORCEAX is allowed only if an AXIC entry is also present.
2. Axisymmetric shell loads must be selected with the Case Control command LOAD = SID.
3. A separate entry is needed for the definition of the force associated with each harmonic.
4. See [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide* for further discussion.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: “Sn1Tn2” where n1 is the start of the sequence and n2 is the end of the sequence (e.g., for harmonics 0 through 10, the field would contain “S0T10”).

## FORCUDS

### User-defined Enforced Motion at Grid Points

Defines enforced motion at grid points specified by a user subroutine. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
FORCUDS	LID	GROUP	UNAME						
+	G1	G2	THRU	G3					

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE extvel 'SCA.MDSolver.Obj.Uds.Dytran.Loads '
```

*In Bulk Data:*

FORCUD	1	EXTVEL							
+	1	2	3	5					

Describer	Meaning
LID	Number of a set of loads.(Integer>0;required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; required)
UNAME	User subroutine name associated with the entry. (Character; default=EXTVEL)
Gi	Numbers of the grid points that are constrained. If the word THRU appears between two numbers, all the numbers in the range are included in the list. BY indicates the increment to be used within this range. (Integer or character; required.)

Remarks:

1. LID must be referenced by a TLOAD1 entry.
2. FORCUDS can only be used to specify enforced velocities for grid points. The TYPE field on the TLOAD1 entry must be set to two. The TID on the TLOAD1 entry must be set to zero or blank (no time variation).
3. The GROUP name must be unique from all other GROUP names if more than one FORCUDS is used. Therefore each FORCUDS must have its own group name in a FMS section CONNECT SERVICE statement.
4. The constraint name is passed to the subroutine and can be used to identify the constraint.
5. A THRU specification, including the start and finish points in the range, must be on one line.
6. If the THRU specification is used, all the points in the sequence do not have to exist. Those that do not exist are ignored. The first point in the THRU specification must be a valid grid point. BY can be used to exclude grid points.



7. None of the fields in the list of grid points can be blank or zero, since this designation marks the end of the list.
8. Any number of continuation lines can be used to define the list of grid points.
9. UNAME can be:

Subroutine Name	Function
EXTVEL	user defined velocities on grid points

## FREEPT Fluid Free Surface Point

Defines the location of points on the surface of a fluid for recovery of surface displacements in a gravity field.

Format:

1	2	3	4	5	6	7	8	9	10
FREEPT	IDF	IDP1	PHI1	IDP2	PHI2	IDP3	PHI3		

Example:

FREEPT	3	301	22.5	302	90.0	303	370.0		
--------	---	-----	------	-----	------	-----	-------	--	--

Describer	Meaning
IDF	RINGFL entry identification number. (Integer > 0)
IDPi	Free surface point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point (RINGFL entry) in the fluid coordinate system. (Real)

Remarks:

1. FREEPT is allowed only if an AXIF entry is also present.
2. All free surface point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The free surface points are used for the identification of output data only.
4. Three points may be defined on a single entry.
5. The referenced fluid point (IDF) must be included in a free surface list (FSLIST entry).
6. Output requests for velocity and acceleration can be made at these points.

## FREQ Frequency List

Defines a set of frequencies to be used in the solution of frequency response problems.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ	SID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	F10	-etc.-					

Example:

FREQ	3	2.98	3.05	17.9	21.3	25.6	28.8	31.2	
	29.2	22.4	19.3						

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
Fi	Frequency value in units of cycles per unit time. (Real ≥ 0.0)

Remarks:

1. Frequency sets must be selected with the Case Control command FREQUENCY = SID.
2. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

3. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FREQ1

### Frequency List, Alternate Form 1

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ1	SID	F1	DF	NDF					

Example:

FREQ1	6	2.9	0.5	13					
-------	---	-----	-----	----	--	--	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
F1	First frequency in set. (Real ≥ 0.0)
DF	Frequency increment. (Real > 0.0)
NDF	Number of frequency increments. (Integer > 0; Default = 1)

Remarks:

1. FREQ1 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and DF are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 + DF \cdot (i-1)$$

where  $i = 1$  to  $(NDF + 1)$ .

4. All FREQ<sub>i</sub> entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|,$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQ<sub>i</sub> entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FREQ2 Frequency List, Alternate Form 2

Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ2	SID	F1	F2	NF					

Example:

FREQ2	6	1.0	8.0	6					
-------	---	-----	-----	---	--	--	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
F1	First frequency. (Real > 0.0)
F2	Last frequency. (Real > 0.0, F2 > F1)
NF	Number of logarithmic intervals. (Integer > 0; Default = 1)

Remarks:

1. FREQ2 entries must be selected with the Case Control command FREQUENCY = SID.
2. The units for F1 and F2 are cycles per unit time.
3. The frequencies defined by this entry are given by

$$f_i = F1 \cdot e^{(i-1)d}$$

where  $d = \frac{1}{NF} \ln \frac{F2}{F1}$  and  $i = 1, 2, \dots, (NF + 1)$

In the example above, the list of frequencies will be 1.0, 1.4142, 2.0, 2.8284, 4.0, 5.6569 and 8.0 cycles per unit time.

4. All FREQi entries with the same frequency set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < DFREQ \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

5. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FREQ3 Frequency List, Alternate 3

Defines a set of excitation frequencies for modal frequency-response solutions by specifying number of excitation frequencies between two modal frequencies.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ3	SID	F1	F2	TYPE	NEF	CLUSTER			

Example:

FREQ3	6	20.0	2000.0	LINEAR	10	2.0			
-------	---	------	--------	--------	----	-----	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of modal frequency range in cycles per unit time. (Real $\geq$ 0.0 for TYPE = LINEAR and Real = 0.0 for TYPE = LOG)
F2	Upper bound of modal frequency range in cycles per unit time. (Real > 0.0; F2 $\geq$ F1; Default = F1)
TYPE	LINEAR or LOG. Specifies linear or logarithmic interpolation between frequencies. (Character; Default = "LINEAR")
NEF	Number of excitation frequencies within each subrange including the end points. The first subrange is between F1 and the first modal frequency within the bounds. The second subrange is between first and second modal frequencies between the bounds. The last subrange is between the last modal frequency within the bounds and F2. (Integer > 1; Default = 10)
CLUSTER	Specifies clustering of the excitation frequency near the end points of the range. See Remark 6. (Real > 0.0; Default = 1.0)

Remarks:

1. FREQ3 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ3 entries must be selected with the Case Control command FREQUENCY = SID.
3. In the example above, there will be 10 frequencies in the interval between each set of modes within the bounds 20 and 2000, plus 10 frequencies between 20 and the lowest mode in the range, plus 10 frequencies between the highest mode in the range and 2000.
4. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

5. All FREQ<sub>i</sub> entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter, with a default of  $10^{-5}$ .  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQ<sub>i</sub> entries.

6. CLUSTER is used to obtain better resolution near the modal frequencies where the response varies the most. CLUSTER > 1.0 provides closer spacing of excitation frequency towards the ends of the frequency range, while values of less than 1.0 provide closer spacing towards the center of the frequency range. For example, if the frequency range is between 10 and 20, NEF = 11, TYPE = "LINEAR"; then, the excitation frequencies for various values of CLUSTER would be as shown in Table 17.

$$\hat{f}_k = \frac{1}{2}(\hat{f}_1 + \hat{f}_2) + \frac{1}{2}(\hat{f}_2 - \hat{f}_1)|\xi|^{1/\text{CLUSTER}} \cdot \text{SIGN}(\xi)$$

where

$\xi$  =  $-1 + 2(k - 1) / (\text{NEF} - 1)$  is a parametric coordinate between -1 and 1

$k$  = varies from 1 to NEF ( $k = 1, 2, \dots, \text{NEF}$ )

$\hat{f}_1$  = is the lower limit of the frequency subrange

$\hat{f}_2$  = is the upper limit of the subrange

$\hat{f}_k$  = is the k-th excitation frequency

$\hat{f}$  = is the frequency, or the logarithm of the frequency, depending on the value specified for TYPE

Table 17 CLUSTER Usage Example

Excitation Frequency Number	$\xi$	CLUSTER				
		c=0.25	c=0.50	c-1.0	c-2.0	c-4.0
		Excitation Frequencies in Hertz				
1	-1.0	10.00	10.0	10.0	10.00	10.00
2	-0.8	12.95	11.8	11.0	10.53	10.27
3	-0.6	14.35	13.2	12.0	11.13	10.60
4	-0.4	14.87	14.2	13.0	11.84	11.02
5	-0.2	14.99	14.8	14.0	12.76	11.66
6	0.0	15.00	15.0	15.0	15.00	15.00
7	0.2	15.01	15.2	16.0	17.24	18.34



Table 17 CLUSTER Usage Example

Excitation Frequency Number	$\xi$	CLUSTER				
		c=0.25	c=0.50	c=1.0	c=2.0	c=4.0
		Excitation Frequencies in Hertz				
8	0.4	15.13	15.8	17.0	18.16	18.98
9	0.6	15.65	16.8	18.0	18.87	19.40
10	0.8	17.05	18.2	19.0	19.47	19.73
11	1.0	20.00	20.0	20.0	20.00	20.00

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FREQ4

## Frequency List, Alternate Form 4

Defines a set of frequencies used in the solution of modal frequency-response problems by specifying the amount of “spread” around each natural frequency and the number of equally spaced excitation frequencies within the spread.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ4	SID	F1	F2	FSPD	NFM				

Example:

FREQ4	6	20.0	2000.0	0.30	21				
-------	---	------	--------	------	----	--	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0; F2 > F1; Default = 1.0E20)
FSPD	Frequency spread, +/- the fractional amount specified for each mode which occurs in the frequency range F1 to F2. (1.0 > Real > 0.0; Default = 0.10)
NFM	Number of evenly spaced frequencies per “spread” mode. (Integer > 0; Default = 3; If NFM is even, NFM + 1 will be used.)

Remarks:

1. FREQ4 applies only to modal frequency-response solutions (SOLs 111, 146, and 200 and is ignored in direct frequency-response solutions.
2. FREQ4 entries must be selected with the Case Control command FREQUENCY = SID.
3. There will be NFM excitation frequencies between  $(1 - FSPD) \cdot f_N$  and  $(1 + FSPD) \cdot f_N$ , for each natural frequency in the range F1 to F2.
4. In the example above there will be 21 equally spaced frequencies across a frequency band of  $0.7 \cdot f_N$  to  $1.3 \cdot f_N$  for each natural frequency that occurs between 20 and 2000. See [Figure 9-91](#) for the definition of frequency spread.

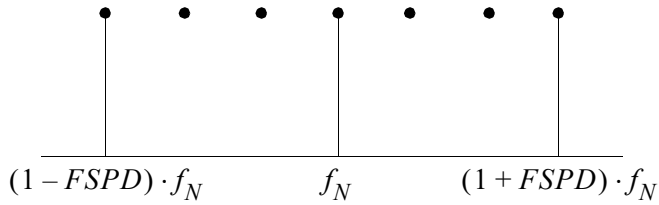


Figure 9-91 Frequency Spread Definition

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

- The frequency spread can be used also to define the half-power bandwidth. The half-power bandwidth is given by  $2 \cdot \xi \cdot f_N$ , where  $\xi$  is the damping ratio. Therefore, if FSPD is specified equal to the damping ratio for the mode, NFM specifies the number of excitation frequency within the half-power bandwidth. See [Figure 9-92](#) for the definition of half-power bandwidth.

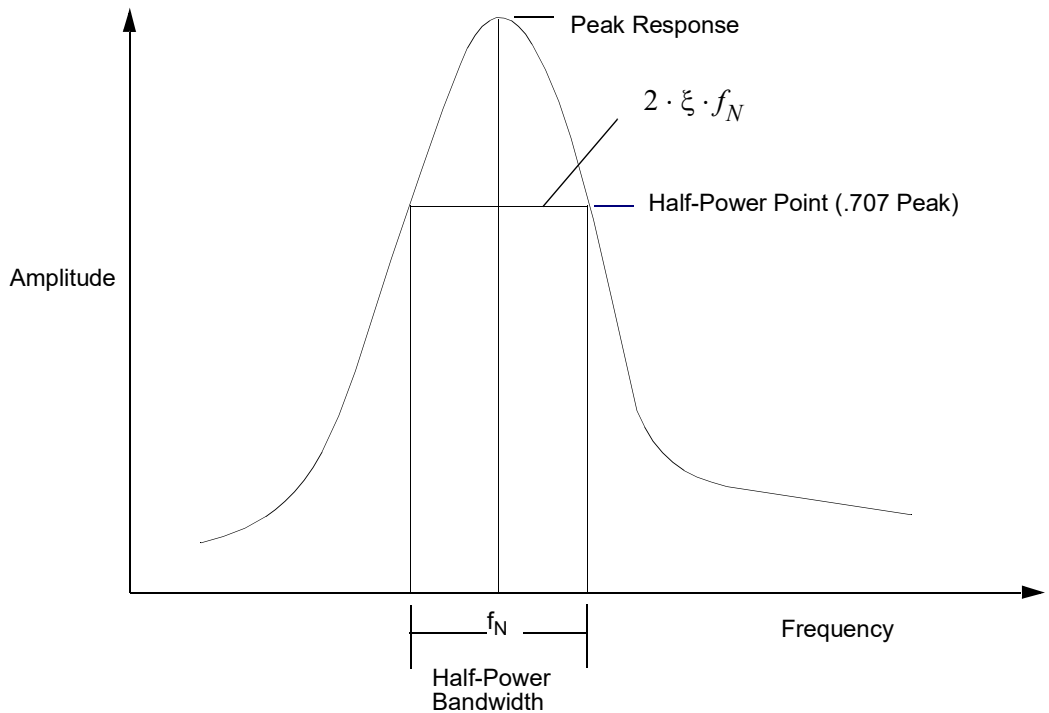


Figure 9-92 Half-Power Bandwidth Definition

6. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.
7. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ . The values  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

8. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
9. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FREQ5 Frequency List, Alternate Form 5

Defines a set of frequencies used in the solution of modal frequency-response problems by specification of a frequency range and fractions of the natural frequencies within that range.

Format:

1	2	3	4	5	6	7	8	9	10
FREQ5	SID	F1	F2	FR1	FR2	FR3	FR4	FR5	
	FR6	FR7	-etc.-						

Example:

FREQ5	6	20.0	2000.0	1.0	0.6	0.8	0.9	0.95	
	1.05	1.1	1.2						

Describer	Meaning
SID	Set identification number. (Integer > 0)
F1	Lower bound of frequency range in cycles per unit time. (Real ≥ 0.0; Default = 0.0)
F2	Upper bound of frequency range in cycles per unit time. (Real > 0.0; F2 > F1; Default = 1.0E20)
FRi	Fractions of the natural frequencies in the range F1 to F2. (Real > 0.0)

Remarks:

1. FREQ5 applies only to modal frequency-response solutions (SOLs 111, 146, and 200) and is ignored in direct frequency response solutions.
2. FREQ5 entries must be selected with the Case Control command FREQUENCY = SID.
3. The frequencies defined by this entry are given by

$$f_i = FR_i \cdot f_{N_i}$$

where  $f_{N_i}$  are the natural frequencies in the range F1 through F2.

4. In the example above, the list of frequencies will be 0.6, 0.8, 0.9, 0.95, 1.0, 1.05, 1.1, and 1.2 times each natural frequency between 20 and 2000. If this computation results in excitation frequencies less than F1 and greater than F2, those computed excitation frequencies are ignored.

Excitation frequencies may be based on natural frequencies that are not within the range (F1 and F2) as long as the calculated excitation frequencies are within the range. Similarly, an excitation frequency calculated based on natural frequencies within the range (F1 through F2) may be excluded if it falls outside the range.

5. Since the forcing frequencies are near structural resonances, it is important that some amount of damping be specified.

6. All FREQi entries with the same set identification numbers will be used. Duplicate frequencies will be ignored.  $f_N$  and  $f_{N-1}$  are considered duplicated if

$$|f_N - f_{N-1}| < \text{DFREQ} \cdot |f_{MAX} - f_{MIN}|$$

where DFREQ is a user parameter with a default of  $10^{-5}$ . The values  $f_{MAX}$  and  $f_{MIN}$  are the maximum and minimum excitation frequencies of the combined FREQi entries.

7. In design optimization (SOL 200), the excitation frequencies are derived from the natural frequencies computed at each design cycle.
8. In modal analysis, solutions for modal DOFs from rigid body modes at zero excitation frequencies may be discarded. Solutions for nonzero modes are retained.
9. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FRFCOMP

### Frequency Response Function (FRF) Component Specification for FRF Based Assembly (FBA)

Specifies the FRF components that are to be assembled as part of an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFCOMP	COMPID	COMPNAME	MEDIUM	UNITNO	LSCALFAC	FSCALFAC			

Examples:

FRFCOMP	10	LEFTWING	OP2	25					
FRFCOMP	20	FRAME	UF	30	1000.0				

Describer	Meaning
COMPID	ID of the component whose FRFs have been generated in a previous Nastran execution. (Integer > 0)
COMPNAME	Name of the COMPID FRF component. See Remark 1. (Up to 8 characters).
MEDIUM	Medium on which the FRF matrices and other related data are stored. Acceptable character values are DB (for database), OP2 (for OUTPUT2 file) or UF (for Universal file). See Remarks 2., 3. and 4.
UNITNO	Fortran unit number for the OP2 and UF options. (Integer > 0). See Remarks 3. and 4.
LSCALFAC	Length scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10. and 11.
FSCALFAC	Force scale factor for the UF option. (Real > 0.0; Default = 1.0). See Remarks 7., 8., 9., 10., and 11.

Remarks:

- COMPNAME may be referenced by the ASMOUT keyword in the FRF Case Control command in order to request output for a specific FRF component in the FBA process. Accordingly, COMPNAME may not have the values of CONNINFO, ALL, COMP or ASSEMBLY as these are all reserved words for use with the ASMOUT keyword in the FRF Case Control command.
- If the DB option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the database for the specified FRF component:  
 ASSIGN dbname = 'frfgen\_job.MASTER  
 DBLOCATE DATBLK = (FRFDB) LOGICAL = dbname

3. If the OP2 option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the OUTPUT2 file for the specified FRF component:  
ASSIGN INPUTT2 = 'frfgen\_job\_op2' UNIT = 25
4. If the UF option is specified, then the following type of ASSIGN should be specified in the FMS section of the FBA job to access information on the Universal file for the specified FRF component:  
ASSIGN UNVFILE = 'frfgen\_job\_unv' UNIT = 26
5. Component ID of 0 is assigned to the assembled FRF configuration resulting from the FBA process.
6. An FRF generation job using an FRF Case Control command with GEN/GENASM and COMPID and COMPNAME keywords specified in it will automatically generate and save an FRFCOMP Bulk Data entry on the assembly punch (.asm) file for that FRF component for subsequent use in an FBA process.
7. LSCALFAC and FSCALFAC are meaningful only for the UF option. They are ignored for the DB and OP2 options.
8. The FRF and other information on the Universal File (UF) is grouped by so-called Universal Dataset Numbers (UDNs). The heart of the information on the UF for a test FRF component is in UDN 58 which contains FRF data for that component. Also of interest, if present, is UDN 15 which contains grid point coordinate data in single precision or UDN 2411 which contains grid point coordinate data in double precision. Further, if UDN 15 or UDN 2411 is present, the data therein may reference definition or displacement coordinate systems. Such coordinate system data is resident in UDN 18.

Details of the various UDNs and their formats can be obtained from the following websites:

<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1>

<http://www.sdrl.uc.edu/universal-file-formats-for-modal-analysis-testing-1/file-format-storehouse/file-formats>

9. The points of a test FRF component that are considered by the FBA process are those that are defined in UDN 58 either as response points or as excitation points. Any such point that appears both as a response point and as an excitation point is regarded as a potential connection point of the test FRF component.
10. Connections between test FRF components and other FRF components are determined by what is supplied on the UF as indicated below.
  - a. If UDN 15 or UDN 2411 is supplied on the UF, then connections between the test FRF component grid points and those of other FRF components will be based on *matching coordinates*.
  - b. If neither UDN 15 nor UDN 2411 is supplied, then normally, connections between the test FRF component grid points and those of other FRF components will be based on *matching grid point IDs*. However, the user can override this by specifying FRFCONN entries, thereby causing connections between *user specified grid points*. In any case, when the connections are based on matching grid point IDs or based on FRFCONN entries, it is the user's responsibility to ensure that the connection grid points have the same physical locations in space so that correct results are obtained from the FBA process. The program clearly cannot check for this condition since the coordinates of one or more of the points are not available.
11. For the UF option, LSCALFAC and FSCALFAC are used as follows:



- For Universal Dataset No. (UDN) 15 or 2411 (if present) on the Universal File:  
The X, Y, and Z coordinate values in the UDN are multiplied by LSCALFAC before they are used in the FBA process.
- For Universal Dataset No. (UDN) 58 on the Universal File:  
The FRF response quantities in the UDN are multiplied by appropriate factors as shown in the following table before they are used in the FBA process.

Table 9-16 Multiplication Factors for FRF Response Quantities in UDN 58 of the Universal File

FRF Response Quantity	Type of Excitation	Factor by Which FRF Response Quantity is Multiplied Before It is Used in the FBA Process
Translational displacement, velocity or acceleration at a structural point	Unit force at an excitation point	LSCALFAC/FSCALFAC
	Unit moment at an excitation point	1.0/FSCALFAC
Rotational displacement, velocity or acceleration at a structural point	Unit force at an excitation point	1.0/FSCALFAC
	Unit moment at an excitation point	1.0/(LSCALFAC*FSCALFAC)
Pressure at an acoustic point	Unit force at an excitation point	1.0/(LSCALFAC**2)
	Unit moment at an excitation point	1.0/(LSCALFAC**3)

## FRFCONN

## FRF Component Explicit Connection for FRF Based Assembly (FBA)

Defines explicit connection data for FRF components in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFCONN	CONNID	COMPID1/ COMPNAM1	POINT1	COMPID2/ COMPNAM2	POINT2				

Examples:

FRFCONN	10	5	100	15	200				
---------	----	---	-----	----	-----	--	--	--	--

FRFCONN	20	STRUT	25	WING	35				
---------	----	-------	----	------	----	--	--	--	--

Describer	Meaning
CONNID	Unique identification number of the FRFCONN entry. (Integer > 0)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAMEi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remarks 2. through 6. (Integer > 0)

Remarks:

1. If neither of the FRF components COMPID1/COMPNAME1 and COMPID2/COMPNAME2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINTi must be among the connection points of the corresponding FRF component COMPIDi/COMPNAMEi. If not, the program terminates the job with a user fatal message.
3. It is, in general, not necessary to have FRFCONN entries when the connections in the FBA process involve only grid points. In the FBA process, all connection grid points of FRF components that have the same basic coordinates are normally connected automatically without any user intervention or specification.
4. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.
5. FRFCONN entries are required if the user wants to specify explicit connections between test FRF component grid points whose coordinates are not available and those of other FRF components. See Remark 10. under the description of the FRFCOMP entry for further details.

6. FRFCONN entries are also required if the user wants to combine scalar points of FRF components in the FBA process.

## FRFFLEX

## FRF Component Flexible Connection Specification for FRF Based Assembly (FBA)

Defines properties for flexible connections between FRF components in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFFLEX	FLEXID	C	COMPID1/ COMPNAM1	POINT1	COMPID2/ COMPNAM2	POINT2	KVALUE/ KTABID	BVALUE/ BTABID	
	GEVALUE/ GETABID								

Examples:

FRFFLEX	10	1	100	15	200	25	100.0	150	
	0.02								

FRFFLEX	20	4	STRUT	120	WING	260	10	0.25	
---------	----	---	-------	-----	------	-----	----	------	--

Descriptor	Meaning
FLEXID	Unique identification number of the FRFFLEX entry. (Integer > 0)
C	A single component number. (Any integer between 1 and 6 for grid points; integer 0 or blank for scalar points.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Integer > 0)
COMPNAMi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 1. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remark 2. (Integer > 0)
KVALUE	Elastic property value (force per unit displacement). See Remark 3. (Real > 0.0 or blank)
KTABID	Identification number of a TABLEDi entry that defines the elastic property value (force per unit displacement) as a function of frequency. See Remark 3. (Integer > 0 or blank)
BVALUE	Damping property value (force per unit velocity). See Remark 3. (Real > 0.0 or blank)
BTABID	Identification number of a TABLEDi entry that defines the damping property value (force per unit velocity) as a function of frequency. See Remark 3. (Integer > 0 or blank)
GEVALUE	Damping coefficient value. See Remarks 4., 5. and 6. (Real > 0.0 or blank)
GETABID	Identification number of a TABLEDi entry that defines the damping coefficient value as a function of frequency. See Remarks 4., 5. and 6. (Integer > 0 or blank)

**Remarks:**

1. If neither of the FRF components COMPID1/COMPNAM1 and COMPID2/COMPNAM2 is part of the FBA process, then this entry is ignored. However, if one of them is part of the FBA process but not the other, the program terminates the job with a user fatal message.
2. POINT<sub>i</sub> must be among the connection points of the corresponding FRF component COMPID<sub>i</sub>/COMPNAM<sub>i</sub>. If not, the program terminates the job with a user fatal message.
3. The KVALUE/KTABID and BVALUE/BTABID fields may not both be blank.
4. The continuation entry is not needed if GEVALUE/GETABID is not to be defined.
5. GEVALUE/GETABID may not be specified unless KVALUE/KTABID is specified.
6. To obtain the damping coefficient, multiply the critical damping ratio  $C/C_0$  by 2.0.
7. It is important to note that this entry by itself does *not* define a connection between the specified points. It merely defines properties for flexible connections between two points whose connection is established either explicitly via an FRFCONN entry or is implied by automatic connection.
8. The flexible connection properties for component C of connection points POINT1 and POINT2 may be defined on more than one FRFFLEX entry.
9. In the absence of FRFFLEX data, the program assumes rigid connections between the corresponding components.
10. A grid point component may not appear on both an FRFFLEX entry and an FRFRELS entry. If it does, the program identifies such usage and terminates the job with a user fatal message.

## FRFRELS

## FRF Component Grid Point Release for FRF Based Assembly (FBA)

Defines the degrees-of-freedom of FRF component connection grid points that are not to be connected in an FRF Based Assembly (FBA) process.

Format:

1	2	3	4	5	6	7	8	9	10
FRFRELS	SID	C	COMPID1/ COMPNAM1	GRIDPNT1	COMPID2/ COMPNAM2	GRIDPNT2	COMPID3/ COMPNAM3	GRIDPNT3	

Examples:

FRFRELS	100	45	10	15	BODY	20	FRAME	30	
---------	-----	----	----	----	------	----	-------	----	--

FRFRELS	20	1	WING	25	NACELLE	35			
---------	----	---	------	----	---------	----	--	--	--

Describer	Meaning
SID	Identification number of the FRFRELS entry. (Integer > 0)
C	Component number(s). See Remark 1. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAMi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
GRIDPNTi	Grid point identification number. See Remarks 3. and 4. (Integer > 0)

Remarks:

1. The grid point component(s) specified by C will not be connected in the FBA process.
2. If FRF component COMPIDi/COMPNAMi is not part of the FBA process, then the release data for that FRF component is ignored.
3. GRIDPNTi must be among the connection points of the corresponding FRF component COMPIDi/COMPNAMi. If not, the program terminates the job with a user fatal message.
4. If GRIDPNTi is not connected to any other grid point in the FBA process, the program ignores the FRFRELS data for this point and issues a warning message indicating this to the user.
5. If the connection points of an FRF component in the FBA process consist of coincident grid points, the program identifies such points via a user information message. All such coincident grid points must be referenced on FRFCONN or FRFRELS entries in order to ensure proper connections in the FBA process. In the absence of such specifications for coincident connection grid points, the program terminates the job with a user fatal error.

## FRFSPC1

## FRF Component Single-Point Constraint for FRF Based Assembly (FBA)

Defines single-point constraints for FRF component connection points in an FRF Based Assembly (FBA) process.

## Format:

1	2	3	4	5	6	7	8	9	10
FRFSPC1	SID	C	COMPID1/ COMPNAM1	POINT1	COMPID2/ COMPNAM2	POINT2	COMPID3/ COMPNAM3	POINT3	

## Examples:

FRFSPC1	100	4	10	12	ENGINE	23	FRAME	31	
FRFSPC1	20	1	STRUT	25	NACELLE	35			

Descriptor	Meaning
SID	Identification number of the single-point constraint set. See Remark 1. (Integer > 0)
C	Component number(s). (Any unique combination of the integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
COMPIDi	Identification number of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Integer > 0)
COMPNAMi	Name of an FRF component whose FRFs have been generated in a previous Nastran execution. See Remark 2. (Up to 8 characters; no blank allowed)
POINTi	Grid or scalar point identification number. See Remarks 3. and 4. (Integer > 0)

## Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. If FRF component COMPIDi/COMPNAMi is not part of the FBA process, then the single-point constraint data for that FRF component is ignored.
3. POINTi must be among the connection points of the corresponding FRF component COMPIDi/COMPNAMi. If not, the program terminates the job with a user fatal message.
4. If POINTi is not connected to any other point in the FBA process, the program ignores the FRFSPC1 data for this point and issues a warning message indicating this to the user.
5. Unlike in non-FBA jobs, wherein a degree-of-freedom that has SPCs specified for it yields exact zero results, a degree-of-freedom that has an FRFSPC1 specified for it in an FBA job will, in general, not give exact zero results, but will give results that are nearly zero or very close to zero, usually around 1.0E-12 or so.

**FRFXIT**

**Unit Load Degree-of-Freedom Specification for Frequency Response Function (FRF) Computations**

Specifies a single degree-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
FRFXIT	PNTID	C	LABEL						

Example:

FRFXIT	10	3	UNIT LOAD AT LEFT CORNER						
--------	----	---	--------------------------	--	--	--	--	--	--

Describer	Meaning
PNTID	Grid or scalar point identification number. (Integer > 0)
C	A single component number. (Integer 0 or blank for a scalar point; any integer between 1 and 6 for a grid point.)
LABEL	A string comprising no more than 48 characters (fields 4 through 9) that will be used in the label portion of the FRF output to identify this unit load specification. See Remarks <a href="#">1.</a> and <a href="#">2.</a>

Remarks:

1. The small field format must be employed for this entry. If the free field or large field format is employed, the results are unpredictable and in many cases may lead to fatal errors and subsequent termination of the job.
2. The LABEL data must have a non-blank entry in field 4.
3. The FRFXIT1 Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
4. Redundant unit load specifications are ignored.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.



## FRFXIT1

## Unit Load Degrees-of-Freedom Specification for Frequency Response Function (FRF) Computations

Specifies degrees-of-freedom where unit loads are to be applied for Frequency Response Function (FRF) generation.

Format:

1	2	3	4	5	6	7	8	9	10
FRFXIT1	C	PNTID1	PNTID2	PNTID3	PNTID4	PNTID5	PNTID6	PNTID7	

Example:

FRFXIT1	123	10	20	30	40				
---------	-----	----	----	----	----	--	--	--	--

Alternate Format and Example:

1	2	3	4	5	6	7	8	9	10
FRFXIT1	C	PNTID1	THRU	PNTID2					
FRFXIT1	123	5	THRU	15					

Describer	Meaning
C	Component numbers. (Any unique combination of integers 1 through 6 with no embedded blanks for grid points; integer 0 or blank for scalar points.)
PNTIDi	Grid or scalar point identification numbers. See Remark 1. (Integer > 0)

Remarks:

1. Points in the THRU range need not all exist.
2. The FRFXIT Bulk Data entry and the DLOAD Case Control request provide alternate means of unit load specification for FRF generation.
3. Redundant unit load specifications are ignored.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

## FSICTRL

Defines the analysis type for OpenFSI fluid structure analysis simulations in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
FSICTRL	SERV_ID	TYPE	FREQ	ANALYSIS					

Examples:

FSICTRL	scafsi	EXPLICIT	1						
---------	--------	----------	---	--	--	--	--	--	--

Describer	Meaning		
SERV_ID	OpenFSI SCA service name associated with the wetted surface loads (see <a href="#">WETLOAD</a> ). The OpenFSI SCA service is defined using the <code>CONNECT SERVICE</code> file management section statement.	Character	None
TYPE	Type of solution strategy coupling between the external code and SOL 700. TYPE can be either <code>EXPLICIT</code> or <code>IMPLICIT</code> .	Character	EXPLICIT
FREQ	External force and displacement update frequency per time step, for the exchange with the external code using the <code>IMPLICIT</code> solution strategy TYPE.	Integer > 0	1
ANALYSIS	OpenFSI service type, it is either DV or U. The DV indicates the service implements OpenFSI interface, the U means the service implements OpenFSI_Ex interface.	Character	DV

Remarks:

1. This entry is used for nonlinear SOL 400 analysis.
2. In the FMS Section:  
`CONNECT SERVICE SERV_ID 'ExternalCodeVendor.OpenFSI '`
3. The SERV\_ID string must be 8 characters or less.

## FSLIST Free Surface List

Defines the fluid points (RINGFL entry) that lie on a free surface boundary.

Format:

1	2	3	4	5	6	7	8	9	10
FSLIST	RHO	IDF1	IDF2	IDF3	IDF4	IDF5	IDF6	IDF7	
	IDF8	-etc.-							

Examples:

FSLIST	1.0-4	AXIS	432	325	416	203	256	175	
--------	-------	------	-----	-----	-----	-----	-----	-----	--

Describer	Meaning
RHO	Mass density at the surface. (Real > 0.0 or blank; the default is taken from DRHO on the AXIF entry.)
IDF1	Identification number of RINGFL entry. (Integer > 0 or Character = "AXIS" or "LAXIS") See Remark 5.
IDF2-IDFi	Identification number of additional RINGFL entries. (Unique Integers > 0)

Remarks:

1. This entry is allowed only if an AXIF entry is also present.
2. The order of the points must be sequential with the fluid on the right with respect to the direction of travel.
3. The word "AXIS" defines an intersection with the polar axis of the fluid coordinate system.
4. If the fluid density varies along the boundary, there must be one FSLIST entry for each interval between fluid points.
5. If the polar axis of the fluid coordinate system is to occur at the first point use AXIS. If the polar axis of the fluid coordinate system is to occur at the last point use LAXIS. See Remark 2.

## FTGDEF Fatigue Element Definitions

Defines elements and their associated fatigue properties to be considered for fatigue analysis for time domain SOLutions 101, 103, 112 and frequency domain SOLutions 108 and 111.

Format (SOL 101/103/112):

1	2	3	4	5	6	7	8	9	10
FTGDEF	ID	TOPSTR	PFTGID	TOPDMG	NENTS	maxENTS	NHS	HSGATE	
	"ELSET"	ELSID1	PFTGID1	ELSID2	PFTGID2	ELSID3	PFTGID3		
		ELSID4	PFTGID4	...	...	...	...		
		-etc.-							
	"SPOTW"	ELSID1	PFTGID1	ELSID2	PFTGID2	ELSID3	PFTGID3		
		ELSID4	PFTGID4	...	...	...	...		
		-etc.-							
	"SEAMW"	ELSID1	PFTGID1	NDSID1	WELD1	TYPE1			
		ELSID2	PFTGID2	NDSID2	WELD2	TYPE2			
		-etc.-							
	"XELSET"	XELSID1	XELSID2	XELSID3	XELSID4	XELSID5	XELSID6	XELSID7	
		XELSID8	...	...	...	...	...	...	
		-etc.-							

Format (SOL 108/112):

1	2	3	4	5	6	7	8	9	10
FTGDEF	ID	TOPRMS							
	"ELSET"	ELSID1		ELSID2		ELSID3			
		ELSID4		...		...			
		-etc.-							
	"XELSET"	XELSID1	XELSID2	XELSID3	XELSID4	XELSID5	XELSID6	XELSID7	
		XELSID8	...	...	...	...	...	...	
		-etc.-							

Examples:

FTGDEF	22	100.0	3						
--------	----	-------	---	--	--	--	--	--	--

FTGDEF	22								
	ELSET	14	3	15	4				

FTGDEF	22								
	SPOTW	44	1	45	2	46			

FTGDEF	22								
	SEAMW	44	99	31	FILLET	TOE			
		45	99		LASER	ROOT			

Describer	Meaning
ID	Unique identification number called out by the FATIGUE case control. (Integer > 0).
TOPSTR	<b>SOL 101, 103, 112:</b> Top stress percentage. Retains and reports only the percentage of entities with the highest simple stress range. Cannot be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank. (-100.0 ≤ Real ≤ 100.0; Default = blank - 100% will be used). See Remark 8.
TOPRMS	<b>SOL 108, 111:</b> Top RMS (root mean square) stress percentage. Only entities with RMS stress levels in this top percentage are retained and report results. (0.0 < Real ≤ 100.0; Default = blank - 100% will be used)
PFTGID	<b>SOL 101, 103, 112 only.</b> ID of a PFTG entry for associating fatigue properties to all elements of the model. Ignored if ELSET flag is present and should be left blank in this case. (Optional, Integer > 0). See Remark 2.
TOPDMG	<b>SOL 101, 103, 112 only.</b> Top damage percentage. Retains and reports only the percentage of entities with the highest damage. (0.0 < Real ≤ 100.0; Default = blank). Cannot be used with SOL 200 or for fatigue analysis of spot and seam welds; leave blank. See Remark 8.
NENTS	<b>SOL 101, 103, 112 only.</b> Specifies the number of actual entities to output in order to limit or further reduce output file sizes based on most damage or maximum stress/strain range. A positive value indicates the criterion for entity filtering is based on the most damaged entities. A negative number indicates the filtering criteria is based on maximum stress/strain range. (Integer != 0; Default = blank). Not valid for SOL 200; leave blank. See Remark 10.
maxENTS	<b>SOL 101, 103, 112 only.</b> This is used only if STROUT=4 is specified in the FATIGUE case control. If the number of entities requested is greater than maxENTS, the job stops to avoid performance issues. Setting this to a large number can severely affect performance. (Integer > 0; Default = 100). See Remark 11.
NHS	<b>SOL 101, 103, 112 only.</b> Specifies the number of hot spots to identify in the model. (Integer > 0; Default = blank). This setting overrides TOPSTR and/or TOPDMG. See Remark 8.
HSGATE	<b>SOL 101, 103, 112 only.</b> Number of element layers to keep around each hot spot defined by NHS. Zero (0) indicates that only the hot spot elements are retained. Otherwise HSGATE layers of connected elements around each hot spot element are retained. (Integer ≥ 0; Default = 0).
ELSET	Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for consideration in the fatigue analysis. (Optional, Character = ELSET) See Remark 9.

Describer	Meaning
ELSDi	ID of a SET1, SET3, or SET4 entry listing entities of the model (elements) to be included in the fatigue analysis. (Integer > 0). See Remark 3.
PFTGi	ID of a PFTG entry, which indicates the fatigue property associated to the preceding entities defined by ELSDi. (Optional, Integer > 0). See Remark 2.
SPOTW	<b>SOL 101, 103, 112</b> only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of spot welds. See Remark 5. and 6.
ELSDi	Same as ELSDi under ELSET above. See Remark 3.
PFTGi	Same as PFTGi under ELSET above.
SEAMW	<b>SOL 101, 103, 112</b> only. Flag indicating that a list of element set and property pairs will follow, defining the elements and their associated properties for fatigue analysis of seam welds. See Remark 5.
ELSDi	Same as ELSDi under ELSET above. These are to elements that make up the seam weld toe, root, or throat. See Remark 3.
PFTGi	Same as PFTGi under ELSET above.
NDSi	ID of a SET1 or SET3 entry listing grids of the elements defined by ELSDi to be retained in the analysis. These grids define the seam line of the seam weld. If left blank, all nodes of the elements are retained. Nodes defined that are not part of ELSDi are ignored. (Optional, Integer >0).
WELDi	Seam weld definition. One of the following: FILLET, OVERLAP, LASER, EDGE, or GENERIC, which define either a fillet, overlap, laser overlap, laser edge overlap or generic seam weld, respectively. (Character; Default=GENERIC). See Remark 7.
TYPEi	The type location on the seam weld that this set of elements represent. One of the following: TOE, ROOT, or THROAT. (Character; Default = TOE for all but WELDi = LASER where Default = ROOT). See Remark 7.
XELSET	Flag indicating that sets of elements to be excluded from the fatigue analysis will follow. (Optional, Character). See Remark 4.
XELSDi	ID of a SET1 or SET3 entry listing elements of the model to be excluded from the fatigue analysis. (Integer>0). See Remark 3.

Remarks:

1. **FTGDEF** bulk data entries are ignored if not selected by a **FATIGUE** case control. If no **FTGDEF** is present for a given fatigue analysis (or only the **FTGDEF** with no **ELSET** line), all elements of the model that have fatigue material properties defined are used with default properties except for fatigue analysis of spot and seam welds. If a mix of standard SN/εN and seam/spot weld analyses is requested, then an **ELSET** line is required, otherwise the standard SN/εN analysis is ignored. See Remark 5. below.
2. If no **PFTGi** or **PTFGi** is specified, default properties are assigned to the entities.

3. If a SET3 is specified, field 3 of the SET3 entry must be set to "ELEM". The SET4 entry must be specified to select elements by property ID. The following elements referenced by PSHELL, PSHEAR, and PSOLID properties are supported for standard S-N and  $\epsilon$ -N fatigue analysis: CQUAD4, CQUADR, CQUAD8, CSHEAR, CTRIA3, CTRIAR, CTRIA6, CHEXA, CPENTA, CPYRAM and CTETRA. The following elements referenced by PBAR, PBEAM, PSOLID, and PWELD properties are supported for fatigue analysis of spot welds: CBAR, CBEAM, CHEXA, CWELD. Only elements referenced by PSHELL properties are available for fatigue analysis of seam welds, excluding TRIA3 as no corner stresses are available from this element.
4. If only the XELSET flag is present, then the entire model is included in the fatigue analysis less the excluded entities.
5. For fatigue analysis of spot or seam welds, the FTGDEF card is required with the corresponding keyword and line defining the elements of interest. If standard SN/ $\epsilon$ N analysis is also requested, then an ELSET line is required, otherwise the standard SN/ $\epsilon$ N analysis is ignored.
6. Elements used to represent spot welds are typically very stiff bar or beam (CBAR/CBEAM) elements, weld (CWELD) elements, or individual solid (CHEXA) elements. These elements connect the two metal sheets defined by shell elements (CSHELL). Bars and beams must connect directly to the grids of the shells, whereas welds and fasteners only need to pierce the shell elements. If individual solid elements are used, face G1-G2-G3-G4 must have its grids connected to shell elements that define the top sheet (sheet 1) via RBE3 rigid elements. And face G5-G6-G7-G8 must have corresponding RBE3 elements connecting the bottom sheet (sheet 2). Some CWELD options auto-generate grids on the top and bottom metal sheets (when only GS grid is defined on CWELD entry with no GA/GB grids); the fatigue results are associated to these generated grids.
7. The normals of the throat elements should point outward toward the welder, except for laser overlap, in which case the normals just need to be consistently the same direction. The elements defining the toe and root of the weld must have the top of the shell (Z2 layer) be the side where the crack is expected to develop. For full descriptions of the throat, root, and toe elements for the various seam welds, please see the *MSC Nastran Embedded Fatigue User's Guide*, which show proper modeling techniques. The WELDi and TYPEi entries are used for labeling purposes only and have no effect on internal calculations.
8. Use of TOPSTR/TOPDMG, or NHS/HSGATE allows the user to significantly speed up the analysis times of very large models and more quickly determine the critical damage locations and reduce output. The preferred method is to use NHS/HSGATE as opposed to TOPSTR and/or TOPDMG. If NHS is defined, it overrides any TOPSTR and/or TOPDMG settings, as they are mutually exclusive.
  - NHS/HSGATE: This method determines high damaged areas called *hot spots*. The algorithm is such that the *hot spots* detected are distinct from one another. In other words, if two adjacent elements show the highest damage, and NHS=2, the two adjacent element only count as one hot spot and another hot spot away from the first one is detected. In order to show nice postprocessing plots, use HSGATE to retain a certain number of element layers around each hot spot element. If HSGATE=0, only a single element for each hot spot is retained, which means that not all the highest damaged elements are likely reported.

- If TOPSTR is used alone (TOPDMG is left unspecified), and is a positive percentage, then a two (2) pass analysis is performed where the first (1st) pass determines the approximate percentage of entities to retain with the highest simple stress range. The second (2nd) pass computes damage on only the remaining elements. The simple range test is done using this equation:

$$\sigma_{assessed} = \sum_k \frac{(Max_k - Min_k) \cdot SCALE_k \cdot \sigma_{vonMises, k}}{LDM_k}$$

where  $Max_k$  and  $Min_k$  are the maximum and minimum values from each channel of loading history (load case  $k$ ). SCALE and LDM are the scale factor and divisor as defined on FTGLOAD entries for each load case  $k$ , otherwise unity is used.

The calculation is similar for strains, except that the output is in strain units, and the calculation of vonMises strain requires a value of the Poisson ratio. For the purposes of this simple ranking procedure, the Poisson ratio is assumed to be 0.3. When the analysis involves a duty cycle (loading sequence), the resulting assessed stress is the largest value from each of the events in the duty cycle.

If a negative percentage is given, then the retained elements are determined based on the actual stress range after combination and superposition using that specified on the FTGPARM entry. As an example, if TOPSTR=10%, and the highest stress range is determined to be 1000MPa, then any entity with stress range higher than 900MPa is retained (10% of 1000 is 100; 1000-100=900). This can have the effect of retaining many less entities than the positive percentage method, but has the drawback of being extremely sensitive to high stress gradients.

- If TOPDMG is used alone (TOPSTR=100% or left unspecified), the percentage of entities returned are simply based on the highest damage. This does not necessarily reduce computation time, but does significantly reduce result recovery time as less entities are retained.
  - If TOPSTR and TOPDMG are both specified, then a 2-pass analysis is performed where the first (1st) pass eliminates all but the highest stressed entities and the second (2nd) pass returns only the percentage of damaged entities based on those retained in the 1st pass.
  - If none of TOPSTR, TOPDMG, or NHS are specified (all are blank or default), then a complete analysis is done on all requested elements. This is the default scenario.
9. Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses through the FTGPARM entry (SRESOLVE field = YES). Otherwise a fatal condition is flagged and the analysis stops.
  10. This NENTS parameter can be used by itself or in conjunctions with TOPSTR, TOPDMG, or NHS. Only entities that pass this filter are reported in the f06 file and/or output to the OUTPUT2 and/or HDF5 files. If NENTS is used with the SRESOLVE option on the FTGPARM entry, then the number of entities printed may be more as the SRESOLVE option saves the number of grids and each element associated with those grids.
  11. If FATIGUE(STROUT=4) case control is used it is highly recommended that output be limited. Temporary CSV files are created for each entity and F06 output can be enormous if not filtered using the ELSET entries. The analysis fails if maxENTS is less than the number of elements requested using ELSET entries when STROUT=4 is specified. Increase maxENTS if necessary after assessing output needs.



## FTGEVNT

## Fatigue Loading Events

Groups simultaneously applied loads into loading events for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random vibration fatigue using SOL 108 or SOL 111 by referencing FTGLOAD entries.

Format (SOL 101, 103, 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGEVNT	ID	FLOAD1	FLOAD2	FLOAD3	FLOAD4	FLOAD5	FLOAD6	FLOAD7	
		FLOAD8	...	...	...	...	...	...	
		-etc-							
	"NAME"	EVNTNAM							

Example:

FTGEVNT	22	4	11						
	NAME	rough_road							

Descriptor	Meaning
ID	Unique ID. See Remark 1.
FLOADi	ID of a FTGLOAD entry (Integer > 0, no default). See Remark 5.
NAME	Flag indicating that a name is to be associated with this load event. See Remark 2.
EVNTNAM	Event name associated with this event (Character). Can span fields 3 through 9, however for practical purposes, the name is truncated in f06 output. No spaces are allowed. See Remark 3. and 4.

Remarks:

- Each FTGEVNT ID must be unique relative to all other FTGEVNT and FTGSEQ IDs.
- Event names are passed to the f06 print file and displayed as the name of the event as opposed to the event ID. If no name is given, the event is referred to by its ID.
- EVNTNAM cannot have numbers at the start of any field. For example, using 8 character formatted fields, the first character of fields 3-9 cannot be numeric (columns 1, 9, 17, etc. of the EVNTNAM field). These would be acceptable:  
 "Flights" or "F\_l\_i\_g\_h\_t\_s".  
 But these would not be acceptable:  
 "1\_Flights" or "F\_l\_i\_g\_2ts"  
 because the "1" and the "2" are in columns 1 and 9 of the EVNTNAM field.

4. EVNTNAM cannot handle spaces within its field as one might expect. If you wish to use spaces, make sure that once a space is used the next character does not start until the start of the next field. For example, using 8 character formatted fields, the following would be acceptable:  
“My\_flights” or “My Flights”.  
But this would not be acceptable:  
“My Flights”  
and would end up just being displayed as “Myts” because once a space is encountered within the 8 character field, the code interprets that as no more character data until it processes the next 8 character field. It is best not to use spaces in EVNTNAM.
5. For random vibration fatigue (SOL 108 and 111),
- Only one FTGLOAD of TYPE=PSD can be referenced
  - Only one FTGLOAD of TYPE=STATIC can be referenced
  - A FTGEVNT referencing a FTGLOAD of TYPE=STATIC by itself is not allowed
  - A FTGEVNT referencing a FTGLOAD of TYPE=SINE or NARROW by itself (or themselves) is allowed for pure deterministic and/or sine sweeps
  - Only single load PSD random input allows mixed deterministic, sine sweep, and static offsets on random loading. Only a static offset (FTGLOAD of TYPE=STATIC) can be mixed with multi-load PSD random input.
  - FTGSEQ entry referencing FTGEVNT entries cannot mix events containing single load and multi-load PSDs.
  - All multi-load PSD events must all have the same number of channels (input loads).
  - When only FTGLOAD entries of TYPE=SINE or NARROW are referenced (i.e., only deterministic loading with no random loading input), it is recommended that the ATYPE field of the FTGPARM entry be set to SINE.

## FTGLOAD

### Fatigue Cyclic Loading Variation

Defines cyclic loading variation for pseudo-static fatigue analysis using SOL 101 or modal analysis using SOL 103 or random PSD, deterministic, sine sweep and/or static offset loading for SOL 108 and SOL 111.

Format (SOL 101 and 103):

1	2	3	4	5	6	7	8	9	10
FTGLOAD	ID	TID	LCID	LDM	SCALE/ MAX	OFFSET/ MIN	TYPE	CHNL	
	"UNITS"	EQUIV	EQNAME						

Format (SOL 108 and 111):

1	2	3	4	5	6	7	8	9	10
FTGLOAD	ID	TID	LCID		SCALE	OFFSET	TYPE		
	"DETLOAD"	F1	G1/H1	B1	F2	G2/H2	B2		
		F3	G3/H3	B3	...	...	...		
		-etc.-							
	"SWEEP"	F1	G1/H1		F <sub>n</sub>	G <sub>n</sub> /H <sub>n</sub>			
		SWTYPE	SWNUM	SWRATE	NSWEEP	FRFUNIT	LDUNIT		

Examples:

FTGLOAD	55	4	2	1.0	1.0	0.0			
FTGLOAD	23	4	5	1.0	1.0	0.0	DB		
	UNITS	5.5	Flights						

Describer	Meaning
ID	Unique ID which is referenced by a FTGEVNT entry or directly by a FATIGUE case control in the case of SOL 101 only for a single load (Integer > 0). See Remark 1.
TID	<ul style="list-style-type: none"> <li>■ <b>SOL 101, 103:</b> Table ID of a TABLFTG (or TABLED1) entry that defines the time variation of the loading or the ID of a UDNAME entry for external definition of the loading time variation (Integer &gt; 0). See Remarks 2. through 6. Ignored if TYPE=CONST or STATIC.</li> <li>■ <b>SOL 108, 111:</b> leave blank unless: <ul style="list-style-type: none"> <li>■ TYPE=STATIC: UDNAME ID referencing an external OP2 file containing stress results for static loading offsets. Leave blank (and set LCID = 0 if a constant offset defined solely by the OFFSET field is desired).</li> <li>■ TYPE = PSD with TIM2PSD: UDNAME ID referencing an external RPC or CSV file when time history to PSD conversion is requested. A TIM2PSD entry must be present in this case, otherwise TID is ignored.</li> </ul> </li> </ul>
LCID	<ul style="list-style-type: none"> <li>■ <b>SOL 101:</b> Subcase ID of a linear static solution SUBCASE (Integer &gt; 0, no default).</li> <li>■ <b>SOL 103:</b> Mode number for SOL 103 (Integer &gt; 0, no default).</li> <li>■ <b>SOL 108, 111:</b> <ul style="list-style-type: none"> <li>■ TYPE = PSD with no TIM2PSD: SID of RANDPS entry(ies). TID should be left blank in this case.</li> <li>■ TYPE = PSD with TIM2PSD: -1 for fully correlated multi-input loading, -2 for uncorrelated multi-input loading.</li> <li>■ TYPE = STATIC: SUBCASE ID of a static SUBCASE (freq=0.0). A TID must also be defined in this case to point to an external OP2 file. Set to zero (0) if a constant offset solely defined by the OFFSET field is desired. TID should be left blank in this case.</li> <li>■ TYPE=SINE or NARROW: SUBCASE ID of transfer function (TF) used for deterministic or sweep loading. TID should be left blank in this case.</li> </ul> </li> </ul>
LDM	<b>SOL 101, 103 only.</b> Largest magnitude of the applied load (in the same units used to define the load time variation in field TID) used to normalize the load (Real, default=1.0). This effectively scales the stress to simulate a stress state due to a unit load. See Remark 7.
SCALE/ MAX	<p><b>SOL 101, 103:</b> Scale factor applied to the load time history (Real, default=1.0). See Remark 7. Or the 1st peak level of a constant amplitude signal if TYPE=CONST (Real, default=1.0).</p> <p><b>SOL 108, 111:</b> Acts as an additional scale factor on the input PSD. This applies only to TYPE=PSD (Real, default=1.0).</p>

Describer	Meaning
OFFSET/ MIN	<p><b>SOL 101, 103:</b> Offset applied to the load time history (Real, default=0.0). See Remark 7. Or the 2nd peak level of a constant amplitude signal if TYPE=CONST (Real, default = -1.0). MAX can be &lt; or &gt; than MIN for TYPE=CONST.</p> <p><b>SOL 108, 111:</b> Offset of resultant stress PSD in stress units. This has the effect of adding a mean stress effect to the random load response. This only applies to TYPE=STATIC and if TID is blank, this is only a constant offset. With TID a variable offset plus this constant offset is applied (Real, default = 1.0).</p>
TYPE	Flag indicating the type of load being defined. Values can be: blank, "DB", "DAC", "RPC", "CONST", "PSD", "SINE", "NARROW" or "STATIC". Default is blank. See Remarks 2. through 6.
CHNL	<b>SOL 101 and 103 only.</b> Channel of referenced RPC file (TYPE=RPC). (Integer >0, Default is blank). See Remark 12.
"UNITS"	<b>SOL 101 and 103 only.</b> Flag indicating that a fatigue equivalent unit name is defined for this loading. See Remarks 9. and 10.
EQUIV	Number of equivalent units (Real>0.0, default=1.0). See Remarks 9. and 10.
EQNAME	Equivalent name of this loading history. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remarks 9. and 10.
"DETLOAD"	<p><b>SOL 108 and 111 only.</b> Flag indicating that deterministic load input is to follow for TYPE=SINE or NARROW.</p> <p>Fi Frequency of sine wave for TYPE=SINE or center frequency of narrow band frequency block for TYPE=NARROW. (Real&gt;0.0; no Default).</p> <p>Gi Height of sine wave (Gi) in units of peak amplitude (FE stress units) for TYPE=SINE.</p> <p>Hi Height of narrow band PSD block (Hi) in units of (stress)<sup>2</sup>/Hz for TYPE=NARROW. (Real&gt;=0.0; no Default).</p> <p>Bi Width of narrow band frequency block. Only supplied for TYPE=NARROW. (Real&gt;=0.0; no Default).</p>
"SWEEP"	<p><b>SOL 108 and 111 only.</b> Flag indicating that sine sweep load input is to follow for TYPE=SINE (not supported for NARROW).</p> <p>F1/Fn F1/Fn Frequency of 1st and last sine waves for TYPE=SINE. (Real&gt;0.0; no Default).</p> <p>G1/H1 Height of 1st (G1) and last (Gn) sine waves in units of peak amplitude (FE stress units) for TYPE=SINE. (Real&gt;=0.0; no Default).</p> <p>Gn/Hn</p> <p>SWTYPE The sweep type: Decibel (DB), Octave (OCT), or Linear (HZ). (Character; Default = HZ)</p> <p>SWNUM Number of sine/narrow bands to use (Integer&gt;0 where N=SWNUM+1; Default = 50)</p> <p>SWRATE Sweep rate per second in HZ (default), DB or OCT. (Real&gt;0.0; No Default)</p> <p>NSWEEP Number of sweep passes (Integer&gt;0; No Default).</p>

Descriptor	Meaning
FRFUNIT	Units of loading used to create FRF - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A)
LDUNIT	Units of loading used to define sweep - can be acceleration (A), velocity (V), displacement (D), or force (F). Used for TYPE=SINE only. (Character; Default = A)

Remarks:

1. If a FATIGUE case control command invokes an ID that is present on both an FTGSEQ entry and a FTGLOAD entry, the FTGSEQ request will be honored and the FTGLOAD entry will be ignored unless it is referenced on a FTGEVNT entry.
2. **For SOL 101 and 103**, if the TYPE field is blank then TID references a TABLFTG or TABLED1 entry. For SOL 108 or 111, TYPE must be supplied.
3. **For SOL 101 and 103**, if TYPE=DB, DAC or RPC, then TID references a UDNAME ID. A UDNAME entry must be supplied in this case to specify the file and path of the externally defined load time variation. This file is expected to be in standard DAC file format or channel data in the form of a RPC file. TYPE=DB is the same as TYPE=DAC. You cannot mix TYPE=RPC designations with any other TYPEs within the same fatigue analysis (FATIGUE=FID). If RPC is used, all FTGLOAD entries must use TYPE=RPC. The same RPC file must be referenced via UDNAME for all FTGLOADs of the same event (FTGEVNT). **For SOL 108 and 111**, if TYPE=STATIC or if TYPE=PSD with TIM2PSD entry active, the UDNAME references an external OP2 file or RPC/CSV channel data file, respectively.
4. **For SOL 101 and 103**, if TYPE is STATIC, the TID field should be left blank as it will be ignored. STATIC indicates that the stress state from the specified LCID is to act as a static offset with no load time variation when performing the linear superposition, which will give every element a different offset defined by the stress state at each element of the specified subcase, as opposed to simply specifying the OFFSET field, which gives every element the same offset. If the "STATIC" flag is specified, there must be at least two FTGLOAD entries defined and called out by a FTGEVNT entry, one of which must be time varying (see Remark 7.). **For SOL 108 and 111**, if TYPE is STATIC, the TID field must be supplied.
5. **For SOL 101 and 103**, if TYPE is CONST, the TID, LDM, and CHNL fields are ignored. The MAX and MIN fields are used to define the maximum and minimum values of a constant amplitude cycle. TYPE=CONST cannot be mixed with any other TYPE values for a given fatigue analysis (FATIGUE=FID). All loading must be of TYPE=CONST. This is sometimes referred to as block loading.
6. For modal analysis using **SOL 103**, the referenced load variation defines the modal participation factors for the referenced mode.
7. **For SOL 101 and 103**, the LDM, SCALE, and OFFSET are used together in the following manner to scale/modify the stress state in order to determine the resulting stress time variation:

$$\sigma_{ij}(t) = \left( P(t) \times \frac{\sigma_{ij,l}}{LDM} \times SCALE \right) + OFFSET$$

8. where  $\sigma_{ij}(t)$  is the resulting stress tensor at time  $t$ ,  $\sigma_{ij, l}$  is the stress tensor from the subcase or mode defined by the LCID field, and  $P(t)$  is the  $y$  value of the load-time history at time  $t$  as defined by the TID field. For multiple loads, the principle of linear superposition is used to combine all loads for a single event. For SOL 108 and 111, LDM and SCALE act as a divisor and multiplier, respectively, to the defined loading. OFFSET is only used for TYPE=STATIC and acts as an additional constant offset over all entities.
9. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the stress time history. If this FTGLOAD is referenced by a FTGEVNT, then the equivalent units on this entry are ignored and those on the FTGSEQ entry take precedent. Only if a FTGLOAD is directly referenced by a FATIGUE case control are the fatigue equivalent units used as defined on the FTGLOAD entry.
10. Example of using equivalent units: If one repeat of the defined time history is equivalent to 5 times around a test track, the equivalent unit name, EQNAME, might be "*laps*," and the equivalent unit, EQUIV, would be 5.0. Fatigue life will be reported in these units if defined, otherwise they are reported as *Repeats* of the loading. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
11. All FTGLOAD entries referenced by a FTGEVNT should reference different SUBCASEs for SOL 101 (or modes for SOL 103) and must have time variations consisting of the same number of points.
12. CHNL is only used for TYPE=RPC for SOL 101 and 103. If supplied, the specified channel of the referenced RPC file is used. If it is left blank, the next available channel sequentially from the last one referenced will be used. For example, if there are three FTGLOAD entries for a specific event and CHNL is blank for all three, the 1st one will use channel 1, the 2nd one will use channel 2 and the 3rd will use channel 3. If in this example the 1st specifies CHNL=11 and the others are blank, then the channels used will be 11, 12, and 13. If the 1st is left blank and the 2nd references CHNL=12, then the channels used will be 1, 12, and 13.

## FTGPARM

## Fatigue Parameters

Defines parameters for a fatigue analysis in time domain SOLs 101, 103, and 112 and in frequency domain SOLs 108 and 111.

Format (SOLs 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
FTGPARM	ID	TYPE	FACTOR	NTHRD	LOGLVL	LAYER			
	"STRESS" or "STRAIN"	COMB	CORR	PLAST	LOC	INTERP	RECOVER	SRESOLVE	
		NANGLE							
	"CERTNTY"	SURV							
	"FOS"	OPTION	LIFE	BACKACC	MAXFAC	MINFAC			
	"SPOTW"	COMB	CORR	NANGLE	SWLOC	MIDDLE	TORSION		
	"SEAMW"	COMB	CORR	THICK	LOCSM	RESENT			
	"MULTI"	MMTHD	NONLWR	NONUPR	BIAXLWR	BIAXMID	BIAXUPR	ZERO	
		GATE							
	"NAVG"	MTHD	OUTPUT	NORMAL					

Format (SOLs 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGPARM	ID	TYPE			LOGLVL	LAYER			
	"STRESS"	COMB	CORR	PLAST	LOC		RECOVER		
	"FOS"		LIFE						
	"VIBFTG"	ATYPE	MAXSTR	CLIPLVL	MAXPEAK	STRBINS	MAXFREQ	NCALC	

Examples:

FTGPARM	22	EN							
---------	----	----	--	--	--	--	--	--	--

FTGPARM	22	SN							
	STRESS	SGVON	NONE		NODA		CUBIC		

FTGPARM	22	SN							
	SEAMW		FKM	0					

FTGPARM	22	SN							
	SPOTW	STNDRD	SIMPLE	18	3	0	0		



FTGPARM	22	EN							
	MULTI	AUTO	0.25	0.5	-0.6	0.25	0.6	10.0	
		20.0							

Describer	Meaning
ID	Unique ID of the FTGPARM entry called out by a FATIGUE case control (Integer > 0). See Remark 1.
TYPE	<p><b>SOL 101, 103, 112:</b> Type of fatigue analysis: "SN" or "EN" (Character, Default=SN).</p> <p><b>SOL 108, 111:</b> Type of fatigue analysis: "SN" or "EN" (Character, Default=SN). This is used only when there is a conflict with the defined material properties, i.e., both S-N and <math>\epsilon</math>-N properties exist for the same material MID. Otherwise it is ignored and the material properties define the type of fatigue analysis to be performed..</p>
FACTOR	<b>SOL 101, 103, 112</b> only. Global scale factor to be applied to combined resultant stress output (Real>0.0, Default=1.0).
NTHRD	<b>SOL 101, 103, 112</b> only. Number of threads to use for parallel processing for this fatigue analysis (Integer $\geq$ 0; Default = Blank). Blank denotes that the number of threads used is based on SMP setting and the existence of an SMP license. Zero (0) is used as a flag to tell the code to use all available threads. Otherwise the number of threads designated is used if available. An SMP license is only necessary to take advantage of multiple treads if left blank.
LOGLVL	<p><b>SOL 101, 103, 112:</b> Level of messaging sent to the log file (Integer <math>\geq</math> 0, Default = 0; 0=None, 1=Error, 2=Info, 3=Low, 4=Medium, 5=High). Note that setting LOGLVL &gt;0 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest. LOGLVL=5 produces an HDF5 or a .dtout file that shows all parameter settings. This file can be significantly large.</p> <p><b>SOL 108, 111:</b> Level of output to generate (Integer <math>\geq</math> 0, Default = 0; 0=Standard, 1=Output all PSD responses, 2=Additionally output all Rainflow data). Note that LOGLVL &gt;0 can result in a significant performance penalty and should be used for debugging purposes only and limited to the entities of interest only. See Remark 13.</p>
LAYER	For shell elements, the output results layer to print to the f06 file. Values can be 0=Worst, 1=Top(Z2), 2=Bottom(Z1). (Integer, Default = 0). This is for printed output only. The analysis produces results for both layers, which are always available through the MASTER/DBALL, Output2, or other files for graphical postprocessors. See the <a href="#">FATIGUE</a> case control entry.
"STRESS" or	Flag indicating that stress is used in the fatigue calculation. See Remark 2.

Describer	Meaning
"STRAIN"	SOL 101, 103, 112 only. Flag indicating that strain is used in the fatigue calculation. Not valid for TYPE = SN. See Remark 2.
COMB	Stress/strain combination to use in the fatigue analysis. Acceptable values are listed in Table 9-17 after the Remarks below (Character; Default t= ABSMAXPR).
CORR	Mean stress correction to use in the fatigue analysis. Acceptable values are listed in Table 9-18 and Table 9-19 after the Remarks below (Character; Default=None).
PLAST	SOL 101, 103, 112: Plasticity correction for TYPE = EN. Value can currently only be "NEUBER" for Neuber method. (Character; Default=NEUBER). See Remark 3.
LOC	SOL 108, 111: Only NEUBER correction is available. SOL 101, 103, 112: Location to report fatigue lives. Valid values are "NODA", "NODE", or "ELEM" based on nodal averaged, element nodal, or element centroid stresses or strains, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4. Also see the "NAVG" field and parameters.
INTERP	SOL 108, 111: Location to report fatigue lives. Valid values are "NODE", "NODA", or "ELEM" based on element nodal, nodal averaged, or element center stresses, respectively. (Character; Default = NODA). See the RECOVER field and Remark 4. SOL 101, 103, 112 only. Interpolation limit for multi-curve mean stress correction method (Integer $\geq 0$ ; 0=Use Max Curve, 1=Extrapolate; Default = 0).
RECOVER	Stress recovery method to determine stresses/strains for the fatigue calculation. Valid values are "SGAGE", "CORNER", "BILIN", "CUBIC" or "CENTER" (Character; Default = CORNER if LOC=NODA or NODE; Default = CENTER if LOC=ELEM and any other setting is ignored). These are the same values used for the STRESS (or STRAIN) case control output request. See LOC field and Remark 4.
SRESOLVE	SOL 101, 103, 112 only. Request for surface resolved, 2D stress state. Used only for 3D solid elements. (Character YES or NO; Default = NO). See Remark 5.

Describer	Meaning
NANGLE	The number of calculation angles for 2D critical plane analysis (COMB=CRITICAL) ( $1 \leq \text{Integer} \leq 360$ ; Default = 36, i.e. every 10 degrees). Practical values are 360, 180, 120, 90, 72, 60, 45, 40, 36, 30, 24, 18, 15, 12, 10, 9, 8, 6, 5, 4 for every 1, 2, 3, 4, 5, 6, 8, 9, 10, 12, 15, 20, 24, 30, 36, 40, 45, 60, 72, 90 degrees, respectively. The more angles computed, the more compute intensive the analysis will be. In actuality, only half the angles are computed since the principals are the same 180 degrees opposite any computed angle.
"CERTNTY"	SOL 101, 103, 112 only. Flag indicating that the parameter that follows defines the certainty of survival in fatigue analysis.
SURV	Certainty of survival based on the scatter in the S-N or e-N curves. ( $0.1 \leq \text{Real} \leq 99.9$ ; Default = 50). See Remark 6.
"FOS"	SOL 101, 103, 112: Flag indicating that parameters that follow are used in a factor of safety analysis. The presence of this flag triggers a factor of safety analysis. See Remark 7. and 11.
	SOL 108, 111: Flag indicating that the total signal time duration desired is to be specified for use in calculating margin of safety. Only the LIFE field is required in this case and defaults to 1.0. A margin of safety is always calculated.
OPTION	Supported option is LIFE, requesting a life-based factor of safety analysis or NONE. (Character; default = LIFE).
LIFE	SOL 101, 103, 112: The targeted design life given in user defined life units (such as laps, miles, etc.) as defined by "UNITS" line on FTGSEQ or FTGLOAD entry, or <i>Repeats</i> of the cyclic loading if no "UNITS" line exists (Real>0, no default).
	SOL 108, 111: Total time duration required for margin of safety calculation in time units specified by TUNIT field on FTGSEQ entry.
BACKACC	SOL 101, 103, 112 only. The back calculation accuracy used to control back calculation iterations that determine the scale factor on the applied stress level to achieve the target design life. Defined as a percentage error on the target design life. ( $0.01 < \text{Real} \leq 100.0$ , Default = 1.0).
MAXFAC	SOL 101, 103, 112 only. The maximum safety factor to calculate and report. When this threshold is exceeded, the analysis will go on to the next element and report the maximum for the exceeded element ( $2.0 \leq \text{Real} \leq 5.e6$ , Default = 5.0).
MINFAC	SOL 101, 103, 112 only. The minimum safety factor to calculate and report. If the result is below this threshold, the analysis will report MINFAC as the safety factor for this element and go on to process the next element. ( $0.0 < \text{Real} \leq 0.5$ , Default = 0.2).

Describer	Meaning
"SPOTW"	SOL 101, 103, 112 only. Flag indicating that parameters for fatigue analysis of spot welds are to follow. See Remark 8.
COMB	Stress combination to use in the fatigue analysis. Acceptable values are listed in Table 9-17 after the Remarks below (Character; Default=STNDRD, which is basically a critical plane analysis).
CORR	Mean stress correction to use in the fatigue analysis of spot welds. Only NONE, FKM, or SIMPLE are valid for fatigue analysis of spot welds. See Table 9-18 and Table 9-19 below. (Character; Default = NONE).
NANGLE	The number of calculation angles in 360 degrees around the spot weld. ( $10 \leq \text{Integer} \leq 360$ ; Default = 18, i.e., every 20 degrees)
SWLOC	Location on the spot welds to report fatigue life. Zero (0) reports worst case angle and location (top/bottom sheet or nugget); one (1) reports worst case angle for each location; two (2) reports worst case location for each angle; three (3) reports all locations and angles. (Integer 0, 1, 2, or 3, Default=0).
MIDDLE	Whether to process middle sheets if there are more than two sheets in the weld. (Integer 0 or 1, Default = 1- process middle sheets). See Remark 10.
"SEAMW"	SOL 101, 103, 112 only. Flag indicating that parameters for fatigue analysis of seam welds are to follow. See Remark 8.
COMB	Stress/strain combination to use in the fatigue analysis of seam welds. Only ABSMAXPR and CRITICAL are supported. See Table 9-17 below. (Character; Default = ABSMAXPR).
CORR	Mean stress correction to use in the fatigue analysis of seam welds. Only NONE or FKM are valid for fatigue analysis of seam welds. See Table 9-18 and Table 9-19 below. (Character; Default=NONE).
THICK	Thickness correction to be applied. (Integer 0 or 1; Default =0 - no correction)
LOCSM	Location on seam weld to report life. Valid values are "NODE", "SGAGE", "CORNER", "BILIN", or "CUBIC" (Character; Default = NODE). If "STRESS" line is also include, LOCSM must be the same as LOC. LOC=ELEM is not valid for fatigue analysis of seam welds and cannot be mixed with LOC = NODE. See Remark 4.
RESENT	Result entity type used in the fatigue analysis of seam welds; only STRESS is currently supported. (Character; Default = STRESS).
"MULTI"	SOL 101, 103, 112 only. Flag indicating that parameters for biaxial/multiaxial assessment are to follow. See Remark 9. and 11.

Describer	Meaning
MMTHD	<p>Can be set to the following:</p> <ul style="list-style-type: none"> <li>■ NONE = No multiaxial assessment is done.</li> <li>■ SIMPLE = Calculates simple biaxiality ratios only.</li> <li>■ STANDard =Standard method of assessment, which merely returns the results of the assessment.</li> <li>■ AUTO = Performs the standard method, but then may recalculate fatigue damage depending on the results of the assessment.</li> </ul>
NONLWR NONUPR	<p>Used only when MMTHD = AUTO. Lower and Upper thresholds used to check if the loading is proportional. This is used in combination with the biaxiality ratio thresholds.</p> <p>NONLWR: (<math>0 &lt; \text{Real} \leq \text{NONUPR}</math>, Default = 0.25)  NONUPR: (<math>\text{NONLWR} &lt; \text{Real}</math>, Default = 0.5)  NONUPR not used if TYPE=SN</p>
BIAXLWR BIAXMID BIAXUPR	<p>Used only when MMTHD = AUTO. Lower, Middle, and Upper threshold ratios used to check if the loading is proportional. This is used in combination with the non-proportionality factor thresholds.</p> <p>For TYPE=EN:  BIAXLWR: (<math>-1 \leq \text{Real} \leq \text{BIAXMID}</math>, Default = -0.6)  BIAXMID: (<math>\text{BIAXLWR} \leq \text{Real} \leq \text{BIAXUPR}</math>, Default = 0.25)  BIAXUPR: (<math>\text{BIAXMID} \leq \text{Real} \leq 1</math>, Default = 0.6)</p> <p>For TYPE=SN:  BIAXLWR: (<math>-1 \leq \text{Real} \leq 0</math>, Default = -0.6)  BIAXMID: Not used  BIAXUPR: (<math>0 &lt; \text{Real} \leq 1</math>, Default = 0.6)</p>
ZERO	<p>Used only when MMTHD = AUTO. Stress range below which damage is assumed zero and therefore no recalculation is performed. Specified as a percentage of UTS. (<math>0 &lt; \text{Real} \leq 100</math>, Default=10.0).</p>
GATE	<p>Used only when MMTHD = AUTO. This gate value is used to prevent small stresses from adversely affecting the biaxiality calculation. Stresses below this value are not included in the biaxiality calculation; their biaxiality ratio will be set to zero (0). The value is set as a percentage of UTS. (<math>0 &lt; \text{Real} \leq 100</math>, Default=20.0).</p>

Describer	Meaning
"NAVG"	<p>SOL 101, 103, 112 only. Flag indicating that parameters for nodal (grid point) stress (or strain) averaging are to follow. The LOC field must be set to NODA, otherwise this line is ignored. These parameters are the same as those specified in the SURFACE and VOLUME case control for requesting grid point stresses/strains and the user is referred to the OUTPUT(POST) section for more details. All defaults are used if this line is omitted. See Remark 12.</p>
MTHD	<p>Method to calculate average grid point stresses/strains:</p> <ul style="list-style-type: none"> <li>■ TOPO = Topological method (default)</li> <li>■ GEOM = Geometric method.</li> </ul>
OUTPUT	<p>Coordinate system in which to translate stresses/strains before averaging:</p> <ul style="list-style-type: none"> <li>■ -1 : Specifies the element coordinate system for output</li> <li>■ 0 : Specifies the basic coordinate system for output (default)</li> <li>■ CID : Specifies the coordinate system defined on a CORDij bulk data entry for output</li> </ul> <p>The X-axis of this system is used to define the X stress direction. For shell elements the X-axis of the specified coordinate system is projected onto the surface or, that is, the resulting averaged stresses are a projection onto the surface.</p>
NORMAL	<p>For shell elements defining a surface, this specifies the method to define the surface normal. For solid elements defining a volume, this specifies the method to define the stress Z-axis. Indirectly this also sets the reference direction for positive fiber and shear stress output, but has no effect when OUTPUT=-1. Can be set to R (default), X1, X2, or X3.</p> <p>M Specifies the reverse of the direction given by R, X1, X2, or X3 and must be entered as MR, MX1, MX2, or MX3 with no space between the M and the following letter. R Specifies the radius vector from the origin of reference coordinate system to the grid point.</p> <p>The Y stress direction is the cross product of the Z and X stress directions defined by the NORMAL and the X-axis of the OUTPUT coordinate system.</p>
"VIBFTG"	<p>SOL 108, 111 only. Flag indicating that vibration fatigue parameters follow. The line is optional and defaults are used if not present.</p>
ATYPE	<p>Analysis method to use to generate the PDF of rainflow ranges: DIRLIK, NARROW, STEIN, SINES. (Character; default = DIRLIK)</p>

Describer	Meaning
MAXSTR	Maximum stress to use as a function of the number of RMS stress levels to determine maximum stress in the rainflow cycle count. (Real>0.0; Default=10.0).
CLIPLVL	Value of stress where all stresses are "clipped" (kept at that level) when doing fatigue life calculation. Real>0.0; (Default CLIPLVL=MAXSTR). CLIPLVL=6 means +/- 3*RMS amplitudes or 6*RMS in terms of range.
MAXPEAK	Used to calculate an estimation of maximum and minimum stress (number of RMS levels) in the random response. It is not used in any way to calculate damage or life. This is also used when the maximum elastic-plastic peak strain is calculated. Where a mean load is included the peak stress is calculated as the mean_stress + MAXPEAK*RMS and the minimum stress is calculated as the mean_stress - MAXPEAK*RMS. This is similar to the time domain approach where the maximum and minimum stress are retained from the rainflow cycle count. (Real>0; Default=CLIPLVL/2 or, if CLIPLVL not defined, MAXSTR/2).
STRBINS	Number of bins to generate in rainflow cycle count. Default = 1280 for ATYPE=SINES otherwise 32. Not recommended to use anything over 5000.
MAXFREQ	Percentage of frequency content retained as a function of the first spectral moment $m_0$ of the PSD defining the maximum frequency used to integrate the spectral moments for $m_1$ , $m_2$ , and $m_4$ . $0 \leq \text{Real} \leq 100.0$ ; Default=99.9). 100% is not a recommended setting as it can cause numerical instability related to the calculation of the 4th spectral moment.
NCALC	Alternative options for the way "N" (life to failure) is calculated for a given stress bin. Valid values are MID, UPR, or AVG for middle, upper or average. (Character; Default=AVG). See <a href="#">Table 9-20</a> .

Remarks:

1. FTGPARM bulk data is ignored if not selected by a FATIGUE case control entry. If a FTGPARM entry is not defined, default properties are used for the requested fatigue analysis.
2. **SOL 101, 103, 112:** For total life or stress-life (TYPE=SN), only STRESS results can be used. For crack initiation or strain-life (TYPE=EN), the fatigue analyzer may use either STRESS or STRAIN results from the finite element analysis. This selection should make no difference to the final results of a crack initiation calculation, as strains will always be calculated. The exception is when shell results are used. In this case, STRESS should be selected because only 2D results are available and the absence of the out-of-plane strains will cause incorrect calculation of combined parameters. It is an error to have both STRESS and STRAIN lines. If both are missing, then STRESS will be assumed with its default values.
3. **SOL 101, 103, 112:** PLAST can only be set to NEUBER. Please note that NEUBER can be used universally for uniaxial stress states. PLAST is only valid for TYPE=EN.

4. If LOC=ELEM (element center), fatigue lives are calculated based on stresses/strains at element centroids (not recommended for anything but shell or 2D solid elements (plane stress/strain and axisymmetric) and the RECOVER field is ignored or automatically set to CENTER.

If LOC=NODE, the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. Note that for LOC=NODE, the RECOVER field can be set to SGAGE, BILIN, CUBIC, or CORNER to control which stresses are used in the fatigue analysis for element nodal stresses and correspond to the same options as on the STRESS case control.

If LOC is set to NODA, the same RECOVER options as LOC=NODE apply, however:

**SOL 101, 103, 112:** With LOC=NODA, fatigue lives are based on stresses/strains from nodal averaged (grid point) stresses, resulting in a single damage value at each node as opposed to using LOC=NODE where the fatigue lives are calculated from the stresses/strains at the element nodes, resulting in multiple damage values per node. The grid point stresses are generated using Nastran's GPSTRESS output request internally (it is not required to include the GPSTRESS case control unless you wish to view the grid point stress also). See the NAVG line settings also on how to control grid point stress calculations.

**SOL 108, 111:** With LOC=NODA, after the nodal transfer function (TF) stresses for each element contribution are converted to the requested stress combination (e.g. COMB=SGVON), the stresses are summed and averaged at each node. This is done at every frequency to give a single TF of stress at each node. Thus a single damage and fatigue life per node is reported as opposed to slightly different values when LOC=NODE. Please note that the coordinate system in which the elements are defined should be consistent (the same). Otherwise the nodal averaging will not be done using consistent coordinate systems. Thus, for solid element, setting CORDM on the PSOLID entry to the Basic coordinate system or a specified CID is required and setting CID=-1 or -2 (element systems) is not recommended. For shell elements, MCID should be set on the respective element entry.

This parameter cannot be changed on a RESTART since it would require a different set of stress/strain data, which is not allowed on a RESTART for a pure fatigue analysis.

LOC=NODE generally gives more conservative answers than LOC=NODA since no averaging of the stresses/strains is done from the contributions of the surrounding elements.

5. **SOL 101, 103, 112:** SRESOLVE is applicable for all three values of LOC, i.e.: LOC = ELEM, NODE or NODA. SRESOLVE is an option to evaluate surface stresses instead of volume stresses. When surface resolved stresses are requested from 3D solid elements, a thin layer of 2D shell elements is internally created encompassing the volume of the elements defined on the FTGDEF entry from which the 2D stress state is then determined. This parameter is necessary if a multiaxial assessment of a model made of 3D elements is necessary, otherwise 3D elements are ignored during a multiaxial assessment. Fatigue life and multiaxial statistics are only reported for these newly defined 2D elements (or reduced set of nodes) on the surface of the volume (free surface). Interior nodes are not included in the fatigue calculation. The internal element IDs generated by this option can be controlled by using MSC Nastran SYSTEM cell 183 (Default is set to 200000000).

Element sets must be the same for all FATIGUE case control IDs that request surface resolved stresses. Otherwise a fatal condition is flagged and the analysis stops.



6. Certainty of survival is based on scatter in the S-N or  $\epsilon$ -N curves. It is used to modify the curves according to the standard error parameters ( $SE_n$ ) defined in the MATFTG material entry. A higher reliability level requires a larger certainty of survival.
7. **SOL 101, 103, 112:** This FOS option will calculate a type of safety factor for over design analysis to be performed. This analysis is in addition to the normal fatigue life/damage output and must be requested by the presence of this FOS flag and its parameters. This analysis method can be very useful for those components which predict infinite life, providing a measure of the risk of fatigue failure. The results of this analysis are factors by which the stress would have to be scaled to attain the specified design life. A value of one suggests that the specified life will be exactly attained whereas a factor less than one means the desired life will not be attained. Factors greater than one are, therefore, most desirable. By definition the resulting life values will be the target life, thus only the scale factor and maximum/minimum stress results are of interest when FOS is defined.

**SOL 108, 111:** This FOS option is actually a margin of safety (MOS) calculation, which requires only the desired or required duration of entire loading sequence in time units. By default this is set to 1.0 and the time units are those defined by the TUNIT field on the FTGSEQ entry.

8. For fatigue analysis of spot and seam welds, besides the “SPOTW” and “SEAMW” lines themselves, “CERTNTY”, and “FOS” are also applicable. “MULTI” is applicable to seam weld, but not spot welds. All others are ignored. TYPE field is ignored as an SN analysis is forced for fatigue of spot and seam welds.
9. Biaxial/Multiaxial assessment can be requested for all but fatigue analysis of spot welds and is ignored by SOL 200 optimization runs. These assessments require a 2D state of stress. For this reason only shell elements are supported with this feature. MMTHD=AUTO is not allowed for fatigue analysis of seam welds.
10. The Rupp method for spot weld life prediction has not been validated for the prediction of fatigue damage occurring at the middle sheets for spot welds joining more than two sheets. It should also be noted that such failures are relatively rare, difficult to reproduce in the laboratory, and difficult to detect in practice. Scrutinize middle sheet results carefully as they may give a false positive failure prediction.
11. Factor of Safety (FOS) analysis is only compatible with the MMTHD=SIMPLE if doing a multi-axial/biaxial (MULTI) assessment in the same run. MMTHD=AUTO is not allowed in fatigue analysis of seam welds (SEAMW). The stress/strain combination (COMB) is ignored if MMTHD=AUTO because the program will determine whether to use a critical plane or absolute maximum principal combination method.
12. When LOC=NODA for requesting nodal average stresses, the grid point stresses generator is used to determine the nodal averaging. A limited set of controls is provided on the NAVG line to control how the averaging method. Internally, SURFACE and/or VOLUME definitions are created for the grid point stress request, just as if the GPSTRESS case control, with its associated OUTPUT(POST) commands, were given. However, only one SURFACE and/or VOLUME is determined based on the elements defined on the FTGDEF entry for requesting fatigue analysis. The user should be aware of possible stress discontinuities that can result from this.

13. By default the response PSD at the most critical location is output using LOGLVL=0 via a `<jobname>PSD.csv` file. If LOGLVL=1, response PSDs for all requested locations are output to that same file. Also a `<jobname>RCC.csv` file is produced with limited data showing the summation of damages from the rainflow cycle count. If LOGLVL=2, both files are created and in the latter, addition the stress and damage BIN data for each event is included. Be aware that this can create a lot of data, resulting in huge output files, and can severely degrade performance. Please use this option with care and limit the output to only the critical elements of interest.

Table 9-17 Allowable Values for the COMB Field

Stress/Strain Combination	SOL 101, 103, 112* Valid for TYPE	SOL 108, 111 Valid for TYPE	Meaning
ABSMAXPR	SN (+Seam Weld) EN	SN EN	Absolute Maximum Principal (default)†
MAXPRINC	SN	SN EN	Maximum Principal Stress‡
SGVON	SN EN	SN EN	Signed von Mises Stress (recommended for SOL 108, 111 but not default)
SGMAXSHR	SN EN		Signed Maximum Shear
CRITICAL	SN (+Seam Weld), EN		Critical Plane Analysis - every 10 degrees by default unless NANGLE is specified otherwise for standard SN/EN analysis only- Not valid for 3D solid or CSHEAR elements - element must have an in-plane stress state.
STNDRD	SN (Spot Weld only)		This is basically a critical plane analysis with calculations being done at NANGLEs around the circumference of the spot weld.
COMPX COMPY COMPZ COMPXY COMPYZ COMPZX		SN EN	Individual stress components

\* In time domain, six multiaxial components of the stress tensor are resolved into one uniaxial, combined value for fatigue calculations at each entity per time step since damage models are based on uniaxial theories. For S-N analysis, the signed von Mises (SGVON) will be smaller than the Absolute Maximum Principal (ABSMAXPR) when there is positive biaxiality and hence this selection would be less conservative. (Note also that some BS weld classes require shear stress to be used.) The signed parameters use the sign of the absolute maximum principal value for conservative fatigue life estimates. It is not recommended to use non-signed values, thus they are not supported.

† In frequency domain, this is a Maximum Principal calculation using accurate phase scanning.

‡ In frequency domain, this is a fast Maximum Principal calculation using an approximate algorithm and is less accurate than ABSMAXPR, but improves computational speed.

Table 9-18 Allowable Values for the CORR Field

Mean Stress Correction	SOL 101, 103, 112 Valid for TYPE	SOL 108, 111 Valid for TYPE	Meaning
NONE	SN (+Spot/Seam Weld) EN	SN EN	No mean stress correction (default)
GOODMAN	SN	SN EN	Goodman mean stress correction
GERBER	SN	SN EN	Gerber mean stress correction
GDMANT	SN	SN EN	Tension only Goodman mean stress correction
GRBERT	SN	SN EN	Tension only Gerber mean stress correction
FKM	SN (+Spot/Seam Weld)		FKM mean stress correction method. Uses M1, M2, M3, M4 slopes as defined in <a href="#">MATFTG</a> entry. See remarks in <a href="#">MATFTG</a> entry.
SIMPLE	SN (Spot Weld only)		Modified FKM mean stress correction method where $M1=M2=M3=M4=-MSS$ as defined in the <a href="#">MATFTG</a> entry.
INTERP	SN		Interpolation method used with multiple SN curves only. TYPE field of <a href="#">MATFTG</a> entry must be set to MEAN, RRATIO, or LIFE. Requires that there be multiple curves defined, one at $R=-1$ for TYPE=RRATIO, or zero (0) mean stress for TYPE=MEAN.
SWT	EN	SN EN	Smith-Watson-Topper mean stress correction
MORROW	EN	EN	Morrow mean stress correction
WALKER		EN	Walker mean stress correction
MMPDS		SN EN	Walker mean stress correction built into the MMPDS-01 material curves

Table 9-19 Allowable S-N vs. Mean Stress Correction Methods for SOLs 101, 103, 112

Mean Stress Correction	S-N Method				
	Standard	Multi Mean Curve	Multi R-Ratio Curve	Haigh	Bastenaire
NONE	YES	YES*	YES†	NO	YES
GOODMAN	YES	YES*	YES†	NO	YES
GERBER	YES	YES*	YES†	NO	YES
GDMANT	YES	YES*	YES†	NO	YES
GRBERT	YES	YES*	YES†	NO	YES
FKM	YES	YES*	YES†	NO	YES
SIMPLE‡	YES	NO	NO	NO	NO
INTERP	NO	YES	YES	YES	NO

\* Allowed but a curve at zero mean stress must be present.

† Allowed but a curve at R = -1 must be present.

‡ Spot Welds only.

Table 9-20 Alternative NCALC options to calculate “N” (cycles to failure)

NCALC option	Meaning
AVG	Calculation of “N” is done by adding up all the values of N within the BIN and then dividing through by the number of N values used. In this context the average value of N is calculated over the BIN.
MID	“N” is extracted at the mid (stress) point of the BIN.
UPPER	“N” is extracted at the upper (stress) point of the BIN.

## FTGSEQ

## Fatigue Load Sequence

Defines the loading sequence for pseudo-static fatigue analysis using SOL 101 or modal transient fatigue analysis using SOL 103 or SOL 112 or vibration fatigue using SOL 108 or SOL 111.

Format (SOL 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
FTGSEQ	ID	EVNTOUT	METHOD						
	FID1	N1	FID2	N2	FID3	N3	FID4	N4	
	FID5	N5	...	...	...	...	...	...	
	-etc.-								
	"UNITS"	EQUIV	EQNAME						

Format (SOL 108, 111):

1	2	3	4	5	6	7	8	9	10
FTGSEQ	ID	EVNTOUT		TUNIT	LDM				
	FID1	N1	FID2	N2	FID3	N3	FID4	N4	
	FID5	N5	...	...	...	...	...	...	
	-etc.-								
	"UNITS"	EQUIV	EQNAME						

Examples:

FTGSEQ	1								
	6	1 5							

Descriptor	Meaning
ID	Unique ID with respect to all other FTGSEQ and FTGEVNT entries called out by a FATIGUE case control. (Integer>0) See Remark 1.
EVNTOUT	Flag for requesting fatigue output for each event individually. (Integer; 0=no or 1=yes; Default = 0). See Remark 7. For SOL 108 and 111, this also controls the written output request for spectral statistics (m0, m1, m2, m4, etc.).
METHOD	SOL 101, 103, 112 only. Event processing method. 0=independent, 1=Combined Full, 2=Combined Fast (Integer; Default = 0). This entry is only honored from the controlling FTGSEQ entry when called out by the FATIGUE case control. Otherwise it is ignored. See Remark 2.
TUNIT	SOL 108, 111 only. Time units of Ni values. SECS, MINS, HRS, DAYS, WKS, MTHS, YRS for seconds, minutes, hours, days, weeks, months, or years, respectively (Character; Default=SECS).

Describer	Meaning
LDM	<b>SOL 108, 111</b> only. FE load magnitude used to create transfer function. Used to normalize the transfer function stresses (Real, Default=1.0).
FIDi	<ul style="list-style-type: none"> <li>■ <b>SOLs 101, 103</b>: ID of a FTGEVNT or another FTGSEQ entry for pseudo-static fatigue analysis using SOL 101 or a modal analysis using SOL 103 (Integer &gt; 0). See Remark 3. and 4.</li> <li>■ <b>SOL 112</b>: A subcase ID that represents the loading event or another FTGSEQ entry for modal transient fatigue analysis using SOL 112. (Integer &gt; 0). See Remark 3. and 4.</li> <li>■ <b>SOL 108, 111</b>: ID of a FTGEVNT entry for random vibration fatigue using SOLs 108 or 111. Nested FTQSEQs are not allowed.</li> </ul>
Ni	<ul style="list-style-type: none"> <li>■ <b>SOLs 101, 103, 112</b>: Number of repeats of this loading sequence or event (Real&gt;0.0, Default=1.0). Ignored if only one event is defined. For METHOD equal 1 and 2, Ni must be a whole number, i.e., 1.0, 2.0, 3.0, etc. In other words, fractions of events are not allowed. Fractional events are also not allowed for any METHOD if the corresponding FID references a FTGSEQ entry. See Remark 3. and 4.</li> <li>■ <b>SOL 108, 111</b>: Duration of each event in TUNIT units. This value is not used for random loading only. These values are also not needed and overwritten if SWRATE/NSWEEP are provided on a FTGLOAD entry of TYPE= SINE (sine sweep). (Real&gt;0.0; Default=1.0).</li> </ul>
"UNITS"	Flag indicating that a fatigue equivalent unit name is applied to this loading. See Remark 5. and 6.
EQUIV	Number of equivalent units. (Real>0.0; Default=1.0 ). See Remark 5. and 6.
EQNAME	Equivalent name of this loading event. EQNAME can span across fields 4 through 9. If not defined it will be called <i>Repeats</i> (Character). No spaces are allowed. See Remark 5. and 6.

**Remarks:**

1. FTGSEQ bulk data are called out by FATIGUE case control.
2. Processing of events can be done by determining the damage due to each event independently (default) and then summing the damage due to all events. Or the events can be concatenated and damage determined after rainflow cycle counting over all events. The advantage of the independent method over the combined methods is computational expense versus accuracy. The combined method will close all cycles, whereas the individual method may miss a large damaging cycle if the cycle begins in one event and ends or closes in a subsequent event. The combined fast method performs a rainflow count data reduction to speed up the analysis and determine the most critical locations first and then redoes a full analysis on the critical locations.

3. Once a FTGSEQ bulk data entry is referenced in an FIDi field, it can't be referenced again in any other FTGSEQ entries (within its own associations - the same fatigue analysis) to avoid infinite loops. And if it is referenced by the FATIGUE case control, it cannot appear in any FIDi field of the FTGSEQ bulk data.
4. Different FTGEVNTs can be set up and the user can construct each sequence by specifying how many times to repeat each event in a sequence. One sequence could then be referenced in another sequence to tell the new sequence how many times to repeat that sequence. As an example, assume there are three events an automobile is subjected to: cobble stones, pot holes, speed bumps. One sequence might be five (5) "cobble stones," six (6) "potholes" and three (3) "speed bumps." This sequence may be called "torture track." Also define two more events called "cornering left" and "cornering right." A load sequence of ten (10) "cornering left" and ten (10) "cornering right" might be called "country road." Now with a nested FTGSEQ you can put these together any way you want. So one fatigue analysis might use a sequence of only "country road," another of only "torture track" and another of a combined six (6) "torture tracks," five (5) "country roads," followed by one (1) more "torture track" and one (1) more "country road" This would result in 3 fatigue analyses as shown in [Table 9-21](#) below.
5. If the "UNITS" flag is absent, the default fatigue equivalent unit is 1.0 *Repeats* of the resulting stress time history sequence. Equivalent units specified on FTGLOAD entries are ignored when FTGSEQ entries are used
6. Example of using equivalent units: If one repeat (or total duration) of the sequence is equivalent to five (5) times around a test track, the equivalent unit name, EQNAM, might be "*laps*," and the equivalent unit, EQUIV, would be five (5). Fatigue life will be reported in these units if defined, otherwise they are reported as repeats of the loading sequence. Life output is reported in both *Repeats* and the fatigue equivalent units, if defined.
7. For SOL 200 only EVNTOUT=0 is supported. Also some additional output file formats are not supported if EVNTOUT>0. See the [FATIGUE case control entry](#) for STROUT=4 limitations.
8. **SOL 108, 111:** For vibration fatigue a FTGSEQ cannot reference both an event with single input random loading and multiple input random loading.

Table 9-21 Example of nested FTGSEQ entries

```

SET 1=22,33,44
FATIGUE=1
BEGIN BULK
$
FTGSEQ,22          $<-- 1 country road only
,2
FTGSEQ,33          $<-- 1 torture track only
,3
$combined
FTGSEQ,44          $<-- 6 torture track, 5 country road, + 1 of each
,3,6.0,2,5.0,3,1.0,2,1.0
$country road:
FTGSEQ,2
,8,10.0,9,10.0    $<-- 10 cornering left + 10 cornering right
$torture track:
FTGSEQ 3
,5,5.0,6,6.0,7,3.0 $<-- 5 cobble stones, 6 potholes, 3 speed bumps
$

```



FTGEVNT,5... \$cobble stones  
FTGEVNT,6... \$potholes  
FTGEVNT,7... \$speed bumps  
  
FTGEVNT,8... \$cornering right  
FTGEVNT,9... \$cornering left

## GBAG Gas-Bag Pressure Definition

Defines the pressure within an enclosed volume. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
GBAG	GID	BSID	TRIGGER	TRIGGERV	PORID	INFID	HTRID	INTID	
	CPGAS	GPGASV	RGAS	PENV		REVERSE	CHECK	PINIT	
	TINIT	TENV							

Examples:

GBAG	101	37			11	12			
	CONSTANT	1000.	297.0	101325.		ON	ON		
	293.0								

Describer	Meaning		
GID	Unique gas-bag number.	I > 0	Required
BSID	Identification number of a BSURE, BCBOX, BCPROP, BCMATL, or BCSEG entry defining the surface of the gas-bag.	I > 0	Required
TRIGGER	The time-dependent parameters are offset in time.	Character	TIME
	TIME The offset is defined at TRIGGERV.		
TRIGGERV	The value of the offset in time.	Real	Required
PORID	Number of a set of LEAKAGE entries, that defines the porosity (permeability) and holes for the gas-bag surface and/or subsurfaces.	Integer > 0	No porosity
INFID	Number of a set of ABINFL entries, that defines the one or more inflators on subsurface(s) of the GBAG surface.	Integer > 0	No inflators
HTRID	Number of a set of HEATLOS entries, that defines the heat transfer definitions for the gas-bag surface and/or subsurfaces.	I > 0	No heat transfer
INTID	ID of an INITGAS entry specifying the initial gas composition for this GBAG.	I > 0	No initial gas composition

Describer	Meaning		
CPGAS	The variation of the specific heat constant at constant pressure. CONSTANT The specific heat is constant and specified in CPGASV.	Character	CONSTANT
CPGASV	The specific heat of the gas.	Real	Required
RGAS	Gas constant of the inflowing gas.	Real	Required
PENV	Environmental pressure surrounding the gas bag.	Real	Required
REVERSE	Normal auto-reverse switch.  ON The normals of the surface are automatically reversed if necessary so that they point in the same direction and provide a positive volume.  OFF The normals are not automatically reversed.	C	ON
CHECK	Normal checking switch.  ON The normals of the surface are checked to see if they all point in the same direction and provide a positive volume.  OFF The normals are not checked.  If REVERSE is set to ON, CHECK is automatically set to ON.	C	ON
PINIT	Initial pressure inside the gas bag.	Real	PENV
TINIT	Initial temperature inside the gas bag. See Remark 4.	Real	Required.
TENV	Environmental Temperature.	Real > 0	TINIT

Remarks:

1. The BSURF entry referenced by the BSID field must form a closed volume.
2. The pressure in the gas bag is applied to all the faces of the outer boundary.
3. TINIT is the temperature of the gas inside the volume at  $time = 0$ . At  $time = 0$ , the mass of the gas inside the gas bag is calculated as

$$m = \frac{P_{init}V}{RT_{init}}$$

where,  $P_{init}$  the initial pressure,  $V$  the volume,  $R$  the gas constant, and  $T_{init}$  the initial gas temperature.

- The flow through exhaust openings, leakage areas and user-specified outflow rate is accumulated. The volumetric porosity contributes to the outflow of gas as

$$\dot{m}_{out} = \rho \cdot Q = \frac{p}{R \cdot T} \cdot Q$$

where

$Q$	= volumetric flow rate
$\rho$	= density inside the bag
$p$	= pressure inside the bag
$R$	= gas constant
$T$	= temperature inside the bag
$\dot{m}_{out}$	= mass outflow rate

The value of  $Q$  can be specified as a constant, as a function of the pressure difference, or as a function of time. Negative values for the volumetric flow rate are not allowed, since this would mean inflow of outside air.

- A mixture of BSURF, BCBOX, BCPROP, BCMATL or BCSEG with the same BSID is allowed. However multiple BSID of the same type is not allowed. When using this option, special care must be taken to assure the same element is not part of multiple BSID definitions.

## GBAGCOU

### General Coupling to Gas-Bag Switch

Defines a switch from full gas dynamics to uniform pressure formulation. Used in SOL 700 only.

**Format:**

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
GBAGCOU	ID	CID	GID	TSTART	PERCENT				

**Examples:**

GBAGCOU	1	100	101	0.0	5				
---------	---	-----	-----	-----	---	--	--	--	--

Describer	Meaning		
ID	Unique number of a GBAGCOU entry.	I > 0	Required
CID	Number of a COUPLE entry.	I > 0	Required
GID	Number of a GBAG entry.	I > 0	0.0
TSTART	Time after which the coupling algorithm checks if a switch to the uniform pressure method is valid. It is valid when the following is true:  $\text{Max} \left[ \frac{P_{max} + P_{average}}{P_{average}}, \frac{P_{average} - P_{min}}{P_{average}} \right] < \frac{\text{PERCENT}}{100}$ where Pmax = maximum Eulerian pressure exerted on the SURFACE Pmin = minimum Eulerian pressure exerted on the SURFACE Paverage = average Eulerian pressure exerted on the SURFACE	Real ≥ 0	0.0
PERCENT	Value used in validity check as defined previously.	R ≥ 0.	0.0

**Remarks:**

1. The BSID referenced by the COUPLE entry CID and by the GBAG entry GID must be the same.
2. All Eulerian and general coupling calculations are deactivated after transition from gas dynamics to uniform pressure.

**GENEL**

**General Element**

Defines a general element.

Format:

1	2	3	4	5	6	7	8	9	10
GENEL	EID		UI1	CI1	UI2	CI2	UI3	CI3	
	UI4	CI4	UI5	CI5	-etc.-				

UI<sub>m</sub> -- The last item in the UI list will appear in one of fields 2, 4, 6, or 8.

	"UD"		UD1	CD1	UD2	CD2	-etc.-		
--	------	--	-----	-----	-----	-----	--------	--	--

UD<sub>n</sub> -- The last item in the UD list will appear in one of fields 2, 4, 6, or 8.

	"K" or "Z"	KZ11	KZ21	KZ31	-etc.-	KZ22	KZ32		
	-etc.-		KZ33	KZ43	-etc.-				

KZ<sub>mm</sub> -- The last item in the K or Z matrix will appear in one of fields 2 through 9.

	"S"	S11	S12	-etc.-		S21	-etc.-		
--	-----	-----	-----	--------	--	-----	--------	--	--

S<sub>mn</sub> -- The last item in the S matrix will appear in one of fields 2 through 9.

Example:

GENEL	629		1	1	13	4	42	0	
	24	2							
	UD		6	2	33	0			
	Z	1.0	2.0	3.0	4.0	5.0	6.0	7.0	
	8.0	9.0	10.0						
	S	1.5	2.5	3.5	4.5	5.5	6.5	7.5	
	8.5								

Descriptor	Meaning
EID	Unique element identification number. (Integer > 0)
UI <sub>i</sub> , CI <sub>i</sub> UD <sub>j</sub> , CD <sub>j</sub>	Identification numbers of degrees-of-freedom in the UI or UD list, in sequence corresponding to the [K], [Z], and [S] matrices. UI <sub>i</sub> and UD <sub>i</sub> are grid point numbers, and CI <sub>i</sub> and CD <sub>j</sub> are the component numbers. If a scalar point is given, the component number is zero. (Integer ≥ 0)
KZ <sub>ij</sub>	Values of the [K] or [Z] matrix ordered by columns from the diagonal, according to the UI list. (Real)

Describer	Meaning
Sij	Values of the [S] matrix ordered by rows according to the UD list. (Real)
“UD”, “K”, “Z”, and “S”	Character strings that indicate the start of data belonging to the UD list or the [K], [Z], or [S] matrices.

Remarks:

1. The stiffness approach:

$$\begin{Bmatrix} f_i \\ f_d \end{Bmatrix} = \begin{bmatrix} K & -KS \\ -S^T K & S^T K S \end{bmatrix} \begin{Bmatrix} u_i \\ u_d \end{Bmatrix}$$

The flexibility approach:

$$\begin{Bmatrix} u_i \\ f_d \end{Bmatrix} = \begin{bmatrix} Z & S \\ -S^T & O \end{bmatrix} \begin{Bmatrix} f_i \\ u_d \end{Bmatrix}$$

where

$$\begin{aligned} \{u_i\} &= [u_{i1}, u_{i2}, \dots, u_{im}]^T \\ \{u_d\} &= [u_{d1}, u_{d2}, \dots, u_{dn}]^T \\ [KZ] &= \begin{bmatrix} KZ11 & \dots & \dots & \dots \\ KZ21 & KZ22 & \dots & \dots \\ KZ31 & KZ32 & \dots & \dots \\ \vdots & \vdots & \vdots & \vdots \\ KZ_{m1} & \dots & \dots & KZ_{mm} \end{bmatrix} \text{ and } [KZ]^T = [KZ] \end{aligned}$$

$$[S] = \begin{bmatrix} S11 & \dots & S1n \\ S21 & \dots & \dots \\ S31 & \dots & \dots \\ \vdots & \vdots & \vdots \\ S_{m1} & \dots & S_{mn} \end{bmatrix}$$

The required input is the  $\{u_i\}$  list and the lower triangular portion of  $[K]$  or  $[Z]$ . Additional input may include the  $\{u_d\}$  list and  $[S]$ . If  $[S]$  is input,  $\{u_d\}$  must also be input. If  $\{u_d\}$  is input but  $[S]$  is omitted,  $[S]$  is internally calculated. In this case,  $\{u_d\}$  must contain six and only six degrees-of-freedom.

The forms shown above for both the stiffness and flexibility approaches assume that the element is a free body with rigid body motions that are defined by  $\{u_i\} = [S]\{u_d\}$ . See [General Element Capability \(GENEL\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion.

2. When the stiffness matrix  $K$  is input, the number of significant digits should be the same for all terms.
3. Double-field format may be used for input of  $K$  or  $Z$ .
4. The DMIG entry or the INPUTT4 module offer alternative methods for inputting large matrices.
5. The general element entry in the example above defines the following:

$$[u_i] = [1-1, 13-4, 42, 24-2]^T$$

$$\{u_d\} = [6-2, 33]^T$$

where i-j means the j-th component of grid point i. Points 42 and 33 are scalar points.

$$[Z] = \begin{bmatrix} 1.0 & 2.0 & 3.0 & 4.0 \\ 2.0 & 5.0 & 6.0 & 7.0 \\ 3.0 & 6.0 & 8.0 & 9.0 \\ 4.0 & 7.0 & 9.0 & 10.0 \end{bmatrix} \quad [S] = \begin{bmatrix} 1.5 & 2.5 \\ 3.5 & 4.5 \\ 5.5 & 6.5 \\ 7.5 & 8.5 \end{bmatrix}$$



**GENUDS**

User Data for Notify User Defined Service or Subroutine

The GENUDS defines integer, real and character type data that will be passed to the notify method in runtime info interface. The notify method will be called at start of load case, start of increment, end of increment and end of load case.

Format:

1	2	3	4	5	6	7	8	9	10
GENUDS	SRV_ID								
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	IDATA <sub>n</sub>				
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	RDATA <sub>n</sub>				
	“CHAR”	CDATA1	CDATA2	...	CDATA <sub>n</sub>				

Example:

GENUDS	MY_SRV								
	INT	1	2	100					

Descriptor	Meaning
SRV_ID	The service identifier used in the CONNECT SERVICE statement. (Character, no default)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	User supplied integer data. (Integer, no default)
“REAL”	Keyword indicating the following data is real. (Character)
RDATA <sub>i</sub>	User supplied real data. (Real, no default)
“CHAR”	Keyword indicating the following data is character. (Character)
CDATA <sub>i</sub>	User supplied character data. (Character, no default)

Remarks:

1. The SRV\_ID is the service identifier of SCA service in the CONNECT SERVICE statement. The SCA service should have implemented the RuntimeInfo interface.
2. A CDATA<sub>i</sub> entry cannot be the character “INT”, “REAL” or “CHAR”.

## GMBNDC

## Geometric Boundary - Curve

Defines a geometric boundary consisting of element edges along a curve interface. The boundary may consist of edges of shell, beam, or solid elements.

Format:

1	2	3	4	5	6	7	8	9	10
GMBNDC	BID	GRID1	GRIDF						
	ENTITY	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	-etc.-							

Examples:

GMBNDC	1	101	106						
	GRID	102	103	104	105				

Describer	Meaning
BID	Boundary identification number to be referenced by a CINTC entry. (Integer > 0)
GRID1	Initial grid identification number for boundary. (Integer > 0)
GRIDF	Final grid identification number for boundary. (Integer > 0)
ENTITY	Entity type for defining boundary. GRID (Character)
IDi	Entity identification numbers for boundary of subdomain. Values in the list must be unique. (Integer > 0)

Remarks:

1. All boundary identification numbers must be unique.
2. GRID1 and GRIDF define the end points of the boundary.
3. For each boundary, GRID is required.
4. For the GRID entity type, the entities should be listed in order from the GRID1 to the GRIDF. The GRID1 and GRIDF need not be repeated in the IDi list.
5. Multiple continuation entries may be specified for additional entity identification numbers, IDi.
6. Interface elements may generate high or negative matrix/factor diagonal ratios. If there are no other modelling errors, these messages may be ignored and PARAM,BAILOUT,-1 may be used to continue the run.

## GMNURB

## 3D Contact Region Made Up of NURBS in SOL 600

Defines a 3D contact region made up of NURBS using the MSC Marc style used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
GMNURB	ID	NPTU	NPTV	NORU	NORV	NSUBU	NSUBV	NTRIM	
		G1 or X1	G2 or Y1	G3 or Z1	G4 or X2	G5 or Y2	G6 or Z2	G7	
		G8 or X3	G9 or Y3	G10 or Z3	etc.		[abs(nptu)*n ptv vaues]	See Remark 3	
		Homo1	Homo2	Homo3	Homo4	Homo5	Homo6	Homo7	
		Homo8	Homo9	Homo10	Homo11	etc	[nptu*nptv values]		
		Knot1	Knot2	Knot3	Knot4	Knot5	Knot6	Knot7	
		Knot8	Knot9	Knot10	etc.		[(nptu+noru) +(nptv+norv) values]		
		IDtrim	NPTUtrim	NORUtrim	NSUBtrim		(repeat this and all following lines NTRIM times)		
			Xisoparam	Yisoparam			(NPTUtrim entries)		
			Homo1	Homo2	Homo3	etc	(NPTUtrim entries)		
			Knot1	Knot2	Knot3	etc	(NPTUtrim +NORUtrim entries)		

Example:

GMNURB	901	9	5	4	4	20	5		
		101	102	103	104	105	106	107	
		108	109	110	111	112	113	114	
		115	116	117	118	119	120	121	
		122	123	124	125	126	127	128	
		129	130	131	132	133	134	135	
		136	137	138	139	140	141	142	
		143	144	145					
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1	1	1	1	1	
		1	1	1					
		0	0	0	0	0.16667	0.33333	0.5	
		0.66667	0.833333	1	1	1	1	0	
		0	0	0	0.5	1	1	1	
		1							

Describer	Meaning
ID	Identification number of a surface defined by NURBS. ID is called out on a BCBODY entry with a NURBS2 header. (Integer > 0; Required)
NPTU	Absolute value of the number of control points. Enter NPTU as a positive number if the control points are to be input using GRID points. Enter NPTU as a negative number if the control points are to be entered using x,y,z. (Integer > 0; Required)
NPTV	Number of control points in V direction. (Integer > 0; Required)
NORU	Order along U direction. (Integer > 0; Required)
NORV	Order along V direction. (Integer > 0; Required)
NSUBU	Number of subdivisions in U direction. (Integer > 0; Required)
NSUBV	Number of subdivisions in V direction. (Integer > 0; Required)
NTRIM	Number of trimming curves. (Integer ≥ 0 or blank)
G1, G2, G3, etc.	Grid point IDs defining control points (Integer > 0; Required). There must be NPTU*NPTV entries.
X1, Y1, Z1, X2, Y2, Z2, etc.	Alternate method to define control points without using GRID points. There must be abs(NPTU)*NPTV (x,y,z) entries.

Describer	Meaning
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0). There must be NPTU*NPTV entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) in the V direction. There must be (NPTU+NORU)+(NPTV+NORV) entries. (Real)
IDtrim	ID of trimming vector. There must NTRIM of these entries and those entries that follow. (Integer > 0)
NPUTtrim	Number of control points for this trimming vector. (Integer > 0)
NORUtrim	Order for this trimming vector. (Integer > 0)
NSUBtrim	Number of subdivisions for this trimming vector. (Integer > 0)
Xisoparam	First coordinate of point in isoparametric space. (Real)
Ysoparam	Second coordinate of point in isoparametric space (Real)
Homo1, Homo2, Homo3, etc	Homogeneous coordinates (0.0 to 1.0) of this trimming vector. There must be NPTUtrim entries. (Real)
Knot1, Knot2, Knot3, etc	Knot vectors (0.0 to 1.0) of this trimming vector. There must be NPTUtrim+NORUtrim entries. (Real)

#### Remarks:

1. GMNURB is recognized only in SOL 600.
2. **WARNING:** For rigid contact, the right hand rule determines the interior side of the rigid surface. A deformable surface which contacts a rigid surface must be on the exterior side of the rigid surface (i.e. in the direction opposite to the right hand rule). If a rigid surface is described backwards, contact will not occur because the deformable body is already inside the rigid body at the start of the analysis. For 3D patches, if all need to be reversed, the parameter PARAM,MARCREVR,1 may be entered to automatically reverse all 3D patches.
3. For NURBS, enter either IDs of NPTU\*NPTV grid points as G1, G2, G3 etc. (set positive NPTU) or coordinates of abs(NPTU)\*NPTV grid point as (X1, Y1, Z1), (X2, Y2, Z2), etc. (set negative NPTU).

**GMOVR3**

RBE3 SOL400 Rigid = Lagrange perturbation coincident grid override.

Allows override values to the MDLPRM, GMOVR3 entry.

Format:

1	2	3	4	5	6	7	8	9	10
GMOVR3	ELEM1	$\Delta X_1$	$\Delta Y_1$	$\Delta Z_1$	$\Delta$ GMOVR3T1				
	ELEM2	$\Delta X_2$	$\Delta Y_2$	$\Delta Z_2$	$\Delta$ GMOVR3T2				
	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-				

Example:

GMOVR3	36955001	0.1							
--------	----------	-----	--	--	--	--	--	--	--

Describer	Meaning
ELEMi	RBE3 element ID. (Integer>0)
$\Delta X_i$	Perturbation value of X coordinate (Real; default = 0.01)
$\Delta Y_i$	Perturbation value of Y coordinate (Real; default = 0.01)
$\Delta Z_i$	Perturbation value of Z coordinate (Real; default = 0.01)
$\Delta$ GMOVR3Ti	Adjustment to MDLPRM, GMOVR3T value. Zero Tolerance = $\text{GMOVR3T} + \Delta$ GMOVR3T $\geq 0.0$ . (Real; Default=0.0)

Remarks:

1. This entry is only active if MDLPRM,GMOVR3,3 or MDLPRM,GMOVR3,4 is present on the MDLPRM Bulk Data entry.
2. This entry only affects RBE3 elements meeting the following criterion:
  - a. Let  $x^q, y^q, z^q$  be the coordinates of the REFGRID and  $x^k, y^k, z^k$  the coordinates of one of the connected Gi,j nodes.
  - b. If  $|x^k - x^q|, |y^k - y^q|$ , and  $|z^k - z^q|$  are all less than or equal to  $\text{GMOVR3T} + \Delta$ GMOVR3T) for ALL connected Gi,j nodes, the grids are considered coincident.

Any entry not meeting the above requirements will be ignored.

## GMQVOL

### Define Volumetric Heat Loads

Defines a rate of volumetric heat generation in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
GMQVOL	LID	METHOD	FIELD1		EID1	EID2	EID3	EID4	
	EID5	etc.							

Example:

GMQVOL	100	TABLE	20		18	23	7		
--------	-----	-------	----	--	----	----	---	--	--

Describer	Meaning		
LID	Load set identification number.	Integer > 0	Required
METHOD	Method used to specify the data (EQUATION, TABLE, CONSTANT). See Remark 2.	Character	Required
FIELD1	Volumetric heat load data. See Remark 2.	Integer or Real	Required
EID <sub>i</sub>	A list of heat conduction elements.	Integer > 0	Required

Remarks:

- For steady-state analysis, the load set is selected in the Case Control Section (LOAD=LID).
- METHOD specifies the data type of FIELD1 to be constants, equation IDs, or table IDs. Values in FIELD1 and FIELD2 are:

METHOD	FIELD1
EQUATION	ID of a DEQATN entry defining the volumetric heat generation rate as a function of location (Integer > 0).
TABLE	ID of a TABLE3D entry defining the volumetric heat generation rate as a function of location (Integer > 0).
CONSTANT	Value of volumetric heat generation rate (Real).

Note that the fifth field will be reserved for the future development of temperature dependent functions.

## GRAV Acceleration or Gravity Load

Defines acceleration vectors for gravity or other acceleration loading.

Format:

1	2	3	4	5	6	7	8	9	10
GRAV	SID	CID	A	N1	N2	N3	MB		

Example:

GRAV	1	3	32.2	0.0	0.0	-1.0			
------	---	---	------	-----	-----	------	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0; Default = 0)
A	Acceleration vector scale factor. (Real)
Ni	Acceleration vector components measured in coordinate system CID. (Real; at least one Ni ≠ 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 10. (Integer; Default = 0)

Remarks:

- The acceleration vector is defined by  $\vec{a} = A\vec{N}$ , where  $\vec{N}$  is the vector defined by (N1, N2, N3).  
The magnitude of  $\vec{a}$  is equal to A times the magnitude of  $\vec{N}$ . The static loads generated by this entry are in the direction of  $\vec{a}$ .
- A CID of zero references the basic coordinate system.
- Acceleration or gravity loads may be combined with “simple loads” (e.g., FORCE, MOMENT) only by specification on a LOAD entry. That is, the SID on a GRAV entry may not be the same as that on a simple load entry.
- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- At most nine GRAV entries can be selected in a given run either by Case Control or the LOAD Bulk Data entry. Multiples or reflections of a given acceleration or gravity load can be economically accomplished by use of the LOAD Bulk Data entry.



6. In cyclic symmetry solution sequences, the T3 axis of the coordinate system referenced in field 3 must be parallel to the axis of symmetry. In dihedral cyclic symmetry (where STYPE = "DIH" on the CYSYM entry), the T1 axis must, in addition, be parallel to Side 1 of segment 1R of the model.
7. For image superelements, the coordinate system must be rotated if the image is rotated relative to its primary superelement.
8. Acceleration or gravity loads do not include effects due to mass on scalar points.
9. The RFORCE entry may be used to specify rotational accelerations.
10. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

## GRDSET

### GRID Entry Defaults

Defines default options for fields 3, 7, 8, and 9 of all GRID entries.

Format:

1	2	3	4	5	6	7	8	9	10
GRDSET		CP				CD	PS	SEID	

Example:

GRDSET		16				32	3456		
--------	--	----	--	--	--	----	------	--	--

Descriptor	Meaning
CP	Identification number of coordinate system in which the location of the grid points are defined. (Integer $\geq 0$ or blank)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors of the grid point are defined. (Integer $\geq -1$ or blank)
PS	Permanent single-point constraints on the grid point. (Any combination of Integers 1 through 6 with no embedded blanks, or blank.)
SEID	Superelement identification number. (Integer $\geq 0$ or blank)

Remarks:

1. The contents of fields 3, 7, 8, or 9 of this entry are assumed for the corresponding fields of any GRID entry whose field 3, 7, 8, and 9 are blank. If any of these fields on the GRID entry are blank, the default option defined by this entry occurs for that field. If no permanent single-point constraints are desired, one of the coordinate systems is basic, or the grid is assigned to the residual structure then the default may be overridden on the GRID entry by making one of fields 3, 7, 8, or 9 zero (rather than blank). Only one GRDSET entry may appear in the Bulk Data Section.
2. The primary purpose of this entry is to minimize the burden of preparing data for problems with a large amount of repetition (e.g., two-dimensional pinned-joint problems).
3. At least one of the fields CP, CD, PS, or SEID must be specified.

## GRID Grid Point

Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
GRID	ID	CP	X1	X2	X3	CD	PS	SEID	

Example:

GRID	2	3	1.0	-2.0	3.0		316		
------	---	---	-----	------	-----	--	-----	--	--

Free Field Large Format Example:

GRID\*, 2, , 1.0, -2.0,

\*, 3.0, , 136

Translates to:

1	2	3	4	5	6	7	8	9	10
GRID*	2				1.0		-2.0		+
*+	3.0				136				

The comma immediately following the -2.0 is required if NASTRAN IFPSTAR=NO is used. The comma is always recommended even if NASTRAN IFPSTAR=YES (Default) is used.

Describer	Meaning
ID	Grid point identification number. (0 < Integer < 100,000,000, see Remark 7.)
CP	Identification number of coordinate system in which the location of the grid point is defined. (Integer ≥ 0 or blank*)
X1, X2, X3	Location of the grid point in coordinate system CP. (Real; Default = 0.0)
CD	Identification number of coordinate system in which the displacements, degrees-of-freedom, constraints, and solution vectors are defined at the grid point. (Integer ≥ -1 or blank, see Remark 3.)*
PS	Permanent single-point constraints associated with the grid point. (Any of the Integers 1 through 6 with no embedded blanks, or blank*.)
SEID	Superelement identification number. (Integer ≥ 0; Default = 0)

\*See the GRDSET entry for default options for the CP, CD, PS, and SEID fields.

## Remarks:

1. All grid point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points.
2. The meaning of X1, X2, and X3 depends on the type of coordinate system CP as follows (see the CORDij entry descriptions):

Type	X1	X2	X3
Rectangular	X	Y	Z
Cylindrical	R	$\theta$ (degrees)	Z
Spherical	R	$\theta$ (degrees)	$\phi$ (degrees)

See [Grid Point and Coordinate System Definition](#) in the *MSC Nastran Reference Guide*, for a definition of coordinate system terminology.

3. The collection of all CD coordinate systems defined on all GRID entries is called the global coordinate system. All degrees-of-freedom, constraints, and solution vectors are expressed in the global coordinate system. It is recommended that points on the z-axis not have their displacement directions defined with cylindrical or spherical coordinates. (See further remarks on the CORD1S, CORD2S, CORD1C, and CORD2C.)
4. The SEID field can be overridden by use of the SESET entry.
5. If CD = -1, then this defines a fluid grid point in coupled fluid-structural analysis. This type of point may only connect the CAABSF, CHACBR, CHACAB, CHEXA, CPENTA, CPYRAM, and CTETRA elements to define fluid elements.
6. A zero (or blank if the GRDSET entry is not specified) in the CP and CD fields refers to the basic coordinate system.
7. For SOL 600, ID may range from 1 to  $2^{31} - 1$  (2147483647) if there are no OUTR options specified on the SOL 600 entry. If any OUTR option is specified the limit is 100000000.
8. For RC network solver in thermal analysis, the CD, PS and SEID are ignored.

## GRIDA

## Associative GRID for PAA

Defines an associative GRID point to be used in the COMBINE step of PAA.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDA	GID	ORIG_GID							
PARTNAME									

Example:

GRIDA	1	1000							
Left_outboard_wing_tank									

Descriptor	Meaning
GID	GRID ID to be used in the current run. (Integer > 0)
ORIG_GID	GRID ID in Part being referenced. (Integer > 0)
PATNAMEi	Name of Part ORIG_GID is in. (Character, C64, no internal blank spaces)

Remarks:

- GRIDA entries are used only in the COMBINE step of PAA processing. They are used to allow the other entries in the current bulk data to reference GRID points or SPOINTs in Parts that are being combined to create the Assembly.
- ORIG\_GID must exist in the referenced Part. If the Part has been reduced, it must be a boundary GRID for that Part (in the A-set of that Part).
- The GRIDA id may be treated the same as if it were defined using a GRID or SPOINT entry in the current run. The program will handle the connection to the referenced GRID/SPOINT in the Part.
- Only dof which are in the Part A-set will be available for connection. If a GRIDA references a GRID in a Part, which has fewer than 6 dof in the Part A-set (dof available in the Part matrices), although the GRIDA will have 6 dof, only the dof which exist in the Part matrices will be connected to the Part.

**GRIDB**

## Axisymmetric Grid Point

Defines the location of a geometric grid point on a fluid point (RINGFL entry) for an axisymmetric fluid model and/or axisymmetric structure. Also defines the boundary of the fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDB	ID			PHI		CD	PS	IDF	

Example:

GRIDB	30			30.0		3	345	20	
-------	----	--	--	------	--	---	-----	----	--

Describer	Meaning
ID	Grid point identification number. (0 < Integer < 1000000)
PHI	Azimuthal position of the fluid in degrees. (Real)
CD	Identification number of the coordinate system in which the displacements are defined at the grid point. (Integer ≥ 0 or blank)
PS	Permanent single-point constraints associated with grid point. (Any combination of the Integers 1 through 6 with no embedded blanks, or blank.)
IDF	Identification number of a RINGFL entry. (Integer > 0)

Remarks:

1. GRIDB is allowed only if an AXIF entry is also present. The AXIF entry must define a fluid coordinate system.
2. All GRIDB identification numbers must be unique with respect to other scalar, structural, fluid and extra (EPOINT) points.
3. The referenced RINGFL entry must be present and be included in a boundary list (BDYLIST entry).
4. If no harmonic numbers on the AXIF entry are specified, no fluid elements are necessary.
5. The collection of all CD coordinate systems defined on all GRID and GRIDB entries is called the global coordinate system.
6. Fields 3, 4, and 6 are ignored, which facilitates the conversion of GRID entries to GRIDB entries. Note that the fields are the same except for fields 1 and 9 when a cylindrical coordinate system is used.

## GRIDF Fluid Point

Defines a scalar degree-of-freedom for harmonic analysis of a fluid.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDF	ID	R	Z						

Example:

GRIDF	23	2.5	-7.3						
-------	----	-----	------	--	--	--	--	--	--

Describer	Meaning
ID	Identification number of axisymmetric fluid point. (0 < Integer < 1000000)
R	Radial location of point in basic coordinate system. (Real > 0.0)
Z	Axial location of point in basic coordinate system. (Real)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. Grid points on slot boundaries are defined on GRIDS entries. Do not also define them on GRIDF entries.
4. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. Pressures will be plotted as displacements in the basic Z direction.
5. Load and constraint conditions are applied as if GRIDF were a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.

## GRIDS

## Slot Surface Point

Defines a scalar degree-of-freedom with a two-dimensional location. Used in defining pressure in slotted acoustic cavities.

Format:

1	2	3	4	5	6	7	8	9	10
GRIDS	ID	R	Z	W	IDF				

Example:

GRIDS	25	2.5	-7.3	0.5					
-------	----	-----	------	-----	--	--	--	--	--

Describer	Meaning
ID	Identification number of the slot point. (Integer > 0)
R	Radial location of point in basic coordinate system. (Real $\neq$ 0.0)
Z	Axial location of point in basic coordinate system. (Real)
W	Slot width or thickness at the GRIDS point. (Real $\geq$ 0.0 or blank)
IDF	Identification number to define a GRIDF point. (Integer > 0 or blank)

Remarks:

1. This entry is allowed only if an AXSLOT entry is also present.
2. ID (and IDF if present) must be unique with respect to all other scalar, structural, fluid and extra (EPOINT) points.
3. If W is blank, the default value on the AXSLOT entry will be used.
4. The IDF number is referenced on the CAXIFi entry for central cavity fluid elements next to the interface. The IDF number is entered only if the grid point is on an interface. In this case, the IDF should also be defined on a GRIDF entry.
5. If IDF is nonzero, then R must be greater than zero.
6. For plotting purposes, the R location corresponds to the basic X coordinate. The Z location corresponds to the basic Y coordinate. The slot width, W, corresponds to the basic Z coordinate. The pressure will be plotted in the basic Z direction.
7. Load and constraint conditions are applied as if the GRIDS is a scalar point. Positive loads correspond to inward flow. A single-point constraint causes zero pressure at the point.



## GUST

## Aerodynamic Gust Load Description

Defines a stationary vertical gust for use in aeroelastic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
GUST	SID	DLOAD	WG	X0	V				

Example:

GUST	133	61	1.0	0.	1.+4				
------	-----	----	-----	----	------	--	--	--	--

Describer	Meaning
SID	Gust set identification number. (Integer > 0)
DLOAD	Set identification number of a TLOADi or RLOADi entry that defines the time or frequency dependence. (Integer > 0)
WG	Scale factor (gust velocity/forward velocity) for gust velocity. (Real ≠ 0.0)
X0	Streamwise location in the aerodynamic coordinate system of the gust reference point. (Real)
V	Velocity of vehicle. See Remark 5. (Real > 0.0)

Remarks:

1. The GUST entry must be selected with the Case Control command GUST = SID.
2. The gust angle is in the +z direction of the aerodynamic coordinate system. The value is

$$WG \cdot T\left(t - \frac{X - X0}{V}\right)$$

where  $T$  is the tabular function.

3. In random analysis, a unit gust velocity ( $WG = 1/\text{velocity}$ ) is suggested. The actual rms value is entered on the TABRNDG entry.
4. X0 and V may not change between subcases under one execution.
5. V must be equal to VELOCITY on the AERO Bulk Data entry.

## HADACRI Mesh Adaptivity Criterion and Corresponding Parameters - SOL 101 & 400 (Linear)

Specifies Mesh refinement criterion for adaptive mesh refinement and corresponding parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HADACRI	CRITID	TYPE	F1	F2	F3	F4	F5	F6	

Example:

HADACRI	1	1	0.9						
---------	---	---	-----	--	--	--	--	--	--

Describer	Meaning
CRITID	Identification number referenced by the CRITID field in the HADAPTL Bulk Data entry. (Integer > 0; no Default)
TYPE	Type of Mesh refinement adaptivity criteria. See Remark 2. (Integer $\geq 1$ and $\leq 4$ ; Default = 1)
F1 to F8	Criteria specific parameters. See Remark 2. (Real; no Default)

Remarks:

1. The mesh refinement criteria currently available (and selected in the TYPE field) are:

TYPE	Name of Mesh Refinement Criterion
1	Error indicator based
2	Element within a spatial spherical region
3	Elements within a spatial cubic region
4	Elements in contact criteria

2. The following table describes the different refinement criteria and corresponding parameters:

TYPE	Description
1	<p>In this case a scalar error indicator <math>E_e</math> is computed for all elements 'e' in the finite element mesh. Then, an element 'e' will be refined if</p> $E_e^2 \geq F_1 \bar{E}^2$ <p>where <math>F_1</math> is a weight factor (<math>0 \leq F_1 \leq 1</math>) specified in the F1 field and <math>\bar{E}</math> is the quadratic mean of the error indicator defined as</p> $\bar{E}^2 = \frac{1}{N} \sum_{e=1}^N E_e^2$ <p>with <math>N</math> the total number of elements in the element set where element 'e' belongs. For this criteria the fields F2 to F6 are ignored.</p>
	<p>The elemental error indicator <math>E_e</math> is computed using the grid point stresses following the procedure utilized by the ELSDCON Case Control command and described in <a href="#">Mesh Stress Discontinuities at Grid Points</a> in the <i>MSC Nastran Reference Guide</i>. This procedure can be summarized as follows:</p> <ul style="list-style-type: none"> <li>Let                     <math display="block">\sigma_{aij} = \sum_{e=1}^{N_a} W_a^e \sigma_{aij}^e</math> <p>be the weighted average over all elements 'e' concurrent to a given node 'a' of each component 'ij' of the grid point stresses <math>\sigma_{aij}^e</math> where <math>W^e</math> is a weighting factor assigned to element 'e' and <math>N_a</math> is the number of elements connected to the given node 'a'.</p> </li> <li>An estimate of the error in a particular component of stress 'ij' at a grid point 'a' is then be computed as                     <math display="block">E_{aij}^2 = \sum_{f=1}^N W_a (\sigma_{aij}^e - \sigma_{aij})^2</math> </li> <li>Averaging the latter over the different stress components, 'ij', over the different shell fibers (for shell elements) and over the different grid points 'a' connected by a given element 'e' we obtain the elemental, scalar error indicator <math>E_e</math>.</li> </ul>

TYPE	Description
2	In this case the user specifies a spherical region in space with center given by (F1,F2,F3) and radius given by F4. Then, all elements with at least one node with basic coordinates (X,Y,Z) with the spherical region (i.e., such that $\ (X, Y, Z) - (F1, F2, F3)\  < F4$ ) will be refined. For this criteria the fields F5 and F6 are ignored.
3	In this case the user specifies a hexahedral region in space, aligned with the basic coordinates system, with corners given by (F1,F2,F3) and (F4,F5,F6). Then, all elements with at least one node with basic coordinates (X,Y,Z) within the specified hexahedral region (i.e., such that $F1 \leq X \leq F4$ , $F2 \leq Y \leq F5$ , $F3 \leq Z \leq F6$ ) will be refined.
4	In this case all elements with at least one node involved in contact either as touching or touched nodes in deformable contact bodies are refined. For this criteria the fields F1 to F6 are ignored.

3. Each criteria must have a unique ID (specified by the CRITID field and referenced by the CRITID field of the Bulk Data entry, [HADAPTL](#)).
4. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI](#)).

## HADAPTL

## Local Adaptive Mesh Refinement Control Parameters - SOL 101 & 400 (Linear)

Specifies Local Adaptive Mesh Refinement control parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HADAPTL	ID			REPEAT	CRITID	WHEREMET	WHEREID	SNAPMETH	
	MAXLEVEL								

Example:

HADAPTL	1			10	1	PROP	5		
---------	---	--	--	----	---	------	---	--	--

Descriptor	Meaning
ID	Identification number referenced by the Case Control HADAPT command. (Integer > 0; no Default)
REPEAT	Maximum number of analysis performed before the adaptive mesh refinement process is stopped. See Remark 3. (Integer ≥ 0; Default = 6)
CRITID	Identification number of a mesh refinement criterion to be selected with the HADACRI Bulk Data entry. See Remarks 1. and 2. and the HADACRI Bulk Data entry. (Integer > 0; no Default)
WHEREMET	Method to specify the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. It can take the values: "ALL" or "SUPER" or "PROP". See Remark 4. (Character; Default = ALL)
WHEREID	ID of the mesh refinement region subjected to the refinement criterion referenced by the field CRITID. Must be specified if WHEREMET is "SUPER" or "PROP". If WHEREMET=ALL, this field will be ignored. See Remark 4. (Integer ≥ 0; Default = 0)
SNAPMETH	Method to project, snap or relax new grid pints created on mid-edge or mid-faces on the mesh boundary during the refinement process onto the analysis domain boundary: 0 No projection; New grids are placed in the mid-side of edges. 1 New grid points are projected onto a smooth approximation of the analysis domain boundary interpolated from the initial mesh boundary. (Integer ≥ 0; Default = 0)
MAXLEVEL	Maximum refinement level allowed for each individual element in the mesh. No elements in the mesh will be refined to a level bigger then MAXLEVEL. (Integer > 0; Default = REPEAT)

Remarks:

1. The adaptive mesh refinement occurs when a particular refinement criterion is satisfied. Data for the refinement criterion is specified by the Bulk Data entry HADACRI referenced by the CRITID field.

2. Multiple mesh refinement criteria can be selected in different subsets of the model. To this end, the user needs to define multiple HADAPTL entries with the same ID. Each entry might specify a different criteria (referenced in the CRITID field and defined on the corresponding HADACRI Bulk Data entry) on different subsets of the mesh (defined in the WHEREMET and WHEREID fields).
3. When multiple HADAPTL entries with the same ID are specified, NASTRAN will chose for the REPEAT, SNAPMETH, and MAXLEVEL field the maximum among all multiple instances.
4. The fields WHEREMET and WHEREID refer respectively to the Method to specify the mesh refinement region (subjected to the refinement criterion referenced by the field CRITID and defined with the Bulk Data entry, [HADACRI](#)) and its corresponding ID. For example, WHEREMET=SUPER, WHEREID=3 means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in superelement 3. Likewise, WHEREMET=PROP, WHEREID=5 (see the previous Example) means that local adaptive mesh refinement (with the criteria referenced by the CRITID field) should be effected only in those element with Property ID equal to 5. Finally, WHEREMET=ALL imply mesh refinement in all elements.
5. In partitioned superelements, the HADAPT entry must be specified in the main bulk data section. Entries specified in the Bulk Data Section corresponding to individual parts (sections beginning with BEGIN SUPER) will be ignored.
6. When using regular superelements, the Bulk Data Section must begin with BEGIN SUPER as opposed to BEGIN BULK, in order for the refinement to be appropriately propagated across superelement boundaries. If BEGIN BULK is used, grid points on the superelement boundaries will be duplicated and not shared by the joining superelements.
7. The user should avoid the use of MPC sets 90000000 to 99999999 which are reserved for hanging nodes constraints generated during the adaptive mesh refinement process.
8. The user might need to adjust the VARPHI parameter to ensure proper singular geometric feature detection (such as sharp edges or corners) (See the Parameter, [VARPHI](#)).
9. When SNAPMETH=0, all mid-edge nodes belonging to straight edges are placed on the mid-side of its edge. By contrast, when SNAPMETH=1, mid-edge nodes belonging to the boundary of the mesh are projected to a smooth approximation of the analysis domain boundary interpolated from the mesh boundary.
10. Mid-face nodes belonging to bilinear quadrilateral faces are placed at the baricenter of its face.

## HEATLOS

## Heat Loss Through Convection or Radiation of the Airbag Surface - SOL 700

Defines the heat-transfer model to be used with GBAG or COUPLE. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HEATLOS	HID	HTRID	SUBID	HTRTYPE	HTRTYPID	COEFF	COEFFV		

Example:

HEATLOS	101	83		HTRCONV	2	TABLE	14		
---------	-----	----	--	---------	---	-------	----	--	--

Describer	Meaning
HID	Unique number of a HEATLOS entry. (Integer > 0; Required)
HTRID	Number of a set of HEATLOS entries HTRID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)
SUBID	(Integer ≥ 0, 0) > 0            Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface as defined on the GBAG or COUPLE entries. = 0            HEATLOS definitions are used for the entire surface as defined on the GBAG or COUPLE entries.
HTRTYPE	Defines the type of heat transfer. (Character; Required) HTRCONV    The HTRCONV logic is used to model heat transfer through convection in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface. HTRRAD     The HTRRAD logic is used to model heat transfer through radiation in an air bag. The area of convection through which the energy is transported is equal to the area of the subsurface multiplied by COEFFV. A value of COEFFV = 1.0 will expose the complete subsurface area, while a value of COEFFV = 0.0 will result in no heat transfer through the subsurface.
HTRTYPID	Heat transfer ID. References existing HTRTYPE entry. (Integer > 0; Required)
COEFF	Method of defining the area coefficient. (Character, CONSTANT) CONSTANT    The area coefficient is constant and specified on COEFFV.



Describer	Meaning	
	TABLE	The area coefficient varies with time. COEFFV is the number of a TABLED1 entry giving the variation with time.
COEFFV		The area coefficient or the number of a TABLED1 entry depending on the COEFF entry. ( $0.0 \leq R \leq 1.0$ or $1 > 0, 1.0$ )

### Remarks

1. A combination of multiple HEATLOS with different HTRTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to HEATLOS) or an Eulerian model (COUPLE to HEATLOS). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).
3. For the same BSURF multiple, different types of heat transfer may be defined.
4. A more detailed description can be found in Porosity in Air Bag for more details.

## HGSUPPR

## Hourglass Suppression Method - SOL700

Defines the hourglass suppression method and the corresponding hourglass damping coefficients. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HGSUPPR	HID	PROP	PID	HGTYPE	HGCMEM	HGCWRP	HGCTWS	HGCSOL	
+	RBRCOR	VALUE							

Example:

HGSUPPR	1	SHELL	100	FBV	0.1	0.1	0.1		
+	YES	10000							

Describer	Meaning
HID	Hourglass suppression definition number. (Integer > 0; Required.)
PROP	Property type. (Character; See Remark 1.)
PID	Property number. (Integer > 0; See Remark 1.)
HGTYPE	Hourglass suppression method: (Character; See Remark 2.)
FBV	For shells only (default for shells)
FBS	For shells and solids (default for solids)
Dyna	For solids only
HGCMEM	Membrane damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)
HGCWRP	Warping damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)
HGCTWS	Twisting damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)
HGCSOL	Solid damping coefficient. (0.0 ≤ Real ≤ 0.15; default=0.1)
RBRCOR	Rigid body rotation correction: (Character; default=NO)
NO	No rigid-body rotation correction is applied to hourglass resisting forces.
YES	Rigid-body rotation correction is applied to hourglass resisting forces.
VALUE	Number of steps. (Integer > 0; See Remark 3.)

Remarks:

1. The property type definition and the property number are required. Since property numbers are unique within a certain class of element types, the property type and the property number uniquely define to what elements the hourglass suppression method and coefficients apply. The following property types are valid entries:

BAR	For bar elements
BEAM	For beam elements
BELT	For belt elements
COMP	For composite shell elements
DAMP	For damper elements
ELAS	For spring elements
EULER	For Eulerian elements
ROD	For rod elements
SHELL	For shell elements
SOLID	For solid Lagrangian elements

It must be noted however, that only shell CQUAD4 and Lagrangian CHEXA and CPENTA elements can suffer from undesired hourglass modes. All HGSUPPR entries referring to other types of elements are ignored.

2. There are three types of hourglass suppression methods available in Nastran. These are standard DYNA viscous (DYNA) hourglass damping, the Flanagan-Belytschko Stiffness (FBS) hourglass damping, and the Flanagan-Belytschko Viscous (FBV) hourglass damping.  
 Lagrangian solid elements can address DYNA and FBS suppression; shell elements can address DYNA and FBV suppression. The default for the Lagrangian solid elements is FBS. The default for the shell elements is FBV.
3. The rigid-body rotation correction on the hourglass forces is only necessary in cases where shell elements undergo a large rigid-body rotation. If the RBRCOR field is set to YES, and the VALUE field is left blank, the correction is applied during each time step. If the VALUE field is set to a number, the rotation correction is applied only when the rigid-body rotation would result in a rotation of the element over 90° in less than VALUE time steps. Usually, if the rigid-body rotation correction is necessary; 10000 is a good value. This option saves some CPU time.  
 The RBRCOR option applies to the Key-Hoff shell formulation only; for all other element types and formulations, the option is ignored.
4. The membrane, warping and twisting coefficients apply to shell elements only; for all other element types, the data is ignored. The solid damping coefficient applies to solid Lagrangian elements only; for all other element types, the data is ignored.

## HTRCONV

## Air Bag Convection

Defines the heat transfer through convection for a COUPLE and/or GBAG surface.

Convection is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HTRCONV	HTRID	HTRCFC	HTRCFT	TENV					

Example:

HTRCONV	8		14	293.0					
---------	---	--	----	-------	--	--	--	--	--

Describer	Meaning
HTRID	Unique number of a HTRCONV entry. Referenced from HEATLOS. (Integer > 0; Required)
HTRCFC	Constant heat transfer convection coefficient. See Remark 3. (Real ≥ 0)
HTRCFT	The heat transfer convection coefficient is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer ≥ 0)
TENV	Environmental temperature. (Real > 0; Required)

Remarks:

1. The HTRCONV entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gas constants ( $\gamma$ ,  $R$ ,  $c_v$  and/or  $c_p$ ) have to be defined on the EOSGAM entry.
3. Either HTRCF-C or HTRCF-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

## HTRRAD

## Air Bag Radiation

Defines the heat transfer through radiation for a COUPLE and/or GBAG surface.

Radiation is heat transfer from the air bag to the environment through the air bag surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HTRRAD	HTRID	GASBMI-C	GASEMI-T	TENV	SBOLTZ				

Example:

HTRRAD	2	0.15		293.0	5.676E-8				
--------	---	------	--	-------	----------	--	--	--	--

Descriptor	Meaning
HTRID	Unique number of a HTRRAD entry. Referenced from HEATLOS. (Integer > 0; Required)
GASBMI-C	Constant gas emissivity. See Remark 3. (Real ≥ 0)
GASEMI-T	The gas emissivity is a tabular function of time. The number given here is the number of a TABLED1 entry. See Remark 3. (Integer ≥ 0)
TENV	Environmental temperature. (Real > 0; Required)
SBOLTZ	Stephan-Boltzman constant. (Real > 0; Required)

Remarks:

1. The HTRRAD entry can be referenced from a HEATLOS entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state. Two of the four gasconstants ( $\gamma$ ,  $R$ ,  $c_v$  and/or  $c_p$ ) have to be defined on the EOSGAM entry.
3. Either GASEMI-C or GASEMI-T must be specified.
4. Energy will only transfer out of the air bag if the temperature inside the air bag is higher than the environmental temperature.

## HYBDAMP

## Hybrid Modal Damping for Direct Dynamic Solutions

Specifies hybrid damping parameters.

Format:

1	2	3	4	5	6	7	8	9	10
HYBDAMP	ID	METHOD	SDAMP	KDAMP	PRTEIG				

Example:

HYBDAMP	101	2000	2001	NO	YES				
---------	-----	------	------	----	-----	--	--	--	--

Describer	Meaning
ID	Identification number of HYBDAMP entry (Integer > 0; Required)
METHOD	Identification number of METHOD entry for modes calculation. (Integer > 0; Required)
SDAMP	Identification number of TABDMP1 entry for modal damping specification. (Integer > 0; Required)
KDAMP	Selects modal “structural” damping. See Remark 1. (Character: “Yes” or “NO”; Default = “NO”)
PRTEIG	Print eigenvalue summary from hybrid damping calculation (Character: “Yes” or “NO”; Default = “NO”)

Remarks:

1. For KDAMP = “YES”, the viscous modal damping is entered into the complex stiffness matrix as structural damping.
2. Hybrid damping is generated using modal damping specified by the user on TABDMP entries.

$$BH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} b(\omega_1) & & & \\ & b(\omega_2) & & \\ & & \ddots & \\ & & & b(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

For KDAMP = “YES”

$$KH = [M] \begin{bmatrix} \phi_1 & \phi_2 & \dots & \phi_n \end{bmatrix} \begin{bmatrix} g(\omega_1) \\ g(\omega_2) \\ \vdots \\ g(\omega_n) \end{bmatrix} \begin{bmatrix} \phi_1^T \\ \phi_2^T \\ \vdots \\ \phi_n^T \end{bmatrix} [M]$$

where

$\phi_i$  = modes of the structure

$[M]$  = structural mass matrix

$b(\omega_i)$  = modal damping values,  $b(\omega_i) = g(\omega_i)\omega_i m_i$

$g(\omega_i)$  = twice the critical damping ratio determined from user specified TABDMP entry

$\omega_i$  = natural frequency of mode  $\phi_i$

$m_i$  = generalized mass of mode  $\phi_i$

3. Hybrid damping is based on a modal solution and the mass matrix. Even if user requested all the modes and specified a flat value of G the result is still dependent on the Rank of the Mass matrix. Also note that Hybrid damping does not include residual vectors to compensate for high frequency contribution and other possible massless degrees of freedom effects in the Rank of the mass matrix. PARAM, COUPMASS, 1 may improve the mass rank.
4. HYBDAMP is referenced by the DAMPING and ROTHYBD Bulk Data entries.
5. When hybrid damping is used during FRF component generation, it is recommended to set the KDAMP field to YES on the HYBDAMP entry. See remarks under SDAMPING Case Control description for further details.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## HYDROC

### Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using fluid cavities.

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using fluid cavity data. The HYDROC entry is NOT supported in SOL101 because SOL 101 does not support the Fluid Structure Interface.

Format:

1	2	3	4	5	6	7	8	9	10
HYDROC	SID	PSOLID	PNOM	G	PCH	PLD4	OVRD		
	CID/GRD	HGHTB	HABOV						

Example:

HYDROC	66	33	42.	980.		1			
	99	0.	10.						

Descriptor	Meaning
SID	Load set identification number. (Integer > 0)
PSOLID	PSOLID ID of the cavity. Fluid density will be obtained from the associated MAT10 entry. (Integer > 0); HYDROC requires ACMODL,DIFF (the default value).
PNOM	Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; default = 0.0)
G	Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)
PCH	0 - default (no punch); 1 - Punch Set of structural grids. 2 - Punch Set of structural elements. 3 - Punch both structural grids and structural elements. (Integer ≥ 0)
PLD4	0 - default (no punch); 1 - Punch the resulting PLOAD4 entries. (Integer ≥ 0)
OVRD	0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued. 1 - Allow non unit value of G.
CID/GRD	Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer ≥ 0; a coordinate system ID is used. If Integer < 0; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal error. See Remark 1.; (Integer - default 0 -basic).

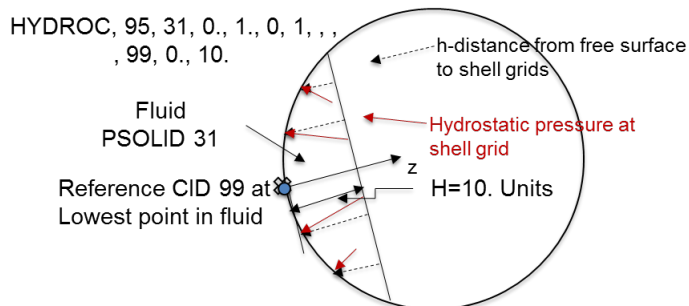
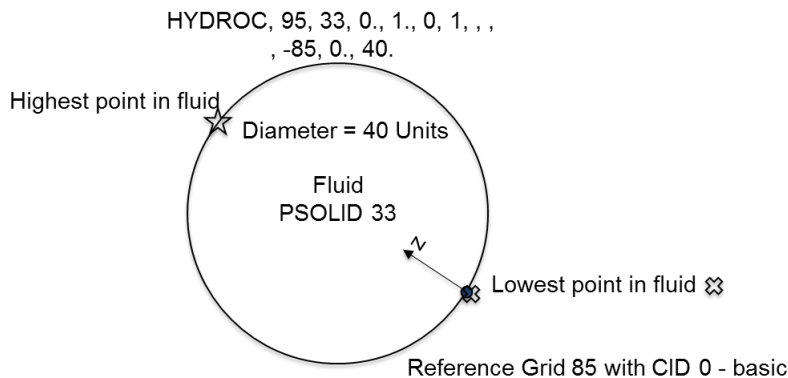


Describer	Meaning
HGHTB	The z-distance to the lowest fluid point measured from reference origin. See Remark 1.. (Real; Default = 0.0)
HABOV	The height of the fluid above the lowest point measured in the z-direction from reference origin. See Remark 1. (Real; Default = 0.0)

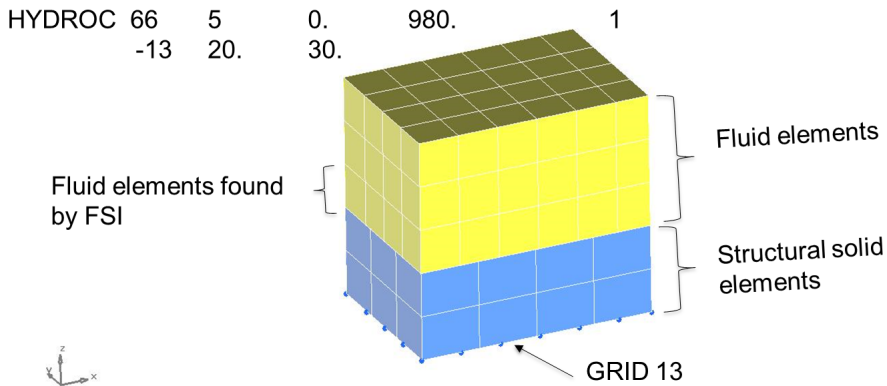
## Remarks:

- On the HYDROC entry, HGHTB along with the z-coordinate direction, are used to determine the location of the “bottom” of the fluid body in the case where the fluid cavity is full and there is no free surface.

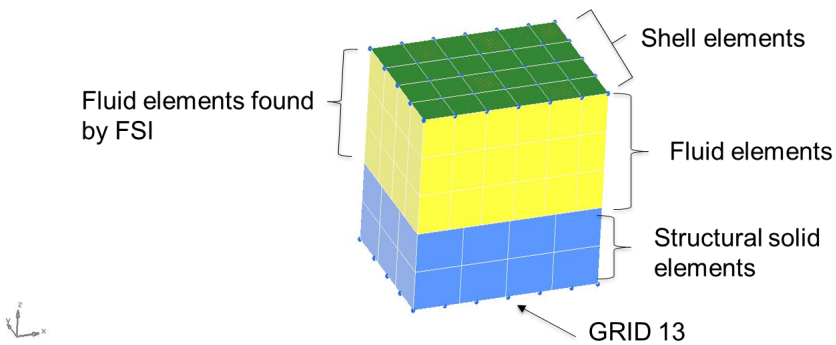
In the case where a free fluid surface has been modeled in the cavity, CID/GID and HGHTB are for reference only. The lowest fluid level is determined as the lowest wetted grid in the FSI coupling measure in the direction defined by CID/GID. From the FSI coupling, free surface formed by examining wetted grids may not be perpendicular to the z-direction of CID/GID. Hence, HABOV will be considered as average free surface and hydrostatic pressure will be computed with respect to HABOV. However, a fatal message will occur if the average free surface specified by HABOV is significantly above the free surface of wetted grids. Also note that some wetted grids/elements may not get loaded if HABOV is too low.



2. The HYDROC entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID
3. Each fluid cavity should have a unique PSOLID entry as HYDROC uses the PSOLID to identify the cavity.
4. The PNOM is applied only from fluid surface to bottom of fluid.
5. HYDROC interaction with the Fluid Structure Interface (FSI) algorithm is shown in the two figures below. The top figure shows a column of fluid over some structural elements. The column of fluid is 30 units high, however, only its bottom surface is in contact with structural elements. Therefore, the FSI will only see a fluid that is 10 units high. A HYDROC entry for this model, with a HABOV = 50 units would fail to compute any hydrostatic loading, because 50 units is well outside the tolerance of the FSI interface, which is at 30 units.



The second figure shows the same fluid column, but a layer of shell elements has been laid on top of the fluid column.



HYDROC	66	5	0.	980.	1
	-13	20.	50.		

In this case, the FSI sees a column of fluid 50 units high. Thus a HABOV = 50 units will generate PLOAD4 entries using a height of 30 units for the fluid. For this model, any HABOV with values (30 units ≤ HABOV ≤ 50 units) will yield valid PLOAD4 entries on the fluid structure interface of the structural solid elements with the height used depending on the value of HABOV.

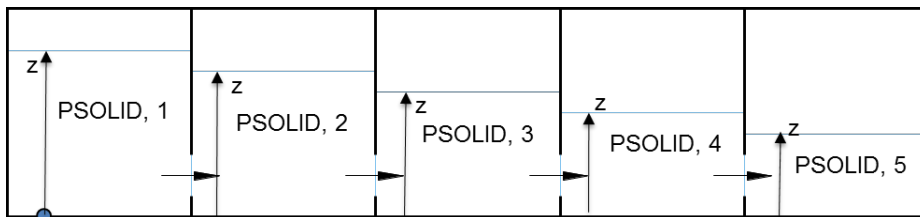
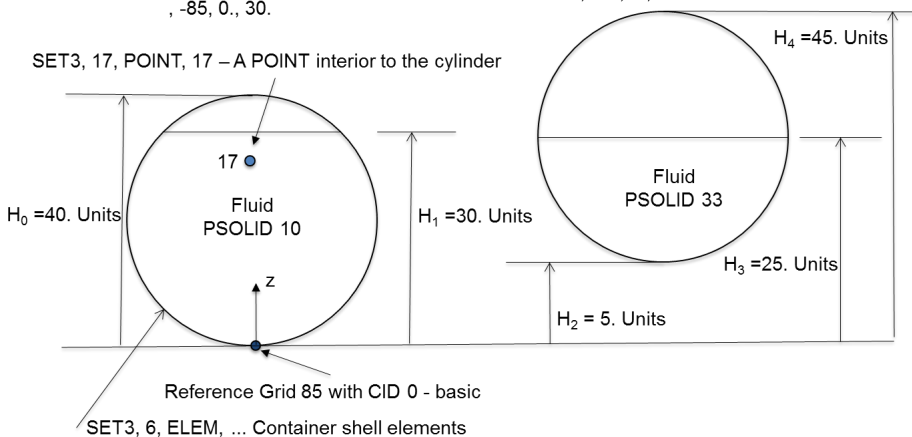
- The figures below show additional HYDROC PNOM rules associated with the FSI interface. Also note how fluid cavities are defined by PSOLID ID.

HYDROS (1) applies a hydrostatic loading of 55. + 0.\*h\*0. interior to the cylinder with POINT 17  
 HYDROC (2) applies a hydrostatic loading 0. + ρ\* h \* 980.  
 HYDROC (3) applies a hydrostatic loading 42. + ρ\* h \* 980. – above the fluid there is no pressure of 42. units

1. HYDROS, 66, 55., 0., 0., 6, ,1 , ,  
 , -85, 0., 40., 17

2. HYDROC, 66, 10, 0., 980., 0, 1, , ,  
 , -85, 0., 30.

3. HYDROC, 66, 33, 42., 980., 0, 1, , ,  
 , -85, 5., 25.



Reference origin with CID

Staggered grid for flooding simulation.

**HYDROS**

**Hydrostatic Pressure Loading on Surface and Faces of Shell or Solid Elements using element or grid sets**

Defines a hydrostatic pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element using element set data.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
HYDROS	SID	PNOM	RHO	G	SETE	SETG	PLD4	OVRD	
	CID/GRD	HGHTB	HABOV	CNTL					

Example:

HYDROS	77	55.	0.08	17.	300		1		
	200		97.	83					

Describer	Meaning
SID	Load set identification number. (Integer > 0)
PNOM	Nominal or atmosphere pressure. If G=0.0, PNOM only is applied. (Real; Default = 0.0)
RHO	Mass density. (Real > 0.0)
G	Acceleration (e.g. of gravity) in the direction from HABOV toward the origin defined by the CID/GRD field. (Real; Default = 1.0)
SETE	The ID of a SET3 entry using (DES= ELEM) or SET1 to identify the element IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer ≥ 0)
SETG	The ID of a SET3 entry using (DES= GRID) or SET1 to identify the element grid IDs of structure elements at fluid-structure interface used for hydrostatic loading. (Integer ≥ 0)
PLD4	0 - default (no punch); 1 - Punch the resulting PLOAD4 entries. (Integer ≥ 0)
OVRD	0-default. If PARAM, INREL, is present and G not equal to 1.0, then a fatal error message will be issued. 1 - Allow non unit value of G.
CID/GRD	Coordinate ID of a reference coordinate system to measure the coordinate z-direction. If Integer ≥ 0; a coordinate system is used. If Integer < 0; a GRID ID is used with the z-direction of the GRID, CD field. Note: A spherical coordinate system will result in a fatal. See Remark 1.; (Integer - default 0 -basic). Note: A spherical coordinate system will result in a fatal error.
HGHTB	The z-distance to the lowest fluid point measured from reference origin. (Real; Default = 0.0)



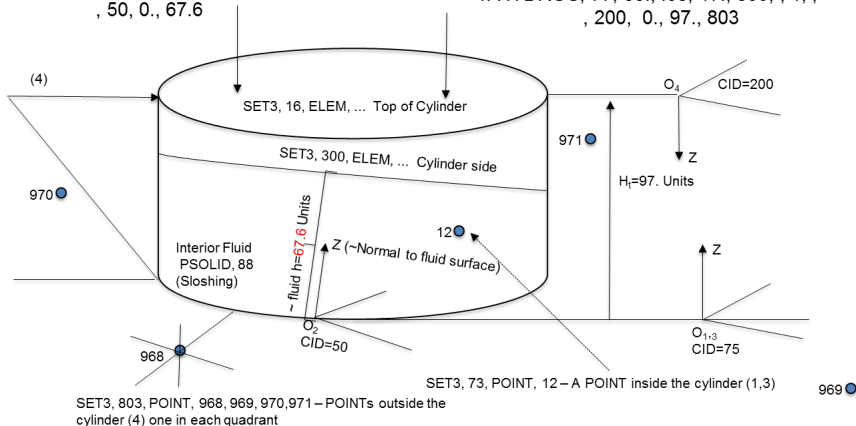
Describer	Meaning
HABOV	The height of the fluid above the lowest point measured in the z-direction from reference origin. (Real; Default = 0.0)
CNTL	SET3 ID of control grids to get pressure application direction. (Integer ≥0, see Remark 2.).

Remarks:

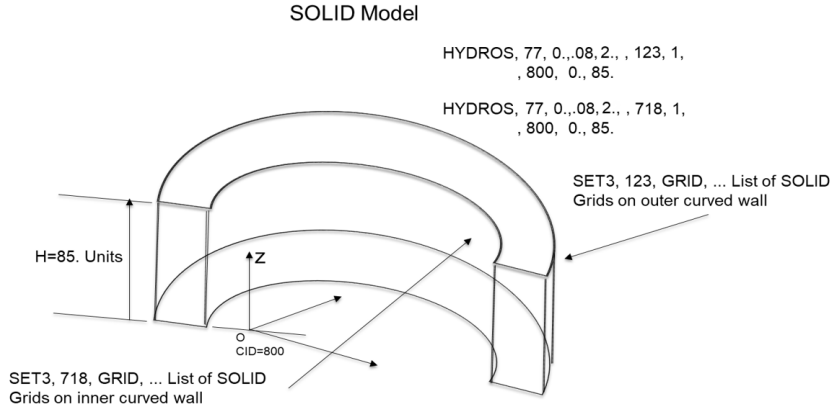
- The HYDROS entry will internally be converted to an equivalent set of PLOAD4 entries with the same load SID.

HYDROS (1) applies a hydrostatic loading of  $25. + .08 \cdot h \cdot 0$ . interior to the cylinder with POINT 12  
 HYDROC (2) applies a hydrostatic loading  $0. + \rho \cdot h \cdot 1$ ; h – measured normal to sloshed surface  
 HYDROS (3) applies a hydrostatic loading  $55. + .08 \cdot h \cdot 980$ . (h is constant 97. units for this set)  
 HYDROS (4) applies a hydrostatic loading  $55. + .08 \cdot h \cdot 17$ . for all elements on exterior cylinder side.

- |   |  |
|---|--|
| 1. HYDROS, 77, 25., .08, 0., 300, , 1, ,<br>, 75, 0., 97., 73 | 3. HYDROS, 77, 55., .08, 980., 16, , 1, ,<br>, 75, 0., 97., 73   |
| 2. HYDROC, 77, 88, 0., 1., 0, 1, , ,<br>, 50, 0., 67.6        | 4. HYDROS, 77, 55., .08, 17., 300, , 1, ,<br>, 200, 0., 97., 803 |



SET3, 803, POINT, 968, 969, 970, 971 – POINTS outside the cylinder (4) one in each quadrant



2. The pressure direction can be defined with or without CNTL entry in HYDR0S card for closed and open geometries.
  - For closed geometry like sphere, cylinder, cube, etc., SET3 id should have only one grid defined inside the geometry. Use of inside grid, will result in internal pressure to be applied. If the external pressure needs to be applied, then change the sign of Pnom and Gravity in the HYDR0S card. ( $P = P_{nom} + \rho \cdot g \cdot h$ )
  - For open geometry, the direction of the pressure is based on the relation between the direction of control point grid to the element and the element normal. Refer 1(a) and 1(b) for direction of pressure for an open geometry.
  - Without control point defined in the HYDR0S card the pressure direction is aligned with the element normal direction. If the element normal is outward, then the pressure direction is outward and vice versa.

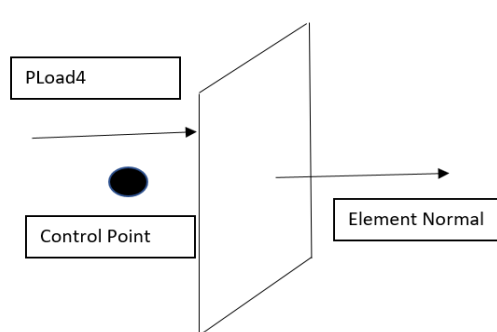


Figure 1 (a)

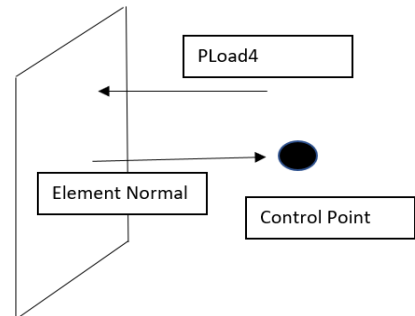


Figure 1 (b)

## HYDSTAT

## Hydrostatic Preset of Density in Euler Elements

Initializes the Euler element densities in accordance to a hydrostatic pressure profile. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
HYDSTAT	HID	MID	GID	CID	XCG	YCG	ZCG	PATM	

Example:

HYDSTAT	101	4			0.0	0.0	0.0	100000.	
---------	-----	---	--	--	-----	-----	-----	---------	--

Describer	Meaning
HID	Identification number of the HYDSTAT entry referenced from the COUPLE entry. (Integer > 0; Required)
MID	Material MATDEUL id to which the hydrodynamic pressure profile will be applied. (Integer > 0; Required)
GID	Number of a grid point at the free surface. See Remark 4. (Integer ≥ 0)
CID	Local coordinate system. See Remark 4. (Integer ≥ 0)
XCG,YCG, ZCG	Coordinates of a point at the free surface. See Remark 4. (Real)
PATM	Pressure at free surface. (Real; Required)

Remarks:

1. It is assumed that each Euler domain contains at most two Eulerian materials and includes the GRAV entry. One material has to be a fluid using EOSPOL the other a gas or void. This EOSPOL material is given by the MID entry. The interface between gas and fluid is the free surface and is assumed to be normal to the gravity vector as specified on the GRAV entry. For example if the gravity vector points in the z-direction then the interface between the gas and the fluid has to be horizontal.
2. The hydrostatic preset changes the density of the fluid like material in order to conform to the hydrostatic preset. It overrules the material densities as specified on the TICEL and TICVAL entries. Densities of the gas like material are not changed.
3. The free surface has to match with material fractions as defined in the initialization of Euler elements by the TICEL and TICEUL entries. The hydrostatic preset only changes densities, it does not change material fractions.
4. There are two options to enter the location of the free surface. The first option is to enter a grid point number. In that case the fields CID and XCG-ZCG have to be left blank. The GRIDPOINT entry already has the option of using a local coordinate system. When coordinates are used then the field GID has to be left blank.

5. If there is no structural grid point indicating the free surface then a new grid point can be defined that will only be used for determining the free surface level. If the Gravity vector points in the z-direction, only the z-coordinate of the grid point will be used. The x and y ordinate can be chosen arbitrarily. Similar remarks hold when the gravity vector is in one of the other coordinate directions. The same holds when using coordinates instead of a grid point.
6. PATM should be equal to the pressure in the air.
7. When coupling surfaces are present then the HYDSTAT ID needs to be referenced by at least one coupling surface. For each coupling surface a different HYDSTAT entry can be defined. Several COUPLE entries can refer to the same HYDSTAT ID. If no HYDSTAT ID is specified on a COUPLE entry then Euler elements associated to this coupling surface will not be initialized with a hydrostatic preset.



## IMPCASE

Defines a collection of imperfection cases

Defines a collection of imperfection cases by listing of identification numbers of IMPGEOM entries. Used in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
IMPCASE	ID	IMPFID1	IMPFID2	...	IMPFIDi				

Example:

IMPCASE	5	1	2						
---------	---	---	---	--	--	--	--	--	--

Describer	Meaning
ID	Identification number (integer > 0, required)
IPMFIDi	Identification number of imperfection case (integer >0, required)

Remarks:

1. When the ID is referenced by case control IMPERFECT command, the list of IMPGEOM entries are used to apply geometric imperfections.
2. "THRU" can be used to specify IMPFIDi.

## IMPGEOM

### Defines Geometric Imperfection

Defines geometric imperfection by selecting subcases, steps, mode numbers or increments from imperfection input files. Used in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
IMPGEOM	IPMFID			SETID	SCALE		UNIT	MODID	
	SUBCASE1	STEP1	MODEINC1	SETID1	SCALE1		UNIT1	MODID1	
	SUBCASE2	STEP2	MODEINC2	SETID2	SCALE2		UNIT2	MODIDi	
	etc.								

Example:

IMPGEOM	1								
	1	3	1		0.011		34		
	1	3	2		0.002		34		

Describer	Meaning
IPMFID	Identification number of imperfection case (integer >0, required)
SETID	Default of SETIDi (integer ≥ 0, default=0)
SCALE	Default of SCALEi (real, default=0.0)
UNIT	Default of UNITi (integer ≥ 0, default=0)
MODID	Default of MODIDi (integer ≥ -1, default=-1), see remark 11.
SUBCASEi	Identification number of a subcase (integer ≥ 0, default=0)
STEPi	Identification number of a step (integer ≥ 0, default=0)
MODEINCI	Mode number if the selected step by SUBCASEi and STEPi or subcase by SUBCASEi is a result of buckling analysis or modal analysis, increment number if the selected result is from a static analysis. (integer ≥ 0, default=0)
SETIDi	Identification number of a SET1 or SET3 bulk entry (integer ≥ 0, default = 0). When it is defined, only the grid points with the id listed in SET1 or SET3 have geometric imperfection effect. If a SET3 is used, it must be GRID type.
SCALEi	Scale factor (real, default = 0.0)
UNITi	Imperfection file unit number specified by ASSIGN HDF5IN, INPUTT2 or IMPFIN FMS command.
MODIDi	Module ID (integer ≥ -1, default=-1), see remark 11.

Remarks:

1. All IMPGEOM identification numbers must be unique. An IMPGEOM entry is referenced by IMPERFECT case control directly or by an IMPCASE bulk entry.
2. Geometric imperfection effect is applied to grid points as coordinates variation, the variation is superposed by eigenvectors of multiple modes or displacement of increments defined in multiple continuation lines.
3. An imperfection input file can be an MSC Nastran HDF5 result file, a Nastran op2 file or a text based file called IMPF. To use it, an ASSIGN HDF5IN, INPUTT2 or IMPFIN FMS command is required.
4. The imperfection input file of HDF5 or op2 can be a result file of SOL 101, 103, 105 or a SOL 400 with ANALYSIS type of BUCK, MODES, STATIC, NLSTATIC or NLTRAN, and must contain eigenvectors or displacement results.
5. IMPF file format is described in remark 12. It can be DISP format or GEOM format. If an IMPF file is used in an IMPGEOM entry, only one continuation line of IMPGEOM is allowed, and SUBCASEi, STEPi and MODEINCi are ignored. If format is GEOM, SCALE and SCALEi are ignored too.
6. SUBCASEi must be specified if the imperfection input file is an op2 or hdf5 file.
7. STEPi is ignored when the imperfection input file is a result of SOL 101, 103 or 105. It must be specified for a SOL 400 result file.
8. MODEINCi is ignored when the file is a result of SOL 101 or a linear static step of SOL 400. It is required for other cases if the imperfection file is an hdf5 or op2 file.
9. UNITi must be specified if UNIT is 0 or not specified.
10. Geometric imperfection is applied to residual structure for superelement models. In the case of part superelement, it is advised to adjust boundary grid point searching tolerance due to geometry changing. This can be done by adding or modifying SEBULK or SECONCT entries.
11. MODID and MODIDi are only used when modules are present. -1 means the whole model, 0 or a positive number is a module identification number.
12. Format of IMPF file

The IMPF file is a csv-like file, delimiters of a line can be spaces, a comma or a tab space. A line starting with a "\$" or "#" is a comment line. If the first line is a comment line and contains string "GEOM" or "GRID", then it is a GEOM file, i.e., the grid coordinates provided here will replace the grid coordinates in the FE model; otherwise it is a DISP file, i.e., the displacements provided in the file will be superimposed to the coordinates of the corresponding grids.

For each line, the first column is grid point id, followed by x,y and z. For DISP format, x,y and z are coordinate variations from the original coordinates, and coordinate system is MSC Nastran global system, this means that x,y,z are in the coordinate system indicated by CD field of GRID entries in the original input file. For GEOM format, x,y and z are "imperfect" coordinates of the grid points, coordinate system is MSC Nastran input system, this means the values of x,y,z respect CP field of the GRID entries in the original input file.

When modules are present, BEGIN MODULE=modid and ENDMODULE can be used to specify module id for a section of grids.

Below is a DISP format of IMPF file.

disp\_1.impf

```
-----
18 , 0.000000000000E+00 , 0.000000000000E+00 , 0.000000000000E+00
19 , 0.000000000000E+00 , 0.000000000000E+00 , 0.000000000000E+00
20 , 1.882586315422E-04 , -1.382379038267E-03 , -5.111620104649E-05
21 , 3.080245224512E-04 , -1.360710847814E-03 , -5.111620104647E-05
22 , 4.254461607540E-04 , -1.328686826189E-03 , -5.111620104646E-05
23 , 5.396298968822E-04 , -1.286550695534E-03 , -5.111620104647E-05
24 , 6.497067236577E-04 , -1.234623137246E-03 , -5.111620104650E-05
25 , 7.548388899626E-04 , -1.173299351397E-03 , -5.111620104657E-05
26 , 8.542262765314E-04 , -1.103046049026E-03 , -5.111620104668E-05
27 , 9.471124853400E-04 , -1.024397900185E-03 , -5.111620104681E-05
28 , 1.032790596252E-03 , -9.379534647748E-04 , -5.111620104691E-05
29 , 1.110608547111E-03 , -8.443706371464E-04 , -5.111620104698E-05
30 , 1.179974096322E-03 , -7.443616391243E-04 , -5.111620104702E-05
31 , 1.240359330172E-03 , -6.386875995705E-04 , -5.111620104705E-05
```

Below is a GEOM format of IMPF file.

geom\_example.impf

```
-----
# GEOM
1, 0.000000000000E+00, 0.000000000000E+00, 0.000000000000E+00
2, 1.000000014901E-01, 4.367130542590E-24, 1.464331138939E-08
3, 2.000000029802E-01, 7.552163438029E-24, 3.787209513278E-08
4, 2.999999821186E-01, 8.873575793200E-24, 5.096678857312E-08
5, 3.999999463558E-01, 0.000000000000E+00, 0.000000000000E+00
6, 4.99999701977E-01, 0.000000000000E+00, 0.000000000000E+00
7, 5.99999642372E-01, 0.000000000000E+00, 0.000000000000E+00
8, 6.99999880791E-01, 0.000000000000E+00, 0.000000000000E+00
9, 7.999998927116E-01, 0.000000000000E+00, 0.000000000000E+00
10, 8.999999165535E-01, 0.000000000000E+00, 0.000000000000E+00
11, 9.999999403954E-01, 0.000000000000E+00, 0.000000000000E+00
12, 1.099999904633E+00, 0.000000000000E+00, 0.000000000000E+00
13, 1.199999928474E+00, 0.000000000000E+00, 0.000000000000E+00
14, 1.299999952316E+00, 0.000000000000E+00, 0.000000000000E+00
```

Impf file with modules, mod\_example.impf

```
-----
----
# below lines are in module 0
```

```
1, 0.01, 0.0, 0.0
2, -0.001, -1.e-3, 0.0
.....
# below are for module 10
begin module=10
    1,      -1.0000000E-2, 0.0000000E+00, 0.0000000E+00
    2,      -7.0000000E-02, 0.0000000E+00, 0.0000000E+00
    3,      -1.3900000E-01, 0.0000000E+00, 0.0000000E+00
.....
endmodule
# below are for module 20
BEGIN MODULE 20
    1,      -1.0000000E-3, 0.0000000E+00, 0.0000000E+00
    2,      -7.10E-02, 0.0000000E+00, 0.0000000E+00
    3,      -1.3900000E-01, 0.0000000E+00, 0.0000000E+00
ENDMODULE
.....
```

**INCLUDE**

## Insert External File

Inserts an external file into the input file. The INCLUDE statement may appear anywhere within the input data file.

**Format:**

```
INCLUDE 'filename'
```

**Describer:**

filename	Physical filename of the external file to be inserted. The user must supply the name according to installation or machine requirements. It is recommended that the filename be enclosed by single right-hand quotation marks.
----------	---

**Example:**

The following INCLUDE statement is used to obtain the Bulk Data from another file called MYBULK.DATA:

```
SOL 101  
CEND  
TITLE = STATIC ANALYSIS  
LOAD = 100  
BEGIN BULK  
INCLUDE 'MYBULK.DATA'  
ENDDATA
```

**Remarks:**

1. INCLUDE statements may be nested; that is, INCLUDE statements may appear inside the external file. The nested depth level must not be greater than 10.
2. The total length of any line in an INCLUDE statement must not exceed 72 characters. Long file names may be split across multiple lines. For example the file:

```
/dir123/dir456/dir789/filename.dat
```

may be included with the following input:

```
INCLUDE '/dir123  
                  /dir456  
                  /dir789/filename.dat'
```

3. See the for more examples.

## INFLCG

## Airbag Cold Gas Inflator Model

Defines the cold gas-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLCG	INFLID	TANKVOL	INITPRES	INITTEMP	INITMAS	$\gamma$ GASNAM	$C_v$	R	
	$C_p$								

Example:

INFLCG	11	0.875	131325	293	1.37	1.4		286	

Describer	Meaning
INFLID	Unique number of an INFLCG entry. Referenced from ABINFL. (Integer > 0; Required)
TANKVOL	Tank Volume. (Real > 0; Required)
INITPRES	Initial tank pressure. (Real > 0; Required)
INITTEMP	Initial tank temperature. (Real > 0; Required)
INITMAS	Initial gas mass of inflator. (Real > 0; Required)
$\gamma$ , GASNAM	Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or 1)
$C_v$	Specific heat at constant volume. See Remark 6. (Real > 0)
R	Gas constant. See Remark 6. (Real > 0)
$C_p$	Specific heat at constant pressure. See Remark 6. (Real > 0)

Remarks:

1. The INFLCG entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either INITPRESS or INITMASS has to be specified, but not both. The relation between INITMASS and INITPRESS is given by

$$INITPRES = R \frac{INITMAS}{TANKVOL} INITTEMP$$

4. The cold gas inflator is a reservoir filled with high pressure gas. It is assumed that the volume stays constant at TANKVOL. The mass inside the inflator will steadily decrease due to flow into the Euler domain or to a GBAG. Due to inertia it can happen that the pressure of the inflator becomes less than the outside pressure. In that case some inflow into the inflator occurs. Transport between inflator and the Euler domain or GBAG is based on the constancy of total temperature. This is equivalent to the pressure method.
5. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
6. Specify only two of the four gas constants. They are related as:

$$\nu = \frac{c_p}{c_v} \quad R = c_p - c_v$$



## INFLFRC

## Hybrid Inflator Gas Fraction Definition

Defines the gas fractions as a function of time for hybrid inflators. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLFRC	FRACID	TYPE							
	"TIME"	TIME1	FRAC1	FRAC2	FRAC3	etc.			
	"TIME"	TIME2	FRAC1	FRAC2	FRAC3	etc.			

Example:

INFLFRC	12								
	TIME	14.0E-3	11.342	13.391	9.626	57.019	7.31	1.312	
	0.0	0.1	0.2						
	TIME	14.1E-3	43.332	6.2817	4.5155	26.747	3.4291	0.4898	
	15.077	0.0	0.4						

Descriptor	Meaning
FRACID	Unique number of an INFLFRC entry.(Integer > 0, required.)
TYPE	Specifies whether mass fractions or molar fractions will be given (Character, default=MASS.)  MASS The fractions on INFLFRC are mass fractions.  MOLAR The fractions on INFLFRC are molar fractions. See Remark 5.
TIMEID	Defines a new line of data (Character, required.)  TIME Specifies that data for a new time increment will be given. See Remark 3.
TIMEi	Time for which the gas fractions are given. (Real≥0.0, required.)
FRACi	Fraction of gas i at the specified time..See Remark 7. (Real≥0.0, required.)

Remarks:

1. The INFLFRC entry is referenced with FRACID from an AIRBAG entry with the option "INFLATOR" or from INFLHB.
2. Fraction values of the inflowing gas will be linearly interpolated between the specified time increments.
3. Use as many continuation lines as necessary to completely define the gas fractions. The data for a time step are preceded by a TIME keyword. Missing entries will be set to 0.0.

4. The order of the gases for which the fractions are specified is identical to the order in which the gases are specified on the AIRBAG entry with the option "INFLATOR".
5. At least one line of gas fractions must be given.
6. If molar fractions (TYPE=MOLAR) are to be used, the universal gas constant must be specified through PARAM, UGASC.
7. At least one of the fractions for each time step must have a value greater than 0.0.
8. Fractions for each timestep should add up to 1.0. If this is not the case, they will be scaled so that they do.

## INFLGAS

### Inflator Gas Definition

Defines a thermically ideal gas to be used with a standard or hybrid inflator. Used in SOL 700 only.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
INFLGAS	GASNAM	TYPE	VALUE	CPGAS	V1	V2	V3	V4	

Example:

INFLGAS	101	RSPEC	189.	CONSTAN T	846.				
---------	-----	-------	------	--------------	------	--	--	--	--

Describer	Meaning	
GASNAM	Unique number of an INFLGAS entry. (Integer; Required)	
TYPE	Specific gas constant or molar weight specified. (Character; RSPEC)	
	RSPEC	Specific gas constant
	MOLWT	Molar weight, see Remark 2.
VALUE	Value of the variable TYPE. (Real > 0; Required)	
CPGAS	The variation of the specific heat constant at constant pressure. (Character; CONSTANT)	
	CONSTANT	The specific heat is constant and specified in V1.
	TABLE	The specific heat constant is temperature-dependent. V1 is the number of a TABLED1 entry giving the variation of the specific heat with the temperature.
	POLY	The specific heat constant is temperature-dependent. V1 through V4 are the coefficients of a polynomial expression, see Remark 3.
V1	The specific heat constant, the number of a TABLED1 entry or the first polynomial coefficient, depending on the value of CPGAS. (Real or Integer > 0; Required)	
V2, V3, V4	Coefficients of polynomial expression when CPGAS equals POLY. (Real; 0.0)	

Remarks:

1. INFLGAS can be referenced by an INFLTR, INFLHB or INITGAS entry.
2. When the molar weight is given, the universal gas constant  $R_{uni}$  must be specified using PARAM, UGASC, so that:

$$R_{spec} = R_{uni} / MOLWT$$

3. A polynomial expression for  $c_p$  is given by:

$$c_p(t) = V1 + V2 \cdot T + V3 \cdot T^2 + \frac{V4}{T^2}$$

4. The specific heat constant at constant volume  $c_v$  is calculated from the specific heat constant at constant pressure  $c_p$ , the universal gas constant and the molecular weight according to:

$$c_v = c_p(T) - R_{spec}$$

5. The ratio of specific heats is given as:

$$\gamma = c_p/c_v$$

## INFLHB

## Hybrid Inflator Model

Defines the hybrid-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLHB	INFLHID	MASFLRT	TBMPT	TEMPC	FRAC				
	GASNAM1	GASNAM2	GASNAM3	-etc.-					

Example:

INFLHB	9	15		650.	12				
	22	25	3						

Describer	Meaning
INFLID	Unique number of an INFLHB entry. (Integer > 0; Required)
MASFLRT	Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)
TEMPT	Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0)
TEMPC	Constant value of the temperature of the inflowing gas. See Remark 3. (Real > 0.0)
FRAC	Number of an INFLFRC entry specifying the fractions of the inflowing gas as a function of time. (Integer > 0; Required)
GASNAMi	ID of an INFLGAS entry. See Remark 4. (Integer)

Remarks:

1. The INFLHB entry can be referenced from a [ABINFL](#) entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. Either TEMPC or TEMPT must be specified. The INFLHB entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition as opposed to the dynamic temperature that refers to the temperature of the moving gas.
4. At least one inflator gas must be specified using an INFLGAS entry. There is no limit to the number of inflator gases per INFLHB.

## INFLTNK

## Airbag Tanktest Inflator Model

Defines the Tanktest-inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLTNK	INFLID	METHOD	TPTABLE	TANKVOL	INFLMAS	INITPRES	ENDTPRES	INITTEMP	
	ENDTEMP	$\gamma$	$c_v$	$R$	$c_p$	IPTABLE	INFLPRES	INFLTEMP	
	INFLAREA	SFTP	SFIP						

Example:

INFLTNK	111	AVTEMP	10	0.12	0.01	0.0			
		1.4		286.					

Describer	Meaning
INFLID	Unique number of an INFLTNK entry. (Integer > 0; Required)
METHOD	Method of calculating the mass-flowrate: (Character; Required)
AVTEMP	Average Temperature Method
INFPRES	Inflator Pressure Method
TPTABLE	Table number of a TABLED1 entry specifying the tank pressure as a function of time. (Integer > 0; Required)
TANKVOL	Tank Volume. (Real > 0; Required)
INFLMAS	Total gas mass generated by inflator. (Real > 0; Required)
INITPRES	Initial tank pressure. See Remark 3. (Real > 0; Required)
ENDPRES	End tank pressure. See Remarks 4. and 5. (Real > 0; Required)
INITTEMP	Initial tank temperature. See Remark 5. (Real > 0; Required)
ENDTEMP	End tank temperature. See Remark 5. (Real > 0; Required)
$\gamma$	Ratio of specific heat constants. See Remark 7. (Real > 0)
$c_v$	Specific heat at constant volume. See Remark 7. (Real > 0)
$R$	Gas Constant. See Remark 7. (Real > 0; Required)
$c_p$	Specific heat at constant pressure. See Remark 7. (Real > 0)
IPTABLE	Table number of a TABLED1 entry specifying the inflator pressure as a function of time. See Remark 5. (Integer > 0; Required)
INFLPRES	Initial inflator pressure. See Remarks 5. and 6. (Real > 0; Required)
INFLTEMP	Temperature of inflowing gas: See Remark 5. (Real > 0 or Character; ATM)

Describer	Meaning	
	ATM	Use average temperature of AVTEMP method.
	Real value	User specified temperature.
INFLAREA	Total area of inflator holes. See Remark 5. (Real > 0; Required)	
SFTP	Scale factor for tank pressure. See Remark 5. (Real > 0; 1.0)	
SFIP	Scale factor for inflator pressure. See Remark 5. (Real > 0, 1.)	

Remarks:

1. The INFLTNK entry can be referenced from an ABINFL entry.
2. When used in an Euler coupled analysis, the entry can only be used with the single material hydrodynamic Euler solver using an EOSGAM (ideal gas) equation of state.
3. The initial tank pressure entry (INITPRES) is interpreted as an absolute pressure and used to define reference pressure at  $t=0$  in the tank. The different between INITPRES and the pressure value at  $t=0$  from the table will be added to the entire pressure curve of TPTABLE.
4. The end tank pressure entry (ENDPRES) is interpreted as an absolute pressure at  $t=tend$  of tank pressure table (TPTABLE). This value is used for calculation of total generated mass in the tank.
5. This field must be specified only when Inflator Pressure Method (INFPRES) is defined in the METHOD field.
6. The initial inflator pressure entry (INFLPRES) is interpreted as an absolute pressure and used to define reference pressure at  $t=0$  in the inflator. The different between INFLPRES and the pressure value at  $t=0$  from the table will be added to the entire pressure curve of IPTABLE.
7. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

## INFLTR

## Airbag Inflator Model

Defines the inflator characteristics of a COUPLE and/or GBAG subsurface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INFLTR	INFLID	MASFLRT	TEMPT	TEMPC	$\gamma$ GASNAME	$c_v$	$R$	$c_p$	
	MID								

Example:

INFLTR	5	100		907.0	1.5	283.0			
	0	0	0						

Describer	Meaning
INFLID	Unique number of an INFLTR entry. (Integer > 0; Required)
MASFLRT	Table number of a TABLED1 entry specifying the massflow-rate as a function of time. (Integer > 0; Required)
TEMPT	Table number of a TABLED1 entry specifying the static temperature of the inflowing gas as a function of time. See Remark 3. (Integer > 0)
TEMPC	Constant value of the static temperature of the inflowing gas. See Remark 3. (Real > 0)
$\gamma$ , GASNAM	Ratio of specific heat constants if real. Name of an INFLGAS entry if character. See Remarks 4. and 5. (Real > 0 or Character)
$c_v$	Specific heat at constant volume. See Remark 5. (Real > 0)
$R$	Gas constant. See Remark 5. (Real > 0)
	Specific heat at constant pressure. See Remark 5. (Real > 0)
MID	Material MATDEUL ID of the Inflator material. See Remark 2. Only used for MMHYDRO solver. (Integer $\geq$ 0)

### Remarks:

1. The INFLTR entry can be referenced from an ABINFL entry.
2. When used in combination with the single material hydrodynamic Euler solver an EOSGAM (ideal gas) equation of state is required. In that case the material number MID can be left blank. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM. The Multi-material solver does not allow the use of gas fractions.



3. Either TEMP-C or TEMP-T must be specified. The INFLTR entry uses the specified temperature as the static temperature of the inflowing gas. In literature the static temperature is also known as total-, rest-, or stagnation temperature and refers to the temperature of the gas when brought to rest from its moving condition, as opposed to the dynamic temperature which refers to the temperature of the moving gas.
4. If this field contains a real entry real or is left blank, the inflator gas constants are given on the INFLTR entry itself, see Remark 5. Otherwise, the entry will be read as the name of an INFLGAS entry. In this case, the remaining entries must be left blank.
5. Specify only two of the four gas constants. They are related as:

$$\gamma = \frac{c_p}{c_v} \quad R = c_p - c_v$$

## INITGAS

## Gasbag or Coupling Surface Initial Gas Fraction Definition

Specifies the initial gas composition inside a gasbag or Euler coupling surface. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
INITGAS	INTID		GASNAM1	FRAC1	GASNAM2	FRAC2	-etc.		

Example:

INITGAS	4		14	0.4	32	0.11			
---------	---	--	----	-----	----	------	--	--	--

Describer	Meaning
INTID	Unique number of an INITGAS entry. (Integer > 0; Required)
GASNAMi	ID of an INFLGAS entry. See Remark 3. (Integer > 0)
FRACi	Mass fraction of gas i. See Remark 4. (Real ≥ 0.0)

Remarks:

1. The INITGAS entry can be used to specify the initial gas composition for a gasbag or for an Eulerian coupling surface. The INTID must be referenced either from a GBAG cad or a COUPLE entry.
2. Use as many continuation lines as necessary to completely define the gas fractions.
3. At least one INFLGAS reference must be given.
4. Fractions should add up to 1.0. If this is not the case, they will be scaled so that they do.

## IPSTRAIN Initial Equivalent Plastic Strain Values in SOL 400

Defines initial equivalent plastic strain values. This is the initial plastic strain option used in SOL 400 only. (NOTE: This entry is SINGLE FIELD ONLY! Double field will fatal)

Format:

1	2	3	4	5	6	7	8	9	10
IPSTRAIN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

Example:

IPSTRAIN	2001	2020	1	4	1	5	0.025		
----------	------	------	---	---	---	---	-------	--	--

Describer	Meaning
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

Remarks:

1. This entry only applies to SOL 400 advanced nonlinear elements (selected with PSHLN1, PSLDN1, etc.) and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.

## IPSTRN Initial Equivalent Plastic Strain Values in SOL 600

Defines initial equivalent plastic strain values. This is the MSC Marc's initial plastic strain option used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
IPSTRN	EID1	EID2	INT1	INTN	LAY1	LAYN	STRAIN		

Example:

IPSTRN	2001	2020	1	4	1	5	0.025		
--------	------	------	---	---	---	---	-------	--	--

Describer	Meaning
EID1	First Element ID to which these strains apply. (Integer > 0)
EID2	Last Element ID to which these strains apply. (Integer; Default = EID1)
INT1	First Integration point for which the strain applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the strain applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the strain applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRAIN	Equivalent plastic strain value at start of analysis. (Real; Default is 0.0)

Remarks:

1. This entry only applies to SOL 600 advanced nonlinear elements and is ignored for other solutions.
2. This entry is normally used for metal forming and represents the amount of plastic deformation that the model was previously subjected to. It is used in work (strain) hardening models.
3. For SOL 600 only, if the extra precision available with large field formats is not required and small field precision is adequate, the header IPSTRAIN may be used instead of IPSTRN.

## ISTRESS Initial Stress Values in SOL 400 and SOL 600

Defines initial stress values. This is used in SOL 400 and MSC Marc's initial stress option via SOL600 only.

Format:

1	2	3	4	5	6	7	8	9	10
ISTRESS	EID1	EID2	INT1	INTN	LAY1	LAYN	STRESS1	STRESS2	
	STRESS3	STRESS4	STRESS5	STRESS6	STRESS7				
	"COORD"	CID							

Example:

ISTRESS	2001	2020	1	4	1	5	45000.	-2000.	
	0.0	4500.	0.0	2350.					

Describer	Meaning
EID1	First Element ID to which these stresses apply. (Integer > 0)
EID2	Last Element ID to which these stresses apply. (Integer; Default = EID1)
INT1	First Integration point for which the stress applies. (Integer > 0; Default = 1)
INTN	Last Integration point for which the stress applies. (Integer > 0; Default = 4)
LAY1	First element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
LAYN	Last element layer for which the stress applies. (Integer > 0; no Default. Enter zero or leave blank if the model does not contain beams or shells.)
STRESS(i)	Up to 7 stress components may be entered. (Real; Default = 0.0)
"COORD"	Indicate that this line defines coordinate system in which STRESS(i) are evaluated
CID	= -1: element coordinate; = 0: basic coordinate (Integer ≥ -1 ≤ 0; default=-1)

Remarks:

1. This entry only applies when using SOL 400 and SOL 600 (where MSC Marc is executed from Nastran) and is ignored for other solutions.
2. Initial stresses must be self-equilibrating and may not produce material nonlinearity.
3. Stress components are as follows:  
 Definitions:  
 s - normal type stress  
 t - shear type stress  
 x,y,z in global system  
 1,2,3 in element local system

3D solid elements (for example type 7)

1 - sxx

2 - syy

3 - szz

4 - txy

5 - tyz

6 - tzx

7 - hydrostatic pressure (Herrmann elements only, otherwise 7 should be blank)

Thick shells (for example type 75)

1 - s11

2 - s22

3 - t12

4 - t23

5 - t31

Thin shells (for example type 72)

1 - s11

2 - s22

3 - t12

Beams (for example type 14 or 98)

1 - s - axial

2 - t - twist

4. STRESS(i) by default are in element coordinate (CID=-1); if CID=0, STRESS(i) will be transformed from basic to element coordinate, for CQUAD4 and CTRIA3 only.

## ITER Iterative Solver Options

Defines options for the iterative solver in SOLs 101, 106, 108, 111, 153 and 400.

Format:

1	2	3	4	5	6	7	8	9	10	
ITER	SID									
		OPTION1VALUE1 OPTION2VALUE2 -etc.-								

Example:

ITER	100									
		ITSEPS=1.0E0-7, MSGFLG=YES, PRECOND=BICWELL, IPAD=3								

Describer	Meaning
SID	Set identification number. (Integer > 0).
PRECOND	Preconditioner option. (Character; Default = "BIC" for real analysis, "BICCMPLX" for complex analysis) See Remarks 3. and 4.
J	Jacobi
JS	Jacobi with diagonal scaling.
C	Incomplete Cholesky.
CS	Incomplete Cholesky with diagonal scaling.
RIC	Reduced incomplete Cholesky.
RICS	Reduced incomplete Cholesky with diagonal scaling.
BIC	Block incomplete Cholesky for real problems.
BICCMPLX	Block incomplete Cholesky for complex problems.
CASI	Element-based third party iterative solver.
USER	User given preconditioning. For direct frequency response (SOL 108): a decomposition will be done for 1st frequency and the factor will be used for all subsequent frequencies as a preconditioner with the iterative solver. Other solutions require a DMAP alter. Please refer to the <i>MSC Nastran Numerical Methods User's Guide</i> description of the SOLVIT module.
CONV	Convergence criterion. (Character; Default = "AREX")
AR	$\ r\ /\ b\ $ where r is the residual vector of current iteration and b is the initial load vector; internal criterion.
GE	Alternative convergence criterion using geometric progression and the differences between two consecutive solution updates; internal criterion.

Describer	Meaning	
	AREX	Same criterion as AR but with the additional consideration of the external convergence criterion. See Remark 2. (Default).
	GEEX	Same criterion as GE but with the additional consideration of the external convergence criterion. See Remark 2.
MSGFLG	Message flag. (Character; Default = "NO")	
	YES	Messages will be printed for each iteration.
	NO	Only minimal messages will be printed from the iterative solver (Default).
ITSEPS	User-given convergence parameter epsilon. (Real > 0.0)	
	Default= 1.E-4	for PRECOND = "CAS1"
	Default= 1.E-8	for PRECOND = "CAS1" with contact in the model
	Default= 1.E-6	for all other PRECOND options
ITSMAX	Maximum number of iterations. (Integer > 0; Default = N/4 where N is the number of rows in the matrix)	
IPAD	Padding value for RIC, RICS, BIC, and BICCMPLX preconditioning. (Integer > 0)	
	Default = 0	for PRECOND = "RIC" or "RICS"
	Default = 2	for PRECOND = "BIC" for purely three-dimensional models and three for two-dimensional and mixed element models. IPAD may be reset automatically by the program to the best value.
	Default = 5	for PRECOND = "BICCMPLX".
IEXT	Extraction level in reduced incomplete Cholesky preconditioning. Block structuring method in block incomplete Cholesky preconditioning. (Integer = 0 thought 7; Default = 0)	
	0	Uses USET/SIL tables (Default).
	1 - 7	The default value of 0 is recommended for all problems. The values 1 - 7 use a heuristic algorithm with a maximum block size equal to IEXT. Although setting IEXT to a value other than 0 could lead to slightly improved performance or reduced disk space use, it should be considered exploratory without the expectation of a benefit.
PREFONLY	Specifies early termination of the iterative solver. (Integer = 0 or -1; Default = 0)	
	0	Runs to completion (Default).
	-1	Terminates after preface giving resource estimates.

Remarks:

1. The optional ITER Bulk Data entry is selected by the SMETHOD Case Control command and is only required to override the defaults specified above.



- The external epsilon is computed as follows:

$$\varepsilon = \frac{(r, x)}{(b, x)}$$

where  $r$  is the final residual vector,  $x$  is the final solution vector and  $b$  is the initial load vector ( $r, x$ ) indicates the inner product of  $r$  and  $x$  and  $(b, x)$  indicates the inner product of  $b$  and  $x$ .

- See the *MSC Nastran Numerical Methods User's Guide* for more information on these options.
- The element-based iterative solver is primarily intended for the solution of very large solid element structural analysis problems. The following restrictions apply:
  - SOLs 101, 200 and 400 only. SOL 200 availability is limited to topology optimization and thus cannot be used for COMPLIANCE, FRMASS, WEIGHT, and VOLUME in DRESP1.
  - SOL 101 convergence parameter epsilon of 1.E-4 may be too large for some models and a decrease to 1.E-8 may be necessary.
  - Solver selection criteria and parameters cannot vary across subcases.
  - No GENEL elements allowed
  - x2GG/x2PP direct input matrix selection is allowed; however, the matrix size is limited to 100 grid points and must be symmetric.
  - No ASET/OMIT reduction allowed
  - Inertia relief is supported in SOL 101 for PARAM,INREL,-1. A SUPORT entry is required. PARAM,INREL,-2 and -4 are not supported.
  - No transfer functions allowed
  - No RFORCE or PLOADX follower forces allowed
  - Follower force stiffness must be symmetricized
  - No heat transfer allowed
  - DMP is not supported with the CASI solver.

Only BAR, BEAM, BUSH, ROD, CONMi, CONROD, DAMPi, ELASi, HEXA, MASSi, PENTA, QUAD4, QUAD8, QUADR, SEAM, SHEAR, TRIA3, TRIA6, TRIAR, TETRA, VISC and WELD elements are allowed.

- GPGPU devices are not supported for iterative methods.
- If Modules are present then this entry may only be specified in the main Bulk Data section.

**LBSH3DG** Tables used by an NLBSH3D to find force as function of displacement, velocity and rotor speed

Defines the force tables of NLBSH3D nonlinear loads as function of relative displacement or velocity and current rotor speed.

Format:

FORM = **ASYM** (Asymmetric matrix representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11	P12	P13	P14	P15	P16			
	P21	P22	P23	P24	P25	P26			
	P31	P32	P33	P34	P35	P36			
	P41	P42	P43	P44	P45	P46			
	P51	P52	P53	P54	P55	P56			
	P61	P62	P63	P64	P65	P66			

FORM = **DIAG** (Only Diagonal terms representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11	P22	P33	P44	P55	P66			

FORM = **SYM** (Symmetric matrix representation)

1	2	3	4	5	6	7	8	9	10
LBSH3DG	LID	FORM	MIN_FREQ	MAX_FREQ					
	P11								
	P21	P22							
	P31	P32	P33						
	P41	P42	P43	P44					
	P51	P52	P53	P54	P55				
	P61	P62	P63	P64	P65	P66			

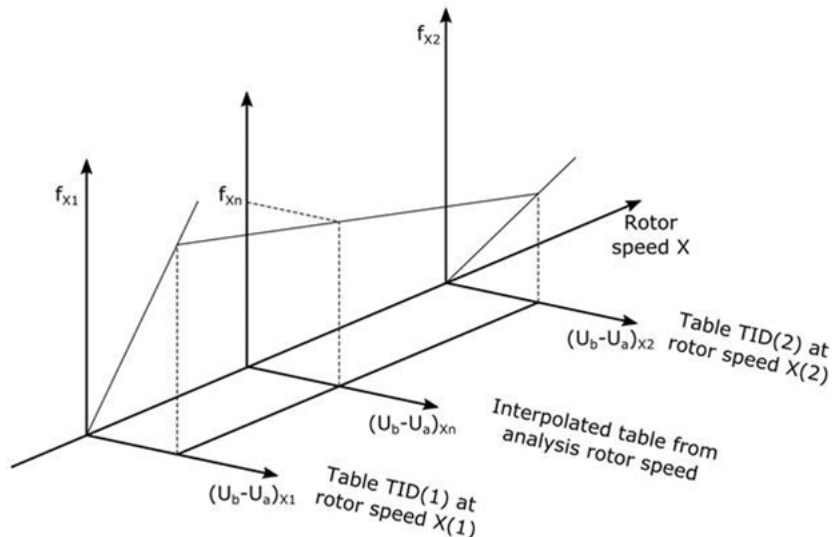
Descriptor	Meaning
LID	Load identification group number for a NLBSH3D. (Integer > 0; Required)
FORM	Type of Input Curves. (Character; Default SYM)
SYM	Symmetric, only lower triangular terms needed
DIAG	Only diagonal terms are needed



Describer	Meaning
	<b>ASYM</b> Asymmetric, all terms are needed
MIN_FREQ	Minimum valid rotor speed for all the table lookups provided. (Real). See remark 4.
MAX_FREQ	Max valid rotor speed for all the table lookups provided. (Real > 0.0 Required). Refer remark 4.
Pij	TABLED5 Id that provides TABLED1 ids as a function of rotor speed in cycles per unit time. The TABLED1 will have force as a function of relative displacement and/or velocity.

### Remarks:

1. Values for the nonlinear forces will be determined using interpolation based on rotor speed and relative deflection/velocity of grid pair. If the rotor speed is beyond the range specified by MIN\_FREQ and MAX\_FREQ, a FATAL message will be produced. (Extrapolation will NOT be used beyond this range).
2. At selected rotor speeds ( $X_i$ ), force vs displacement (for LID-1) or force vs velocity (for LID-2) tables (TID $_i$ ) are defined on TABLED1 entries.
3. For an arbitrary rotor speed  $n$ , linear interpolation is used to determine the force  $f_{x_n}$  to be applied for the displacement/velocity. The illustration shows a graphic representation of interpolation for force vs displacement between tables defined for rotor speeds  $X_1$  and  $X_2$ .



4. For SYNC analysis, rotor speed is obtained from NLFREQ / NLFREQ1 bulk data card. For ASYNC analysis, rotor speed is obtained from RGYRO bulk data card.
5. All the tables (TABLED5 Pij) should provide lookup tables (TABLED1) for the range of rotor speeds between Min\_FREQ and Max\_Freq.

## LEAKAGE

## Mass Loss Through Holes or Permeability of the GBAG or COUPLE Surface

Defines the porosity model to be used with GBAG or COUPLE. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
LEAKAGE	LID	PORID	SUBID	PORTYPE	PORTYPID	COEFF	COEFFV		

Example:

LEAKAGE	7	100	365	PERMEAB	63		0.99		
---------	---	-----	-----	---------	----	--	------	--	--

Describer	Meaning	
LID	Unique number of a LEAKAGE entry. (Integer > 0; Required)	
PORID	Number of a set of LEAKAGE entries PORID must be referenced from a GBAG or COUPLE entry. (Integer > 0; Required)	
SUBID	> 0	Number of a BSURF, BCBOX, BCPROP, BCMATL or BCSEG, which must be a part of the surface, as defined on the GBAG or COUPLE entry.
	= 0	LEAKAGE definitions are used for the entire surface as defined on the GBAG or COUPLE entry.
PORTYPE	Defines the type of porosity. (Character; Required)	
	PORFLOW	The PORFLOW logic is used to model a constant flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLOW entry, with ID as defined on the PORTYPID.
	PORFLWT	The PORFLWT logic is used to model a time dependent flow boundary in the coupling surface. The flow boundary acts on the open area of the coupling surface. The open area is equal to the area of the (sub) surface multiplied by COEFFV. A hole can be modeled when COEFFV is set to 1.0. A closed area results for COEFFV = 0.0. The characteristics of the flow are defined on a PORFLWT entry, with ID as defined on the PORTYPID.

Describer	Meaning
PORHOLE	The PORHOLE logic can be used to model small holes in an air bag. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORHOLE entry, with ID as defined on the PORTYPID.
PERMEAB	The PERMEAB logic is used to model permeable air-bag material. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference between the gas inside the air bag and the environmental pressure. The function is specified on a PERMEAB entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV
PORFGBG	The PORFGBG logic can be used to model gas flow through a hole in the coupling surface connected to a GBAG. A BSURF defines the hole. The open area of the hole is equal to the area of the surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFGBG entry, with ID as defined on the PORTYPID.
PERMGBG	The PERMGBG logic is used to model gas flow through a permeable area in the coupling surface connected to a GBAG. The permeable area can be defined for a BSURF or for the entire coupling surface. The velocity of the gas flow through the (sub) surface is defined as a linear or tabular function of the pressure difference. This function is specified on a PERMGBG entry, with ID as defined on the PORYPID. The area actually used for outflow is the subsurface area multiplied by the value of COEFFV.
PORFCPL	The PORFCPL logic can be used to model gas flow through a hole in the coupling surface connected to another coupling surface. A BSURF defines the hole. The open area of the hole is equal to the area of the (sub) surface multiplied by COEFFV. A value of COEFFV = 1.0 will open up the complete area of the hole, while a value of COEFFV = 0.0 will result in a closed hole. The characteristics for the flow are defined on a PORFCPL entry, with ID as defined on the PORTYPID.
PORHYDS	Prescribes a hydrostatic pressure profile.
PORTYPID	Porosity ID. References existing PORTYPE entry. (Integer > 0; Required)
COEFF	Method of defining the porosity coefficient. (Character; CONSTANT)

Describer	Meaning	
	CONSTANT	The porosity coefficient is constant and specified on COEFFV.
	TABLE	The porosity coefficient varies with time. COEFV is the number of a TABLED1 entry giving the variation with time.
COEFFV		The porosity coefficient or the number of a TABLED1 entry depending on the COEFF entry. (0.0 < Real < 1.0 or Integer < 0, 1.0)

Remarks:

1. The combination of multiple LEAKAGEs with different PORTYPEs is allowed.
2. It allows for setting up the exact same model for either a uniform pressure model (GBAG to LEAKAGE) or an Eulerian model (COUPLE to LEAKAGE). It is then possible to set up the model using the switch from full gas dynamics to uniform pressure (GBAGCOU).

The options PORFGBG and PERMGBG can be used to model air bags with multiple compartments.

## LOAD Static Load Combination (Superposition)

Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, DAREA (if these entries have been converted), PLOAD, PLOAD1, PLOAD2, PLOADB3, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV, ACCEL and ACCEL1 entries (as well as SPCD and SPCR for SOL 600 only). In addition, all the thermal loads are also supported in the NASTRAN thermal analysis (SOL 153 and SOL 400 with analysis=hstat) such as QBDY1, QBDY2, QBDY3, QVECT, QVOL, SLOAD, QHBDY Bulk Data entries.

Format:

1	2	3	4	5	6	7	8	9	10
LOAD	SID	S	S1	L1	S2	L2	S3	L3	
	S4	L4	-etc.-						

Example:

LOAD	101	-0.5	1.0	3	6.2	4			
------	-----	------	-----	---	-----	---	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
S	Overall scale factor. (Real)
Si	Scale factor on Li. (Real)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)

Remarks:

1. The load vector  $\{P\}$  is defined by

$$\{P\} = S \sum_i S_i \{P_{Li}\}$$

2. Load set IDs (Li) must be unique.
3. This entry must be used if acceleration loads (GRAV entry) are to be used with any of the other types.
4. In the static solution sequences, SID must be selected by the LOAD Case Control command. In dynamic solution sequences the EXCITEID of a RLOADi or TLOADi may refer to a LOAD entry.
5. A LOAD may be referenced by another LOAD entry. Nested LOAD references are permitted. A LOAD entry may not reference itself.
6. Note that the LOAD entry scaling for RFORCE uses the square root of the absolute  $S \times S_i$  in rotor dynamics. Thus it is recommended that the LOAD entry not be used with RFORCE in rotor dynamics with a value other than  $S=1.0$  and  $S_i=1.0$  value. See Remark 21. of the RFORCE entry.

7. LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600. In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. In dynamic analysis refer to Remark 2. of the [SPCD](#) entry.
8. If Modules are present then this entry may only be specified in the main Bulk Data section.



## LOADCLID

### Define Solution Load Combinations Using Part Load ID

Define a loading combination in PAA using the Load ID from SUBCASEs of the Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCLID	LID	S							
	S1	LID1							
	PARTNAME1								
	S2	LID2							
	PARTNAME2								
	Etc.								

Example:

LOADCLID	202	1.00							
	1.00	101							
	first_part_name								
	1.0	301							
	second_part_name								

Descriptor	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)
Si	Scale factor applied to Load ID loads. (Real < > 0)
LIDi	Load ID specified during GENERATE by LOAD or CLOAD Case Control directive. (Integer > 0)
PARTNAMEi	Name of Part. (64 -Characters maximum)

Remarks:

- LOADCLID entries are used only in the COMBINE and SOLVE steps of PAA processing.
- The LOADCLID allows a loading combination to be created using the Load id (LIDi) from case control. In each PAA run, each SUBCASE may have a LIDi provided. These are used by the LOADCLID to define a loading combination.
- As with other loading entries, all loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

## LOADCNAM

## Define Solution Load Combination by Load Name

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCNAM	LID	S							
	S1								
	LOADNAME1								
	PARTNAME1								
	S2								
	LOADNAME2								
	PARTNAME2								
	Etc.								

Example:

LOADCNAM	101	1.00							
	2.5								
	gravity_x								
	outboard_right_side_engine_nacelle								
	1.1								
	gravity_x_on_another_part								
	second_part_name								

Describer	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)
Si	Scale factor applied to LOADNAME loads. (Real < > 0)
LOADNAMEi	Name of Load for that Part. (64 -Characters maximum)
PARTNAMEi	Name of Part. (64 -Characters maximum)

Remarks:

- LOADCNAM entries are used only in the COMBINE and SOLVE steps of PAA processing.
- The LOADCNAM allows a loading combination to be created using the LOADNAME from case control. In each PAA run, each SUBCASE may have a LOADNAME provided. These LOADNAMEs are used by the LOADCNAM to define a loading combination.
- All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

## LOADCSUB

### Define Solution Load Combinations Using Subcase Number

Used only in PAA to define a loading combination using the LOADNAMEs used in Case Control for Parts.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCSUB	LID	S							
	S1	SUB1							
	PARTNAME1								
	S2	SUB2							
	PARTNAME2								
	Etc.								

Example:

LOADCSUB	1001	1.0							
	1.0	1							
	part_we_are_using_subcase_1_from								
	1.0	3							
	part_we_are_using_subcase_3_from								

Describer	Meaning
LID	Load set ID. Selected by the LOAD Case Control command. (Integer > 0)
S	Overall scale factor. (Similar to LOAD Bulk Data entry) (Real < > 0)
Si	Scale factor applied to Load ID loads. (Real < > 0)
SUBi	Subcase Number specified during GENERATE or COMBINE run. (Integer > 0)
PARTNAMEi	Name of Part. (Character, C64)

Remarks:

- LOADCSUB entries are used only in the COMBINE and SOLVE steps of PAA processing.
- The LOADCSUB allows a loading combination to be created using the SUBCASE ids from case control. In each PAA run, each SUBCASE has an id. These ids are used by the LOADCSUB to define a loading combination.
- All loads with the same LID are combined. Therefore, care must be exercised when using loading combinations.

## LOADCYH

### Harmonic Load Input for Cyclic Symmetry

Defines the harmonic coefficients of a static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYH	SID	S	HID	HTYPE	S1	L1	S2	L2	

Example:

LOADCYH	10	1.0	7	C	0.5	15			
---------	----	-----	---	---	-----	----	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
HID	Harmonic. See Remark 6. (Integer or blank)
HTYPE	Harmonic type. (Character: “C”, “S”, “CSTAR”, “SSTAR”, “GRAV”, “RFORCE”, or blank).
Si	Scale factor on Li. (Real)
Li	Load set identification number. See Remark 10. (Integer > 0)

Remarks:

1. The LOADCYH entry is selected with the Case Control command LOAD = SID.
2. If HTYPE is blank, the load will be applied to all applicable types in the problem.
3. If HTYPE is “GRAV” or “RFORCE”, GRAV or RFORCE entry loading will be used. Harmonic loads for appropriate available harmonics will be generated automatically in these cases.
4. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
5. If L1 refers to a set ID defined by an SPCD entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX, and SLOAD or converted DAREA entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
6. If HTYPE is “GRAV” or “RFORCE”, the entry in HID will be ignored and therefore may be blank. S2 and L2 must be blank for this case.
7. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYN entries.
8. If HTYPE = “C”, “S”, “CSTAR”, or “SSTAR”, the load on component (HTYPE) of harmonic (HID) is  $L = S(S1 \cdot L1 + S2 \cdot L2)$ .

9. S must be nonzero. In addition, either S1 or S2 must be nonzero.
10. L1 and L2 may reference any of the static or dynamic loading entries including GRAV and RFORCE.

## LOADCYN

### Physical Load Input for Cyclic Symmetry

Defines a physical static or dynamic load for use in cyclic symmetry analysis.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYN	SID	S	SEGID	SEGTYPE	S1	L1	S2	L2	

Example:

LOADCYN	10	1.0	1	R	0.5	17			
---------	----	-----	---	---	-----	----	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
S	Scale Factor. (Real)
SEGID	Segment identification number. (Integer)
SEGTYPE	Segment type. (Character: "R", "L", or blank)
Si	Scale Factors. (Real)
Li	Load set ID numbers. See Remark 8. (Integer > 0)

Remarks:

1. The LOADCYN entry is selected by the LOAD Case Control command.
2. If SEGTYPE is blank, both R and L segments will be used in DIH-type symmetry.
3. L1 and L2 may reference LOAD entries. However, the LOAD entry in such a case must not reference load sets defined via RFORCE and/or GRAV entries.
4. If L1 refers to a set ID defined by SPCD loading entry, the same ID must not reference any nonzero loading specified by the other Bulk Data loading entries given via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, DAREA (if these entries have been converted), MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, and SLOAD entries. In other words, an enforced deformation loading via the SPCD entry is specified by a load set ID L1 or L2 and a zero magnitude loading via a load (such as FORCE) with the same ID.
5. Load set IDs L1 or L2 may not be referenced by load set ID L1 or L2 of LOADCYH entries.
6. The load on the segment (or half-segment) is  $L = S(S1 \cdot L1 + S2 \cdot L2)$ .
7. S must be nonzero. In addition, either S1 or S2 must be nonzero.
8. L1 and L2 may reference any of the static or dynamic loading entries except GRAV and RFORCE.
9. For cyclic buckling loads may only be applied to the first segment and only zero harmonic loads may be applied so the LOADCYH entry should be used.

## LOADCYT

### Table Load Input for Cyclic Symmetry

Specifies loads as a function of azimuth angle by references to tables that define scale factors of loads versus azimuth angles. This entry is used only when STYPE = "AXI" on the CYSYM entry.

Format:

1	2	3	4	5	6	7	8	9	10
LOADCYT	SID	TABLEID1	LOADSET1	METHOD1	TABLEID2	LOADSET2	METHOD2		

Example:

LOADCYT	10	19	27		21	26	1		
---------	----	----	----	--	----	----	---	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
TABLEIDi	Table ID for table load input for load set Li. See Remark 3. (Integer > 0)
LOADSETi	Load set Li. (Integer > 0)
METHODi	Method of interpolation. See Remark 5. (Integer: 0 or 1; Default = 0)
	0 Interpolate the load with the Fourier coefficients specified in the table up to the specified number of harmonics. (Default)
	1 Interpolate the magnitude of the load at corresponding grid points in all segments.

Remarks:

- The LOADCYT entry is selected by the LOAD Case Control command.
- The load set ID given in fields 4 or 7 of this entry may refer to FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, SPCD, TEMP, or TEMPP1 Bulk Data entries.
- Either TABLED1 or TABLED2 type tabular data of azimuth angle (Xi) versus scale factors (Yi) may be used. The azimuth angle values must be in degrees.
- The scale factors given in the tables referenced by TABLEDi entries will be applied only to the magnitudes of the loads defined by LOADSET IDs given in fields 4 or 7.
- For grid point loading entries, (like FORCE, MOMENT, SPCD, and TEMP Bulk Data entries) METHODi = 1 option should be used. For element loading entries (like PLOAD, PLOAD2, PLOAD4, and TEMPP1 Bulk Data entries) either METHODi = 0 or METHODi = 1 option can be used. In particular, if harmonic output of element stresses under temperature loading via TEMPP1 Bulk Data entry, METHODi = 0 option should be used to specify TEMPP1 load set.

**LOADOF** Specifies Table IDs for Individual Degrees of Freedom for the Static Loads with Tables Described using LOADT Entries

Specifies table IDs for individual degrees of freedom for the static loads with tables described using LOADT entries for SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
LOADOF	T0	T1	T2	T3	T4	T5	T6		

Example:

LOADOF	100	101	102	103	104	105	106		
--------	-----	-----	-----	-----	-----	-----	-----	--	--

Describer	Meaning
T0	Table ID specified as a negative Ti value on a LOADT entry (specified as a positive value on this entry. (Integer > 0; no Default)
Ti (i=1 to 6)	ID of a TABLEL1 entry that specifies the time history of each degree of freedom for the loads specified by Li, -Ti on LOADT entries. Up to 6 degrees of freedom may be specified. See Remark 2. (Integer; no Default)

Remarks:

1. LOADOF can only be used in SOL 600
2. This entry must not be used unless there is a negative value of Ti on one or more LOADT entries.



## LOADT Specifies Static Loads that use a Table to Describe Their Variation with Pseudo-Time

Specifies static loads that will use a table to describe their variation with pseudo-time. Available static entries are FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SPCD, SPCR, RFORCE and GRAV for SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
LOADT	SID	L1	T1	L2	T2	L3	T3	L4	
	T4	L5	T5	etc.					

Example:

LOADT	101	1	51	2	52	3	53		
-------	-----	---	----	---	----	---	----	--	--

Descriptor	Meaning
SID	Load set identification number which must have a matching Bulk Data entry LOAD with the same SID. (Integer > 0; no Default)
Li	Load set identification numbers defined on entry types listed above. (Integer > 0)
Ti	ID of a TABLEL1 or TABL3D0 entry that specifies the time history of the entry type. See Remarks 4. through 6. (Integer or Blank; no Default)

Remarks:

- LOADT can only be used in SOL 600.
- In SOL 600 applied displacements must use SPCD (or SPCD) rather than SPC.
- In SOL 600, if any LOADT entries are found in the model, the following parameters are automatically set:
  - PARAM,MARCVERR,11 (Allows table-driven inputs)
  - PARAM,MRCONVER,11 (Uses contact compatible with Version 11)
  - PARAM,MARCTOTT,1 (Allows loads with tables)
- If Ti is positive, the table will be used for all degrees of freedom for the loads specified by Li. If Ti is negative, the absolute value of Ti will be used as the ID of a LOADOF entry providing the table IDs for each degree of freedom for which Li applies.
- If more than one Li references the same force, moment pload4, etc, the first Ti encountered for that entity will be used for all occurrences of that entity for that subcase. Therefore, the 2nd and subsequent occurrences of Ti for these common entities may be left blank. An example can be found in ETL example mloadt03.

6. It is strongly suggested that for a particular dof of a FORCE, MOMENT, etc. that only one table describe the variation for all subcases in the model. For example, if there are 3 subcases the time should normally cover the entire range of 0.0 to 3.0 (rather than having three tables, one for the range 0.0 to 1.0, a second for the range of 1.0 to 2.0 and a third from 2.0 to 3.0).

## LORENZI

## Fracture Mechanics J-Integral in SOL 600

This option gives an estimation of the J-Integral for a crack configuration using the domain integration method. The domain integration method has the advantage that it can also be used for problems with thermal behavior and for dynamic analysis. This procedure is only available for continuum elements. Only the nodes defining the crack front (crack tip in two dimensions) need to be defined. The program automatically finds integrations paths according to the format below. The complete J-Integral is evaluated and output. For the case of linear elastic material with no external loads on the crack faces, the program automatically separates mode I, mode II, and mode III (3-D only) stress intensity factors from the J-Integral.

### Format:

1	2	3	4	5	6	7	8	9	10
LORENZI	IDCR		MSEP	ITYPE	G2D	ISET0			
	DIRECT	ISETD	SV1	SV2	SV3				
	TOP	NREG	JTYPE	TOL	SV1	SV2	SV3		
	GEOM	NREG	JTYPE	TOL	SV1	SV2	SV3	RAD	
		CYLIN							

### Example:

LORENZI	1		0	1	1				
	DIRECT	101	0.0	-2.0E-3	0.0				
	DIRECT	102	0.0	-2.0E-3	0.0				

LORENZI	1		0	1	1				
	TOP	2	1						

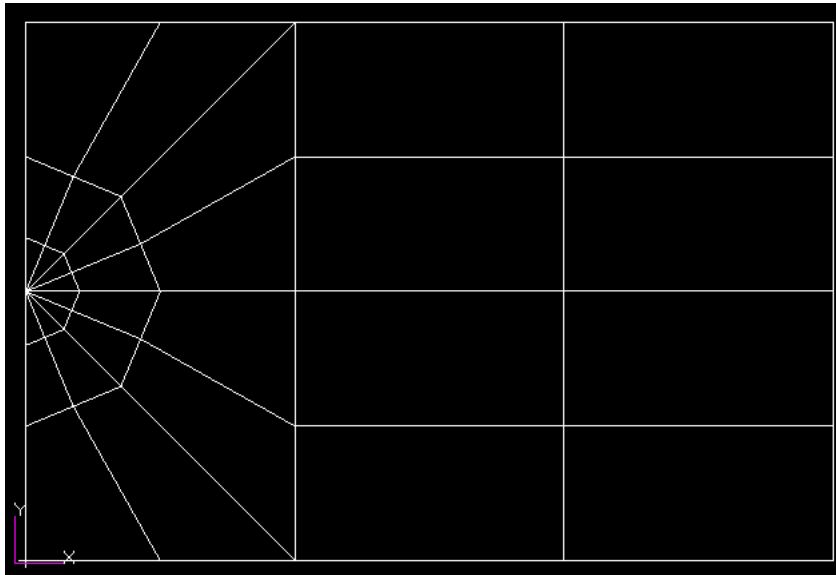
Describer	Meaning
IDCR (4-2)	Identification of this particular crack. (Integer > 0; Required field) IDCR must be unique among all LORENZI cracks.
MSEP (3-1)	Enter 0 for no mode separation or 1 for mode separation. (Integer; Default = 1)
ITYPE (4-1)	Type of crack propagation (Integer; Default = 2) <ol style="list-style-type: none"> <li>1 The “rigid region” will be described using direct inputs of nodes or elements.</li> <li>2 The “rigid region” will be described using an automatic search based on topology.</li> <li>3 The “rigid region” will be described using an automatic search based on geometry.</li> </ol>

Describer	Meaning
G2D (5 for 2D)	If the crack is 2D, enter the grid ID of the node at the crack tip. If the crack is 3D, leave this field blank. (Integer; no Default)
ISET0 (5 for 3D)	If the crack is 3D, enter the ID of a SET3 entry that defines an ordered set of grids or elements along the crack tip (for example in the thickness direction of the crack). If the crack is 2D, enter the grid ID of the node at the crack tip. (Integer; no Default)
DIRECT (Start of 5a)	Enter the string DIRECT if ITYPE=1, otherwise skip this line. See Remark 4. (Character,)
ISETD (6a)	Enter the ID of a SET3 entry defining an ordered set of grids or elements within the rigid region. See Remark 4. (Integer; no Default)
SV1 (7a-1 or 6b-1 or 6c-1)	First component of shift vector. (Real; Default = .0.)
SV2 (7a-2 or 6b-2 or 6c-2)	Second component of shift vector. (Real; Default = 0.0)
SV3 (7a-3 or 6b-3 or 6c-3)	Third component of shift vector, enter only for 3D. (Real; Default = 0.0)
TOP (Start of 5b)	Enter the string TOP if ITYPE=2, otherwise skip this line. (Character)
NREG (5b-1 or 5c-1)	Number of regions for which crack growth is to be estimated. See Remark 5. (Integer; Default = 1)
JTYPE (5b-2 or 6b-2)	Enter 0 if SV1, SV2, SV3 on this line are to be used or enter 1 for automatic determination of SV1, SV2 and SV3. (Integer; Default = 1)
TOL (5b-3 or 5c-3)	Enter tolerance for multiple grids at the crack tip. (Real; Default = 0.0)
GEOM (Start of 5c)	Enter the string GEOM if ITYPE=3, otherwise skip this line (Character)
RAD (7c-1)	Radius of the rigid region to be found. (Real; no Default) Enter only if ITYPE=3.
CYLIN (7c-2)	Relative length of cylinder for the path search. (Real; no Default) Enter only if ITYPE=3.

Remarks:

1. This entry corresponds to Marc's LORENZI model definition option.
2. (i,j) corresponds to Marc Vol C LORENZI entry ith datablock jth field.
3. Repeat the LORENZI as many times as necessary to define all cracks for which J-Integrals should be evaluated.

4. If the DIRECT input is used, repeat the DIRECT line as many times as necessary to define different regions from which crack growth should be estimated. In some cases it is informative to calculate crack growth for several regions and compare the results of these calculations. For example, in the model shown below, the crack tip is at the center on the left side. It might be informative to compare crack growth using a region including all nodes in the first ring (of node) around the crack with that produced by two rings.
5. For the TOP and GEOM methods, only one line per crack should be entered.
6. The continuation line for GEOM is not required except for 3D cracks.
7. For the previous examples, the crack tip is at grid 1 in the following figure. For the direct input, SET3 with ID 101 would reference all grids on and within the ring closest to the crack tip. SET3 with ID 102 would reference all grids on and within the two rings closest to the crack tip. Both examples provide identical results. The topology input is considerably simpler and therefore recommended.
8. The definition of shift vector is the function  $q_1$  in the equation and text described below.



### Background

The J-integral evaluation is based upon the domain integration method. A direct evaluation is not very practical in a finite element analysis due to the difficulties in defining the integration path. In the domain integration method for two dimensions, the line integral is converted into an area integration over the area inside the path. This conversion is exact for the linear elastic case and also for the nonlinear case if the loading is proportional, that is, if no unloading occurs. By choosing this area as a set of elements, the integration is straightforward using the finite element solution. In two dimensions, the converted expression is

$$\bar{J} = \int_A \left( \sigma_{ij} \frac{\partial u_j}{\partial x_1} - W \delta_{1i} \right) \frac{\delta q_1}{\delta x_i} dA$$

for the simplified case of no thermal strains, body forces or pressure on the crack faces.  $A$  is the area inside the integration path and  $q_1$  is a function introduced in the conversion into an area integral. The function can be chosen fairly generally, as long it is equal to one at the crack tip and zero on. The form of the function chosen is that it has the constant value of one at all nodes inside, and decreases to zero over the outermost ring of elements. It can be interpreted as a rigid translation of the nodes inside while the nodes on remain fixed. Thus, the contribution to the above equation comes only from the elements in a ring away from the crack tip. This interpretation is that of virtual crack extension and this method can be seen as a variant of such a technique, although it is extended with the effects of thermal strains, body forces, and pressure on the crack faces. The set of nodes moved rigidly is referred to as the *rigid region* and the function  $q_1$  in the above equation as the **shift vector**. For the evaluation of the J-integral the direction of the shift vector is simply the  $x$  axis in the local crack tip system.

In three dimensions, the line integral becomes an area integral where the area is surrounding a part of the crack front. In this case, the selection of the area is even more cumbersome than in two dimension. The converted integral becomes a volume integral which is evaluated over a set of elements. The rigid region is a set of nodes which contains a part of the crack front, and the contribution to the integral comes from the elements which have at least one but not all its nodes in the rigid region.

## LSEQ Static Load Set Definition

Defines a sequence of static load sets.

Format:

1	2	3	4	5	6	7	8	9	10
LSEQ	SID	EXCITEID	LID	TID					

Example:

LSEQ	100	200	1000	1001					
------	-----	-----	------	------	--	--	--	--	--

Describer	Meaning
SID	Set identification of the set of LSEQ entries. See Remark 5. (Integer > 0)
EXCITEID	The EXCITEID set identification assigned to this static load vector. See Remark 5. (Integer > 0)
LID	Load set identification number of a set of static load entries such as those referenced by the LOAD Case Control command. (Integer > 0 or blank)
TID	Temperature set identification of a set of thermal load entries such as those referenced by the TEMP(LOAD) Case Control command. (Integer > 0 or blank)

Remarks:

1. LSEQ will not be used unless selected in the Case Control Section with the LOADSET command.
2. The number of static load vectors created for each superelement depends upon the type of analysis. In static analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all LSEQ entries in the Bulk Data; in dynamic analysis, the number of vectors created is equal to the number of unique EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data.
3. EXCITEID may be referenced by CLOAD, RLOAD1, RLOAD2, TLOAD1, and TLOAD2 entries in order to apply the static load in nonlinear, static and dynamic analysis.
4. Element data recovery for thermal loads is not currently implemented in dynamics.
5. The SID-EXCITEID number pair must be unique with respect to similar pairs on all other LSEQ entries in the Bulk Data.
6. In a nonsuperelement analysis, LID and TID cannot both be blank. In superelement analysis, they may both be blank as long as static loads are prescribed in the upstream superelements.
7. It is no longer necessary to employ LOADSET/LSEQ data to specify static loading data for use in dynamic analysis. In the absence of LSEQ Bulk Data entries, all static loads whose load set IDs match the EXCITEIDs on all RLOAD1, RLOAD2, TLOAD1, TLOAD2 and ACSRCE entries in the Bulk Data are automatically processed.





## Entries M - O

### MACREEP

### AUTO CREEP Iteration Control in SOL 600

Controls a transient creep analysis. This entry or the MTCREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
MACREEP	ID	Tinc	Ttot	Nmax	NIM	NIK	Tstab	VV1	
	VV2	VV3	IABS						

Example:

MACREEP	1	1.0	1000.	99999	5	1		.01	
	100.	.05	0						

Describer	Meaning
ID	Identification number of a matching Case Control NLPARM command (for statics) or command (for dynamics). (Integer; Required Field)
Tinc	Suggested time increment for creep analysis - The specified value will automatically be adjusted. (Real; Required Value) (2,1)
Ttot	Total time (final time) of the creep analysis. (Real; Required Value) (2,2)
Nmax	Maximum number of time increments allowed in the creep analysis (Integer; Default = 50) (2,3)
NIM	Maximum number of iterations allowed to modify a time step (Integer; Default = 5) (2,4)
NIK	Number of increments between stiffness matrix updates (Integer; Default = 1) (2,5)
Tstab	Stable time step, if known, must be entered for viscoplasticity (Real; Required field for viscoplasticity) (2,7)
VVT	Tolerance value #1. (Real; see below for Defaults) (3,1)
	IABS=0      Enter the tolerance on the creep strain increment divided by the elastic strain (Default = 0.5).
	IABS=1      Enter the maximum creep strain increment allowed. (Default = 0.1)
VV2	Tolerance value #2. (Real, see below for defaults) (3,2)
	IABS=0      Enter the tolerance on the stress change divide by the total stress. (Default = 0.1)
	IABS=1      Enter the maximum stress increment. (Default = 100.0)

Describer	Meaning
VV3	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking (Real; Default = 0.05) (3,3)
IABS	Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (3,5)
	0            Relative checking is used
	1            Absolute checking is used

Remarks:

1. This entry maps to Marc's AUTO CREEP entry.
2. This entry will be used instead of AUTO STEP or AUTO INCREMENT entries in the Marc file for creep analysis. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entry MPCREEP must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

**MARCIN**

Inserts a Text String in Marc from SOL 600

Inserts a text string directly in the Marc input file used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10	
MARCIN	ILOC	ICONT	String							

Example:

MARCIN	-1	0	feature,2301							
--------	----	---	--------------	--	--	--	--	--	--	--

<b>Describer</b>	<b>Meaning</b>	
ILOC	Identification of the location in the Marc file where the string will be placed. (Required; Integer)	
	-1	String is placed at end of Marc's Parameter Section
	0	String is placed at the end of Marc's Model Definition Section
	N (N > 0)	String is placed in load case N after the AUTO STEP, AUTO INCREMENT, etc. entry.
ICONT	(Required; Integer)	
	-1	String is used as the name of an include file which contains all of the direct Marc input data for that portion of Marc (parameter, model definition, etc.). Only one include file per Marc Section is allowed. The string must be entirely in lower case and the include file must be in lower case, except for PC systems where the case does not matter.
	0	Entry is not a continuation of previous MARCIN entry.
	1	Entry is a continuation of previous MARCIN entry and the strings will be placed one after the other on the same Marc line
String	(Required; Character)	
	0	The desired text string. The string is limited to 48 characters per entry. Multiple entries will be placed in the order entered within each Marc location.
	-1	String is an include file name limited to 48 characters.

Remarks:

- Standard Nastran fields 4-9 are ignored for this entry. The string may be entered anywhere within fields 4-9 and will be translated directly to Marc.
- If a long line for the Marc data is required, enter as many MARCIN entries as necessary to describe the entire Marc string using ICONT=1 for each except the first.



3. The total string length including continuation lines is limited to 160 characters.
4. As many MARCIN entries as necessary may be entered to define all desired input.
5. Each entry in the Nastran data file must start with the MARCIN header. Each line in an include file will be translated directly to Marc (there should be no MARCIN, ILOC or ICONT information in the include file(s)).
6. If the direct Marc input is placed in include file(s), separate files are necessary for each portion of Marc (parameter, model definition, etc.) requiring direct input.
7. As part of the Nastran input process, all strings are converted to uppercase. The internal Marc translator will convert them to lower case. For input entered without include files, this will normally make any difference. For include file names, file names must be entirely lower case for computer systems that are case sensitive.
8. MARCIN entries are not always recognized in restart runs and are not recommended.
9. ILOC greater than zero is not available for SOL 600 heat transfer analysis or structural analysis if table-driven loads are requested (for example, by specifying param,marctott,1).
10. For ILOC > 0 if STRING starts with "\$", "comment", or "COMMENT", the string will usually be skipped and no continuation lines should be entered.

## MARCOUT

## Selects Data Recovery Output in SOL 600

Selects output to be saved on the Marc t16 end/or t19 file(s) used in SOL 600 only. This entry is available using the small field format only and should normally be used only when post-processing using the T16 file is to be done (in other words, it should not normally be used if OUTFR on the SOL 600 entry is selected - see PARAM,MARROUTT).

Format:

1	2	3	4	5	6	7	8	9	10
MARCOUT	LAYCODE	IO1	IO2	IO3	IO4	IO5	IO6	IO7	
	IO8	IO9	etc.						

Example:

MARCOUT	125	E11	E21	E41	N1	N2	N35		
---------	-----	-----	-----	-----	----	----	-----	--	--

Describer	Meaning
LAYCODE	Specifies which shell or beam layers are to be output. (Integer; Default is all layers from 1 to the value of PARAM,MARCSLHT) This parameter also applies to composites whether made of shells or solids. For options less than 100, output will be all integration points. For options 100-200, output will only be at the element center. See Remark 13.
0	All shell and beam layer results from 1 to MARCSLHT are output, solid output is available only at the centroid. This option also requires PARAM,MROUTLAY to be set to the maximum numbers of layers desired (MARCSLHT and MROUTLAY should be set to the same values).
1	Top and Bottom layer shell, beam and solid bottom layer results only are output (Marc layer codes 1 and 1000).
2	Top, Bottom and Middle layer results for shell beams and centroid results for solids only are output (Marc layer codes 1, 5000 and 10000)
3	Top and Bottom layer shell, beam and solid bottom layer results are output. In addition, composite layer output listed in the Bulk Data entry, MLAYOUT will be included.
100	Same as option 0 except output is only at the center of each layer
101	Same as option 1 except output is only at the center of each layer
102	Same as option 2 except output is only at the center of each layer
103	Same as option 3 except output is only at the center of each layer.
IO(i)	Indicates the type of Marc output requested according to the following table:

Element Output	Description
-9999	If -9999 is entered in field 3, the defaults for the post file of the Marc version being run will be used. Fields 4-9 must be blank and no continuation lines should be entered. No OUTR options on the SOL 600 entry are available with this option.
“E-USER”*	First user-defined element post code(s) are generated by user subroutine plotv.f
“E-USER1”*	2 <sup>nd</sup> user-defined element post code(s)
“E-USER2”*	3 <sup>rd</sup> user-defined element post code(s)
“E-USER3”*	4 <sup>th</sup> user-defined element post code(s)
“E-USER99”*	100 <sup>th</sup> user-defined element post code(s)
*These outputs are only available in the t16 file, not in op2, xdb, f06, punch. A maximum number of 100 user-defined element post codes may be entered for SOL 600).	
E1 to E7	strain components, VM strain
E8	Equivalent plastic strain
E9	Total Temperature (combined heat/structural analysis)
E10	Incremental Temperature (combined heat/structural analysis)
E11 to E17	stress components, VM stress, Mean stress
E18	Mean normal stress
E20	Element Thickness (thickness can change vs time)
E21 - E27	plastic strain components, VM plastic strain
E28	Plastic strain rate
E29	Value of second state variable if applicable
E31 to E37	Creep strain components and equivalent creep strain
E38	Total swelling strain
E39	Value of third state variable if applicable
E41 to E47	Cauchy stress components, VM Cauchy stress
E49	Thickness strain for plane stress, Mooney or Ogden material
E58	Elastic strain energy density
E59	Equivalent stress/yield stress
E60	Equivalent stress/yield stress at current temperatures
E48	Strain Energy Density
E68	Plastic strain energy density
E69	Current Volume
E71 to E76	Components of thermal strain (separated from total strain)
E78	Original Volume

Element Output	Description
E79	Grain size if applicable
E80	Damage indicator for Cockroft-Latham, Oyane and principal stress criteria
E81 to E86	Components of cracking strain if applicable
E91 to E107	Failure indices associated with failure criteria from MATF
E108 to E109	Interlaminar shear for thick composite shells
E110	Interlaminar shear bond index for thick composite shells
E111 to E116	Components of stress in “preferred coordinate system”
E121 to E126	Elastic strain components
E127	Equivalent elastic strain
E128	Major engineering strain
E129	Minor engineering strain
E171	Porosity
E172	Void ratio
E173	Pore pressure
E174	Preconsolidation pressure
E175	Equivalent viscoplastic strain rate (powder material)
E176	Relative density (powder material)
E177	Void volume fraction (damage model)
E178	Lemaitre damage factor
E179	Lemaitre relative damage
E180	Total temperature (same as E9)
E181 to E183	Components of temperature gradient
E184 to E186	Components of flux
E241	Gasket pressure if applicable
E242	Gasket closure if applicable
E243	Plastic Gasket closure if applicable
E244	Exponential powder parameter kappa
E245	Exponential powder parameter x
E251 to E253	Components of interlaminar normal stress in basic coordinate system
E254 to E256	Components of interlaminar shear stress in basic coordinate system
E257	Interlaminar shear bond index for composite solids
E216	Beam axis or Orientation of CBUSH/CFAST elements
E261	Beam Axis

Element Output	Description
E264-E269	Beam/Bar or Bush Element Forces
	The next 6 lines are for CBUSH Force output (CBUSH stress/strain output and can be obtained with standard stress/strain codes if the stress/strain recovery. Coefficients are defined on the PBUSH entries.)
E264	CBUSH Axial Force (local bush coord sys)
E265	CBUSH Moment Mxx (local bush coord sys)
E266	CBUSH Moment Myy (local bush coord sys)
E267	CBUSH Shear Force Vxy (local bush coord sys)
E268	CBUSH Shear Force Vyx (local bush coord sys)
E269	CBUSH Torque (local bush coord sys)
E270	Bimoment (for beam element)
E301	Total strains tensor
E311	Stress tensor
E321	Plastic strain tensor
E331	Creep strain tensor
E341	Cauchy stress tensor
E371	Thermal strain tensor
E381	Cracking strain tensor
E391	Stress in composite ply (“preferred”) direction
E401	Elastic strain tensor
E411	Stress tensor in basic coordinate system
E421	Elastic strain tensor in basic coordinate system
E431	Plastic strain tensor in basic coordinate system
E441	Creep strain tensor in basic coordinate system
E451	Velocity strains (for fluids)
E461	Elastic strain in preferred direction tensor
E471	Rebar stress components in basic coordinate system in undeformed config if applicable
E481	Rebar stress components in basic coordinate system in deformed config if applicable
E487	Rebar angle
E501	Interlaminar normal stress
E511	Interlaminar shear stress
E531	Volume fraction of Martensite



Element Output	Description
E541	Phase transformation strain tensor
E547	Equivalent phase transformation strain
E548	Equivalent TWIN strain
E549	Equivalent TRIP strain
E551	Equivalent plastic strain in a multiphase aggregate
E552	Equivalent plastic strain in Austenite
E553	Equivalent plastic strain in Martensite
E557	Yield stress of multiphase aggregate
E610 to E617	Strength ratios based on MATF failure modes
E641	Generalized strain curvatures tensor
E661	Generalized stress moments tensor
E681	True strain tensor for continuum elements
E691	Element orientation vector 1
E694	Element orientation vector 2
E697	Layer orientation angle
E704	Real harmonic axial force
E705	Real harmonic Moment Mxx
E706	Real harmonic Moment Myy
E707	Real harmonic Shear force Vxy
E708	Real harmonic Shear force Vyx
E709	Real harmonic Torque
E710	Real harmonic Bimoment
E714	Imaginary harmonic axial force
E715	Imaginary harmonic Moment Mxx
E716	Imaginary harmonic Moment Myy
E717	Imaginary harmonic Shear force Vxy
E718	Imaginary harmonic Shear force Vyx
E719	Imaginary harmonic Torque
E720	Imaginary harmonic Bimoment
E721	Cauchy Stress Tensor in Preferred (Material) Coordinate System
E731	Curvature Tensor (for shell elements)
E741	Moment Tensor (for shell elements)

Nodal Output	Description
“N-USER”*	First user-defined nodal post cod are generated by user subroutine upstnd.f
“N-USER1”*	2 <sup>nd</sup> user-defined nodal post cod are generated by user subroutine upstnd.f
“N-USER2”*	3 <sup>rd</sup> user-defined nodal post cod are generated by user subroutine upstnd.f
“N-USER3”*	4 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f
“N-USER4”*	5 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f
“N-USER99”*	100 <sup>th</sup> user-defined nodal post cod are generated by user subroutine upstnd.f, etc.
*User-defined outputs are only available in the t16 file, not in op2, xdb, f06, punch. A maximum of 100 user-defined nodal post codes may be entered for SOL 600.	
N1,N2	displacements, rotations
N3,N4	Applied forces & moments
N5,N6	reaction forces & moments
N7	Fluid velocity if applicable
N8	Fluid pressure if applicable
N9	External fluid force if applicable
N10	Reaction fluid force if applicable
N11	Sound pressure if applicable
N12	External sound pressure if applicable
N13	Reaction sound pressure if applicable
N14	Temperature
N15	External heat flux
N16	Reaction heat flux
N23	Pore pressure if applicable (soil analysis)
N24	External mass flux if applicable
N25	Reaction mass flux if applicable
N26	Bearing pressure if applicable
N27	Bearing force if applicable
N28,N29	Velocity
N30,N31	Acceleration
N32,N33	Modal mass and rotational modal mass
N34,N35	Contact normal stress/force
N36,N37	Contact friction stress/force
N38,N39	Contact status, Contact touched body

Nodal Output	Description
N40	Herrmann variable
N46,N48	Tying force and moments
N47	Coulomb Force
N49	Generalized nodal stress
N50	Generalized nodal strain
N51,N52	Inertia relief force and moment
N53	J-Integral
N54	Stress Intensity, Mode I
N55	Stress Intensity, Mode II
N56	Stress Intensity, Mode III
N57	Energy release
N58	Energy release rate I
N59	Energy release rate II
N60	Energy release rate III
N62	Crack system local X
N63	Crack system local Y
N64	Crack system local Z
N65	Contact Separation Distance
N66	Normal Breaking Index
N67	Tangential Breaking Index
N68	Total Breaking Index
N69	Normal Delamination Index
N70	Tangential Delamination Index
N71	Total Delamination Index
N73	Glue deactivation status
N74	VCCT Failure Index
N76	Lorentz Force
N77	Wear Index
N78	Wear Rate

**Remarks:**

1. MARCOUT is only available when Marc is executed from within Nastran and controls what results are available in the Marc t16 file. All elements or nodes of each type selected will be placed on the t16 file (in other words, it is not possible to control this output by selecting various sets). Some, but not all, of the results in the t16 file may be used to obtain .op2, .xdb, punch or .f06 results output by specifying OUTF options on the SOL 600 Executive Control statement. They type of results which may be placed in the .op2, .xdb, punch or .f06 files are the same as those available in SOL 106 or 129. Other types of results are only available for postprocessing using the t16 file. Whenever possible, i.e., if your gui allows it, the t16 file should be used instead of .op2, .xdb, punch, or .f06 files for postprocessing.
2. Values such as E1, E21 correspond to Marc's postcodes 1 and 21, respectively.
3. Outputs produced by MARCOUT are the same for all subcases, load steps, iterations, etc.
4. The MARCOUT entry may be repeated as many times as desired, or all entries may be placed on continuation lines.
5. For entries E1, E11 and E21 corresponding entries E2-E7, E12-E17 and E22-E27 will be generated automatically. These terms correspond to 3 normal stress (or strain) and 3 shear stress/strain values plus the equivalent von Mises value. See Marc volume C POST description for more details.
6. If this entry is not used, the following defaults are entered automatically: E301, E341, E47, N1, N2, N35, N37, N39. When creep or heat transfer is present, additional items are added appropriately. If the MARCOUT entry is entered, only those items specified will be output.
7. Only displacements, rotations, Cauchy stresses and one type of strains (total, plastic or elastic) may be transferred to the Nastran database.
8. Displacements, at least one stress tensor and one strain tensor must be selected if any OUTF options are to be used.
9. For SOL 600, Nastran Case Control commands such as SET ID=, DISP=, STRESS=, and STRAIN= only control the output in the .OP2, .XDB, punch, .F06 and jid.marc.out files. The Case Control requests do not affect the t16 output.
10. Default MARCOUT options are sufficient for most needs and it is recommended that the MARCOUT entry only be employed by advanced users.
11. If some (but not all) forces (E264-E269) are specified, the missing ones will be added automatically since the t16op2 conversion requires all be present (this capability is available starting in MD Nastran R2 and the MSC Nastran 2006 release, prior to that, all needed to be specified if any OUTF options were requested.)
12. LAYCODE values 100-103 will automatically set PARAM,MARCCENT,1
13. For LAYCODE=1, if op2.xdb,f06 or punch output is requested element stress/strain results which normally are output at the center of the element will be output on the bottom surface of the element. For composite solids, if output at all layers is desired, postprocessing using the t16 file is required.
14. Additional outputs described in Marc Vol C (POST Section) may be available for certain types of SOL 600 analyses.

**MARPRN**

## Defines "Print" Options for SOL 600

This option corresponds to Marc's PRINT parameter which controls a variety of output and other information. for the most used options, PARAM,MARCPRN should be used.

## Format:

1	2	3	4	5	6	7	8	9	10
MARPRN	IP1	IP2	IP3	IP4	IP5	IP6	IP7	IP8	
	IP9	-etc.-							

## Example:

MARPRN	2	5	39						
--------	---	---	----	--	--	--	--	--	--

Describer	Meaning
IPi	"Print" code from list shown below (Integer; no Default). Include as many options as desired
1	Output element stiffness matrices (this also prints out the shell surface metric for doubly curved shells 4, 8, and 24), consistent mass matrix, and equivalent nodal loads. <b>CAUTION:</b> This produces significant output.
2	Output of the matrices used in tying. (TYING, SERVO LINK, UFORMS.)
3	Forces the solution of a nonpositive definite matrix. This is only recommended for the AUTO INCREMENT option to pass collapse points in the collapse analysis. This can be entered on the CONTROL option.
5	To obtain additional information concerning gap convergence. In contact analysis, set to 5 to obtain information concerning nodes touching or separating from surfaces and also to print out the maximum residual and reaction forces.
6	To obtain output of nodal value array during rezoning.
8	To obtain incremental displacements in local system in contact problems.
9	To obtain latent heat output.
10	To obtain the stress-strain relation in the local coordinate system.
11	To obtain additional information on the interlaminar stress calculation.
12	To output the right-hand side and solution vector. <b>CAUTION:</b> This produces significant output.
13	To obtain additional information regarding CPU resources used.
15	To obtain additional information regarding surface energy balances.
20	To obtain information regarding the evaluation of tables.

Describer	Meaning
21	To obtain information about application of kinematic boundary conditions when table input is used.
22	To obtain information about distributed loads, point loads, films, foundations, and initial conditions when table input is used.
26	To print additional information regarding sink points.
27	To obtain reaction forces at tied nodes.
28	To obtain additional information about convective terms in heat transfer and fluid analysis.
34	To print a description of what independent variables may be used with a physical quantity.
36	To obtain CASI solver debug information (has the least details).
37	To obtain CASI solver debug information (has more details).
38	To obtain CASI solver debug information (has the most details).
39	To obtain detailed information about memory allocation.
43	To obtain information about VCCT
44	To obtain information during progressive failure calculations

**Remark:**

1. See PARAM,MARCP RN for a simpler way to enter the most used print options.

## MASSET

## Mass Combination Definition

The MASSET bulk data entry defines a linear combination of mass cases to form the subcase-dependent mass which is selected by associated MASSET case control. The subcase mass is defined as follows by the MASSET bulk:

$$M_{ID} = S_0 \sum_i S_i M_i$$

Where,  $i$  refers to the MASSID qualifier:  $i = 0$  is the base mass case,  $i > 0$  are the incremental mass cases. ID's of mass combinations defined by MASSET bulk data entry and mass increments defined by MASSID bulk data sections should be unique.

Format:

1	2	3	4	5	6	7	8	9	10
MASSET	ID	S0	S1	ID1	S2	ID2	S3	ID3	
	S4	ID4							

Example:

MASSET	11	1.0	1.0	0	1.0	101			
--------	----	-----	-----	---	-----	-----	--	--	--

Describer	Meaning
ID	Set identification number referenced by MASSET case control. (Integer > 0; Required)
S0	The overall scale factor for the linear combination (Real >= 0.0; Default = 0.0)
Si	The scale factor for the ith mass increment (Real >= 0.0; Default = 0.0)
IDI	The MASSID for the ith mass increment (Integer > 0; No Default)

## MATBV

## Material Bulk Viscosity - SOL700

Defines the bulk viscosity for materials. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATBV	MID	BULK TYP	BULKQ	BULKL	CSCALE	DMPFAC	IIMMREL		

Example:

MATBV	7		1.2						
-------	---	--	-----	--	--	--	--	--	--

Descriptor	Meaning
MID	Material identification. See Remark 1. (Integer > 0, Required.)
BULK TYP	Bulk viscosity type. (Character: default=DYNA) DYNA                   Standard DYNA3D model. DYTRAN                Enhanced DYNA model.
BULKL	Linear bulk-viscosity coefficient. (Real ≥ 0.0; default=1.0)
BULKQ	Quadratic bulk-viscosity coefficient. (Real ≥ 0.0; default=0.0)
CSCALE	When this material model is used with MEMB shell elements (See PSHELL1), the compressive stresses in the principal directions will be scaled with this factor. CSCALE=0.0 results in a tension only material. See Remark 2. (Real ≥ 0.0; default=1.0)
DMPFAC	When this material model is used with MEMB shell elements, damping is applied to the stresses. DMPFAC = 0.05 results in 5% damping applied to membrane stresses. See Remark 2. (Real ≥ 0.0; default=0.0)
IIMMREL	Relaxation factor used with the Initial Metric Method. This option is only used when this material model is used with MEMB shell elements and the IMM method is activated. See Remark 3. (0.0 < Real < 1.0; default=1.0e-3)

Remarks:

- The material number must refer to a basic material definition such as MAT1/MAT2/MAT8/MATORT/MATHE.
- For air bag modeling the following values of CSCALE and DMPFAC are suggested:  
CSCALE = 0.1  
DMPFAC= 0.05 to 0.20
- The Initial Metric Method relaxation factor is used to slow down the expansion of the membrane elements during the calculation. The default is sufficient in most simulations.



## MAT1 Isotropic Material Property Definition

Defines the material properties for linear isotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1	MID	E	G	NU	RHO	A	TREF	GE	
	ST	SC	SS	MCSID					

Example:

MAT1	17	3.+7		0.33	4.28	6.5-6	5.37+2	0.23	
	20.+4	15.+4	12.+4	1003					

Describer	Meaning
MID	Material identification number. (Integer > 0)
E	Young's modulus. (Real $\geq$ 0.0 or blank)
G	Shear modulus. (Real $\geq$ 0.0 or blank)
NU	Poisson's ratio. (-1.0 < Real $\leq$ 0.5 or blank)
RHO	Mass density. See Remark 5. (Real)
A	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 9. and 10. (Real; Default = 0.0 if A is specified.)
GE	Structural element damping coefficient. See Remarks 8., 9., and 4. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied, used only to compute margins of safety in certain elements; and have no effect on the computational procedures. (Real $\geq$ 0.0 or blank). See Remark 14.
MCSID	Material coordinate system identification number. Used only for PARAM,CURV processing. See Parameters. (Integer $\geq$ 0 or blank)

Remarks:

- The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
- The following rules apply when E, G, or NU are blank:
  - E and G may not both be blank.
  - If NU and E, or NU and G, are both blank, then both are set to 0.0.

- If only one E, G, or NU is blank, then it will be computed from the equation:  
 $E = 2 \cdot (1 + \text{NU}) \cdot G$ . If this is not desired, then the MAT2 entry is recommended. If E, G, or NU are made temperature dependent by the MATT1 entry, then the equation is applied to the initial values only. Caution: Inconsistent E and G can result in  $\text{NU} > .5$  which for three dimensional elasticity results in negative stress for positive strain. This can cause large max ratios.
3. If values are specified for all of the properties E, G, and NU, then it is recommended that the following relationship be satisfied:

$$\left| 1 - \frac{E}{2 \cdot (1 + \text{NU}) \cdot G} \right| < 0.01$$

If this relationship is not desired, then the MAT2 entry is recommended.

It should also be noted that some of the properties are not applied in the stiffness formulation of certain elements as indicated in Table 18. Therefore, it is recommended that only the applicable properties be specified for a given element.

Table 18 Material Property Usage Versus Element Types

Element Entry	E	NU	G
CROD CBEAM CBAR	Extension and Bending	Not Used	Torsion Transverse Shear
CQUADi CTRIAi CCONEAX	Membrane, including In-plane Shear, and Bending		Transverse Shear
CSHEAR		Not Used	Shear
CRAC2D		All Terms	Not Used
CHEXA CPENTA CTETRA CRAC3D		All Terms	Not Used
CTRIAX6	Radial, Axial, Circumferential	All Coupled Ratios	Shear

4. MAT1 materials may be made temperature-dependent by use of the MATT1 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
5. The mass density RHO will be used to compute mass for all structural elements automatically.
6. Weight density may be used in field 6 if the value 1/g is entered on the PARAM,WTMASS entry, where g is the acceleration of gravity (see Parameters).
7. MCSID must be nonzero if PARAM,CURV is specified to calculate stresses or strains at grid points on plate and shell elements only.

8. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$ , by 2.0.
9. TREF and GE are ignored if the MAT1 entry is referenced by a PCOMP/PCOMPG entry.
10. TREF is used in two different ways (See Remarks 6. and 7. of TEMPERATURE Case Control entry for details.):
  - Except for SOL106 and SOL400 nonlinear analysis, TREF is used as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, in which case, TREF is ignored.
  - In SOL106 and SOL400 nonlinear analysis, TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection.

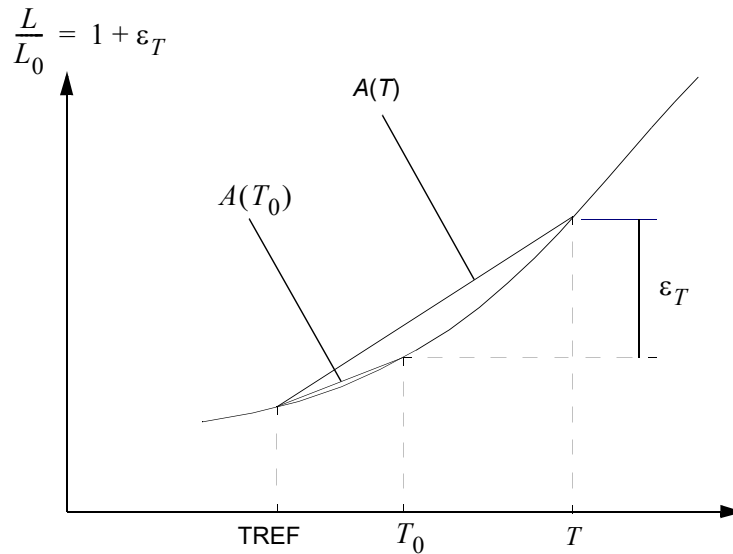


Figure 9-92 Use of TREF in Calculation of Thermal Loads

$$\varepsilon_T = A(T) \cdot (T - TREF) - A(T_0) \cdot (T_0 - TREF)$$

where  $T$  is requested by the TEMPERATURE(LOAD) command and  $T_0$  is requested by the TEMPERATURE(INITIAL) command.

## Notes:

- A is a secant quantity.
- TREF is obtained from the same source as the other material properties; e.g., ASTM, etc.
- If  $A(T)$  constant, then  $\epsilon_T = A \cdot (T - T_0)$
- If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters](#).

11. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILON,INTEGRAL' in bulk data. See [Parameters](#).
12. For SOL 600, E must not be blank or zero.
13. Negative values for ST, SC, and SS lead to no margins of safety being computed.
14. ST, SC, and SS are not used in SOL 400 with advanced elements or SOL 600. Use MATF instead.

## MAT1A Additional Material Properties for MAT1- SOL700

Specifies additional the material properties of MAT1 for solid elements. Use SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1A	MID	EID							

Example:

MAT1A	101	201							
-------	-----	-----	--	--	--	--	--	--	--

Describer	Meaning
MID	Identification number of MAT1. (Integer > 0; required.)
EID	EOSxxx ID. (Integer > 0; required.)

Remarks:

- MAT1A always required for solid elements when MATEP is used. However, when RYIELD option of MATEP is set to VMISES, ORTHOCR or SOIL, MATEOS is not required.

## MAT1F

## Isotropic Material Frequency Dependence

Specifies frequency-dependent material properties on MAT1 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT1F	MID	E	G	NU	RHO			GE	

Example:

MAT1F	33	15	22	16					
-------	----	----	----	----	--	--	--	--	--

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
E	Identification number of a TABLEDi entry for the Young's modulus. (Integer > 0 or blank)
G	Identification number of a TABLEDi entry for the shear modulus. (Integer > 0 or blank)
NU	Identification number of a TABLEDi entry for the Poisson's ratio. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the structural damping coefficient. (Integer > 0 or blank)
RHO	Identification number of a TABLEDi entry for the mass density. (Integer > 0 or blank). This table is only used for isotropic poroelastic material with BEGIN BULK TRMC and is applicable to that trim component only.

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT1 entry.
- The user is reminded that on the MAT1 entry, if any one of the entries E, G, or, NU is left blank, they are connected by the relationship  $G=E/(2(1+NU))$ . Therefore, this entry requires that if any E, G, or NU is to be frequency dependent then Table references must be present for all three of the E, G, NU even if this requires a table of constant values.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT1 then GE table must be blank or 0.

## MAT2 Shell Element Anisotropic Material Property Definition

Defines the material properties for linear anisotropic materials for two-dimensional elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	MCSID	GE11	GE12	GE13	GE22	GE23	GE33		

Example:

MAT2	13	6.2+3			6.2+3		5.1+3	0.056	
	6.5-6	6.5-6		-500.0	0.002	20.+5			
	1003								

Describer	Meaning
MID	Material identification number. See Remark 1. (Integer > 0)
Gij	The material property matrix. (Real) See Remarks 4., 5., 6., and 9.
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient vector. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 10., 11. and 14. (Real or blank)
GE	Structural element damping coefficient. See Remarks 8., 10., and 13. (Real)
ST, SC, SS	Stress limits for tension, compression, and shear are optionally supplied (these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. (Real or blank). See Remark 17.
MCSID	Material coordinate system identification number. Used only for PARAM, CURV processing. See Parameters. (Integer ≥ 0 or blank)
GEij	Structural damping matrix. See Remark 17. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. MAT2 materials may be made temperature dependent by use of the MATT2 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
3. The mass density, RHO, will be used to automatically compute mass for all structural elements.

4. The convention for the  $G_{ij}$  in fields 3 through 8 are represented by the matrix relationship

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{bmatrix} \begin{Bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{Bmatrix} - (T - T_0) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \end{Bmatrix}$$

The  $G$  matrix, including  $G_{13}$  and  $G_{23}$  is an extension to conventional orthotropic plane stress behavior that allows membrane-shear coupling. If the MAT2 is used for plane stress elements or referenced as MID1 or MID2 in a PSHELL, the  $G$  matrix must be invertible and positive definite.

5. When MAT2 is referenced as a MID3 in a PSHELL, then it is used as:

$$\begin{Bmatrix} \tau_{13} \\ \tau_{23} \end{Bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{12} & G_{22} \end{bmatrix} \begin{Bmatrix} \gamma_{13} \\ \gamma_{23} \end{Bmatrix}$$

This may lead to user warning message 6134 which may be ignored. See *The NASTRAN Theoretical Manual*, Section 4.2.

6. When MAT2 is referenced as a MID4 in a PSHELL, then only  $G_{13}$  and  $G_{23}$  are required. One will observe a user warning message 9994, The Material Property Matrix is not positive definite which may be ignored.
7. MCSID must be nonzero if PARAM,CURV is specified to extrapolate element centroid stresses or strains to grid points on plate and shell elements only. CQUAD4 element corner stresses are not supported by PARAM,CURV.
8. To obtain the damping coefficient  $GE$ , multiply the critical damping ratio  $C/C_0$  by 2.0.
9. Unlike the MAT1 entry, data from the MAT2 entry is used directly without adjustment of equivalent  $E$ ,  $G$ , or  $NU$  values.
10. TREF and  $GE$  are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
11. TREF is used in two different ways (See Remarks 6. and 7. of TEMPERATURE Case Control entry for details.):
- Except for SOL106 and SOL400 nonlinear analysis, TREF is used as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, in which case, TREF is ignored.
  - In SOL106 and SOL400 nonlinear analysis, TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See Figure 9-92 in Remark 10 of the MAT1 description.
12. If PARAM,W4 is not specified,  $GE$  is ignored in transient analysis. See [Parameters](#).



13. PCOMP/PCOMPG entries generate MAT2 entries starting from 100,000,001 onwards. Explicitly specified MAT2 IDs must not conflict with internally generated MAT2 IDs. Furthermore, the A1, A2 and A3 fields of MAT2 generated by Nastran from PCOMP that's used for PSHELL's MID4 has a special format. They are  $[G4] \times [\alpha 4]$  and not  $[\alpha 4]$ . These generated MAT2 entries have MID greater than 100,000,000. User supplied MAT2 should avoid using MID greater than 100,000,000 if it's used for MID4 of a PSHELL entry. If MIDs larger than 99999999 are used, PARAM,NOCOMPS,-1 must be specified to obtain stress output.
14. In nonlinear static analysis (SOL 106) the QUAD4, TRIA3, QUADR, and TRIAR thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILON,INTEGRAL' in bulk data. See [Parameters](#).
15. Negative values for ST, SC, and SS lead to no margins of safety being computed.
16. If the MAT2 is referenced by the PCOMP/PCOMPG entry, the transverse shear flexibility for the referenced lamina is zero.
17. Defines structural damping matrix data for advanced composites. If the GEij values are present, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GEij values. The rationale of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDi entries must refer to MAT2 entries.

For any ply that does not point to a MAT2 with extended GEij, the GE value will be taken from the GE field of the corresponding MATi entry (MAT1 field 9, MAT2 field 6 of first continuation, MAT8 field 2 of second continuation.).

The damping when extended GEij is in effect is applied in element material routines by forming the product  $GE_{ij} * G_{ij}$  when forming the  $K_4$  damping matrix. For any ply not using an extended GEij of MAT2 the product  $GE * G_{ij}$  is used.

## MAT2F

## Shell Element Anisotropic Frequency Dependence

Specifies frequency-dependent material properties on MAT2 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT2F	MID	G11	G12	G13	G22	G23	G33		
					GE				
		GE11	GE12	GE13	GE22	GE23	GE33		

**Example:**

MAT2F	34								
		47	48	51	47	48	51		

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT2 entry. (Integer > 0)
Gij	Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)
GEij	Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT2 entry.
- Table references must be present for each item that is frequency dependent.
- If the GEij values are present on the MAT2 entry for any ply, then Nastran will ignore the GE value given on the first continuation entry field (6) and the GE entry given in field (8) of the PCOMP/PCOMPG entry and use the given GEij values. The rationale of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix.

which is the main source of damping. To use this feature in composites, the PCOMP/PCOMPG MIDi entries must refer to MAT2 entries. See Remarks 17. and of MAT2 entry. The damping is applied in element material routines by forming the product  $GE_{ij} * G_{ij}$  when forming the  $K_4$  damping matrix.

4. IF GE=0.0 on corresponding MAT2 then GE table must be blank or 0. For any  $GE_{ij}=0.0$  on corresponding MAT2 the corresponding  $GE_{ij}$  table must be blank or 0.

## MAT3

## CTRIAX6 or PSHLN2 or PLCOMP Material Property Definition

Defines the material properties for linear orthotropic materials used by the CTRIAX6 element entry. It also is allowed with orthotropic materials on the PSHLN2 and PLCOMP entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT3	MID	EX	ETH	EZ	NUXTH	NUTHZ	NUZX	RHO	
			GZX	AX	ATH	AZ	TREF	GE	

Example:

MAT3	23	1.0+7	1.1+7	1.2+7	.3	.25	.27	1.0-5	
			2.5+6	1.0-4	1.0-4	1.1-4	68.5	.23	

Describer	Meaning
MID	Material identification number. (Integer > 0)
EX, ETH, EZ	Young's moduli in the $x$ , $\theta$ , and $z$ directions, respectively. (Real > 0.0)
NUXTH, NUTHZ, NUZX	Poisson's ratios (coupled strain ratios in the $x\theta$ , $\theta z$ , and $zx$ directions, respectively). (Real)
RHO	Mass density. (Real)
GZX	Shear modulus. (Real > 0.0)
AX, ATH, AZ	Thermal expansion coefficients. (Real)
TREF	Reference temperature for the calculation of thermal loads or a temperature-dependent thermal expansion coefficient. See Remark 10. (Real or blank)
GE	Structural element damping coefficient. See Remarks 9. and 11. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. MAT3 materials may be made temperature dependent by use of the MATT3 entry. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE (INIT) Case Control command.
3. The few numbers EX, ETH, EZ, and GZX must be present.
4. A warning message will be issued if any value of NUXTH or NUTHZ has an absolute value greater than 1.0.

5. MAT3 materials may only be referenced by the CTRIAX6 entry or the PSHLN2 or PLCOMP entries.
6. The mass density RHO will be used to automatically compute mass for the CTRIAX6 element.
7. The x-axis lies along the material axis (see [Figure 9-80](#) in the CTRIAX6 entry). The  $\theta$ -axis lies in the azimuthal direction. The z-axis is normal to both.
8. The strain-stress relationship is

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_\theta \\ \varepsilon_z \\ \gamma_{zx} \end{Bmatrix} = \begin{bmatrix} \frac{1}{EX} & \frac{NUTHX}{ETH} & \frac{NUZX}{EZ} & 0 \\ \frac{NUXTH}{EX} & 1 & \frac{NUZTH}{EZ} & 0 \\ \frac{NUXZ}{EX} & \frac{NUTHZ}{ETH} & \frac{1}{EZ} & 0 \\ 0 & 0 & 0 & \frac{1}{GZX} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_\theta \\ \sigma_z \\ \sigma_{zx} \end{Bmatrix} + (T - TREF) \begin{Bmatrix} AX \\ ATH \\ AZ \\ 0 \end{Bmatrix}$$

Note that:

$$\begin{aligned}
 \frac{NUXTH}{EX} &= \frac{NUTHX}{ETH} \\
 \frac{NUZX}{EZ} &= \frac{NUXZ}{EX} \\
 \frac{NUTHZ}{ETH} &= \frac{NUZTH}{EZ}
 \end{aligned}$$

The compliance matrix as shown must be positive definite. This requires that EX, ETH, EZ, and GZX be positive. There are additional restrictions as well, see a standard text on elasticity.

9. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
10. TREF is used in two different ways (See Remarks 6. and 7. of TEMPERATURE Case Control entry for details.):
  - Except for SOL106 and SOL400 nonlinear analysis, TREF is used as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, in which case, TREF is ignored.
  - In SOL106 and SOL400 nonlinear analysis, TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-92](#) in Remark 10 of the MAT1 description.
11. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters](#).

## MAT4 Heat Transfer Material Properties, Isotropic

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	$\rho$	H	$\mu$	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	17	6.66-4	5.01-5	456.2		1.03-6			
------	----	--------	--------	-------	--	--------	--	--	--

Describer	Meaning
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real $\geq$ 0.0)
$\rho$	Density. (Real $\geq$ 0.0; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
$\mu$	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real $\geq$ 0.0; Default = 1.0)
REFENTH	Reference enthalpy. (Real or blank)
TCH	Lower temperature limit at which phase change region is to occur. See Remark 6. (Real or blank)
TDELTA	Total temperature change range within which a phase change is to occur. (Real $\geq$ 0.0 or blank)
QLAT	Latent heat of fusion per unit mass associated with the phase change. (Real > 0.0 or blank)

Remarks:

1. The MID must be unique with respect to all other MAT4 and MAT5 entries. MAT4 may specify material properties for any conduction elements as well as properties for a forced convection fluid (see CONVM). MAT4 also provides the heat transfer coefficient for free convection (see CONV).
2. For a forced convection fluid,  $\mu$  must be specified.
3. REFENTH is the enthalpy corresponding to zero temperature if the heat capacity CP is a constant. If CP is obtained through a TABLEM lookup, REFENTH is the enthalpy at the first temperature in the table.

4. Properties specified on the MAT4 entry may be defined as temperature dependent by use of the entry.
5. For RC network solver in thermal analysis, the REFENTH, TCH, TDELTA and QLAT are ignored.
6. The advanced nonlinear elements should be used for phase change to achieve accurate results.

## MAT5 Thermal Material Property Definition

Defines the thermal material properties for anisotropic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MAT5	MID	KXX	KXY	KXZ	KYY	KYZ	KZZ	CP	
	RHO	HGEN							

Example:

MAT5	24	.092			.083		0.20	0.2	
	2.00								

Describer	Meaning
MID	Material identification number. (Integer > 0)
Kij	Thermal conductivity. (Real)
CP	Heat capacity per unit mass. (Real ≥ 0.0 or blank)
RHO	Density. (Real > 0.0; Default = 1.0)
HGEN	Heat generation capability used with QVOL entries. (Real ≥ 0.0; Default = 1.0)

Remarks:

1. The thermal conductivity matrix has the following form:

$$K = \begin{bmatrix} KXX & KXY & KXZ \\ KXY & KYY & KYZ \\ KXZ & KYZ & KZZ \end{bmatrix}$$

2. The material identification number may be the same as a MAT1, MAT2, or MAT3 entry but must be unique with respect to other MAT4 or MAT5 entries.
3. MAT5 materials may be made temperature-dependent by use of the MATT5 entry.
4. When used for axisymmetric analysis (CTRIAX6), material properties are represented where:  
KXX = radial conductivity component  
KYY = axial conductivity component
5. The KZZ is required if using the advanced nonlinear Shell element with temperature across thickness called out by NLMOPTS, TEMPP, LINE or QUAD option.



## MAT8 Shell Element Orthotropic Material Property Definition

Defines the material property for an orthotropic material for isoparametric shell elements.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	GE	F12	STRN						
	“HFAIL”	HF1	HF2	HF3	HF4	HF10	HF11		
	“HTAPE”	HT1	HT2	HT3	HT4	HT5	HT6	HT10	
		H11	HT12						
	“HFABR”	HFB1	HFB2	HFB3	HFB4	HFB5	HFB6	HFB10	
		HFB11	HFBT12						

Example:

MAT8	171	30.+6	1.+6	0.3	2.+6	3.+6	1.5+6	0.056	
	28.-6	1.5-6	155.0	1.+4	1.5+4	2.+2	8.+2	1.+3	
	1.-4		1.0						
	“HFAIL”	1010.	1020.	33.	34.	1011.	1012.		
	“HTAPE”	2001.	2002.	35.	36.	2003.	1.0	1004.	
		1007.	1008.						
	“HFABR”	3001.	3002.	3003.	3004.	1005.	1005.	1006.	
		1007.	1008.						

Describer	Meaning
MID	Material identification number. Referenced on a PSHELL, PCOMP or PCOMPG entry only. (0 < Integer < 100,000,000)
E1	Modulus of elasticity in X material coordinate direction, also defined as the fiber direction or 1-direction. (Real > 0.0)
E2	Modulus of elasticity in Y material coordinate direction, also defined as the matrix direction or 2-direction. (Real > 0.0)
NU12	Poisson’s ratio ( $\epsilon_2/\epsilon_1$ for uniaxial loading in 1-direction). Note that $\nu_{21} = \epsilon_1/\epsilon_2$ for uniaxial loading in 2-direction is related to $\nu_{12}$ , $E_1$ , and $E_2$ by the relation $\nu_{12}E_2 = \nu_{21}E_1$ . (Real)
G12	In-plane shear modulus. (Real > 0.0; Default = 0.0)
G1Z	Transverse shear modulus for shear in 1-Z plane. (Real $\geq$ 0.0 or blank; Default implies infinite shear modulus.)



Describer	Meaning
G2Z	Transverse shear modulus for shear in 2-Z plane. (Real $\geq 0.0$ or blank; Default implies infinite shear modulus.)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient in i-direction. (Real)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal expansion coefficient. See Remarks 4. and 5. (Real or blank)
Xt, Xc	Allowable stresses or strains in tension and compression, respectively, in the longitudinal direction. Required if failure index is desired. See the FT field on the PCOMP/PCOMPG entry. (Real $> 0.0$ ; Default value for Xc is Xt.)
Yt, Yc	Allowable stresses or strains in tension and compression, respectively, in the lateral direction. Required if failure index is desired. (Real $> 0.0$ ; Default value for Yc is Yt.)
S	Allowable stress or strain for in-plane shear. See the FT field on the PCOMP/PCOMPG entry. (Real $> 0.0$ )
GE	Structural damping coefficient. See Remarks 4. and 6. (Real)
F12	Interaction term in the tensor polynomial theory of Tsai-Wu. Required if failure index by Tsai-Wu theory is desired and if value of F12 is different from 0.0. See the FT field on the PCOMP/PCOMPG entry. (Real)
STRN	For the maximum strain theory only (see STRN in PCOMP/PCOMPG entry). Indicates whether Xt, Xc, Yt, Yc, and S are stress or strain allowables. (Real = 1.0 for strain allowables; blank (Default) for stress allowables.)
“HFAIL”	Keyword indicating that Hashin failure criterion, to calculate its four failure indices.
HF1	Maximum fiber tensile stress, no default
HF2	Maximum fiber compressive stress, default=HF1
HF3	Maximum matrix tensile stress, no default
HF4	Maximum matrix compressive stress, default=HF3
HF10	Maximum in-plane shear stress, no default
HF11	Maximum transverse shear stress, default=HF10
“HTAPE”	Keyword indicating that Hashin-Tape criterion, a variant of the Hashin criterion, adapted for tape type of materials are calculated.
HT1	Maximum tape fiber tensile stress, no default
HT2	Maximum tape fiber compressive stress, default=HT1
HT3	Maximum tape cross-fiber tensile stress, no default
HT4	Maximum tape cross-fiber compressive stress, default=HT3
HT5	Maximum fiber tensile stress for matrix compression; Required if HT6=1.0, otherwise not used.

Describer	Meaning
HT6	Contribution factor for HT5 (Real, 0.0 or 1.0, default = 0.0)
HT10	Maximum in-plane shear stress, no default
HT11	Maximum transverse shear stress, default=HT10
HT12	Maximum z-x transverse shear stress, default=HT11
“HFABR”	Keyword indicating that Hashin-Fabric criterion, a variant of the Hashin criterion, adapted for fabric type of materials are calculated.
HFB1	Maximum first fiber tensile stress, no default
HFB2	Maximum first fiber compressive stress, default=HFB1
HFB3	Maximum second cross-fiber tensile stress, no default
HFB4	Maximum second cross-fiber compressive stress, default=HFB3
HFB5	Maximum thickness tensile stress, no default
HFB6	Maximum thickness compressive stress, default=HFB5
HFB10	Maximum in-plane shear stress, no default
HFB11	Maximum transverse shear stress, default=HFB10
HFB12	Maximum z-x transverse shear stress, default=HFB11

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. If G1Z and G2Z values are specified as zero or blank, then transverse shear flexibility calculations will not be performed, which is equivalent to zero shear flexibility (i.e., infinite shear stiffness). If MAT8 is referenced by Advanced Nonlinear Element, G1Z and G2Z must be positive.
3. An approximate value for G1Z and G2Z is the in-plane shear modulus G12. If test data are not available to accurately determine G1Z and G2Z for the material and transverse shear calculations are deemed essential; the value of G12 may be supplied for G1Z and G2Z. In SOL 106, linear and nonlinear elastic material properties in the residual structure will be updated as prescribed in the TEMPERATURE Case Control command.
4. Xt, Yt, and S are required for composite element failure calculations when requested in the FT field of the PCOMP/PCOMPG entry. Xc and Yc are also used but not required.
5. TREF and GE are ignored if this entry is referenced by a PCOMP/PCOMPG entry.
6. TREF is used in two different ways (See Remarks 6. and 7. of TEMPERATURE Case Control entry for details.):
  - Except for SOL106 and SOL400 nonlinear analysis, TREF is used as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, in which case, TREF is ignored.

- In SOL106 and SOL400 nonlinear analysis, TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-92](#) in Remark 10 of the MAT1 description.
7. If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters](#).
  8. In nonlinear static analysis (SOL 106) the QUAD4 and TRIA3 thermal loads are computed using the secant (default) method. To use the more accurate integral method, specify 'PARAM,EPSILONT,INTEGRAL' in bulk data. See [Parameters](#).
  9. If MAT8 with Hashin is used in SOL 400 with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

## MAT8A Orthotropic Failure Material Properties - SOL700

Defines the failure properties for an orthotropic material for shell elements. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8A	MID	FT	NV	S	ALPHA	TRSFAIL	F12		
	XT	XC	YT	YC	PFD	VALUE	PFDST		
	FBTEN	FBCOM	MXTEN	MXCOM	MXSHR				
	PRDFT	PRDFC	PRDMT	PRDMC	PRDSH				

Example:

MAT8A	7	COMBINAT		100.					
+	200.0	150.0	100.0	110.0	STEPS	200.0			
+	CHANG	STRSS	MODSAI	MODTSAI	STRSS				
+									
+					0011				

Describer	Meaning
MID	Material identification. See Remark 1. (Integer > 0, Required.)
FT	Failure theory to be used to test whether the element layer fails. See Remark 2. (Character; default=blank) Blank            No failure HILL            Tsai-Hill theory TSAI            Tsai-Wu theory MODTSAI      Modified Tsai-Wu theory STRSS           Maximum stress CHANG        Chang-Chang theory COMBINAT    Combination (See Remark 6.) HASHIN        Hashin theory
NV	Number of additional history variables for a user model. See Remark 8. Currently it is not available. (0 < Integer < 10; default=0)
S	Failure stress for in-plane shear. See Remark 4. (Real > 0.0; required.)
ALPHA	Nonlinear shear coefficient. See Remark 5. (Real ≥ 0.0; default=0.0)
TRSFAIL	Transverse shear failure. (Character; default=SUBL)

Describer	Meaning
	ELEM Failure if element fails
	SUBL Failure if sublayer fails
F12	Interaction term in Tsai-Wu theory. (Real, default=0.0)
XT	Tensile failure stress in the large structural direction. See Remark 4. (Real $\geq$ 0.0; default=0.0)
XC	Compressive failure stress in the large structural direction. See Remark 4. (Real $\geq$ 0.0; default=0.0)
YT	Tensile failure stress in the lateral direction. See Remark 4. (Real $\geq$ 0.0; default=0.0)
YC	Compressive failure stress in the lateral direction. See Remark 4. (Real $\geq$ 0.0; default=0.0)
PFD	Post-failure degradation model. See Remark 8. (Character; default=STEPS)
	STEPS Degrade stresses by time steps
	TIME Degrade stresses by time
	VELOC Degrade stresses by velocity
VALUE	Depending on PFD, VALUE gives the number of time steps, time interval, or propagation velocity. (Real > 0; default=100.0)
PFDST	Post-failure degradation start. See Remark 8. (Character; default=INDV)
	INDV Stresses are degraded per distinct failure mode.
	ALL Stresses are degraded if all elastic constants are zero.
FBTEN	Tensile failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)
FBCON	Compressive failure modes in fiber direction for individual failure definition. See Remark 6. (Character; required.)
MXTEN	Tensile failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)
MXCOM	Compressive failure modes in matrix direction for individual failure definition. See Remark 6. (Character; required.)
MXSHR	Shear failure modes for individual failure definition. See Remark 6. (Character; required.)
PRDFT	Property degradation due to fiber-tension failure. See Remark 7. (Integer; default=1111)
PRDFC	Property degradation due to fiber-compression failure. See Remark 7. (Integer; default=1010)
PRDMT	Property degradation due to matrix-tension failure. See Remark 7. (Integer; default=0110)

Describer	Meaning
PRDMC	Property degradation due to matrix-compression failure. See Remark 7. (Integer; default=0110)
PRDSH	Property degradation due to in-plane shear failure. See Remark 7. (Integer; default=0001)

Remarks:

- The material number must refer to a MAT8 material definition.
- If a failure theory is selected other than COMBINAT, the theory defines the following failure modes:

CHANG	Fiber tension, matrix tension/compression
HILL	All modes
TSAI	All modes
MODTSAI	Matrix tension/compression
STRSS	All modes
HASHIN	Fiber tension/compression Matrix tension/compression

For an element to fail completely, both fiber and matrix in all sublayers must fail.

- This material model can only be referenced from a PCOMP entry.
- Failure stresses are required if a failure theory is selected.
- ALPHA is used for all failure theories to define a nonlinear stress-strain relation.
- The individual failure modes are defined according to the corresponding mode in the theory as listed under FT. To be relevant, the theory must define the failure mode (see Remark 2.). You must enter data if FT is set to COMBINAT.
- The property degradation rules due to the various failure modes are listed below:

Material Constant	Failure Mode				
	Fiber Tens	Fiber Comp	Matrix Tens	Matrix Comp	Shear
E1	X	X			
E2	X		X	X	
V12	X	X	X	X	
G12	X				X

The Poisson's ratio Nu21 is treated the same as Nu12.

To override the default model, an integer value is defined as a packed word in the following order:

- 1 Denotes property degradation.
- 0 Denotes no degradation.

The last five fields of the MAT8A Bulk Data entry are input for the user to specify the degradation behavior for each mode of failure.

8. The PFD entry indicates how the stresses are degraded to zero. The PFDST indicates when the stresses start to degrade.

Using ALL means that degradation starts when all material constants (E1, E2, V12, G12) are degraded to zero as specified by the FT entry and the property degradation rules. Note that property degradation means that the stress increments are zero but that the stresses degrade according to PFD.

INDV means that stress degradation starts for the fiber stress if E1=0.0, for matrix stress if E2=0.0, and for shear stress if G12=0.0.

9. Any failure theory introduces five additional sublayer variables. The PFDST entry introduces three additional variables. The number of user variables is defined by NV.



**MAT8F****Shell Element Orthotropic Material Frequency Dependence**

Specifies frequency-dependent material properties on MAT8 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT8F	MID	E1	E2	NU12	G12	G1Z	G2Z		
	GE								

Example:

MAT8F	76								
	97								

Descriptor	Meaning
MID	Material property identification number that matches the identification number on MAT8 entry. (Integer > 0)
E1	Identification number of a TABLEDi entry for the Modulus of elasticity in longitudinal direction, also defined as the fiber direction or 1-direction. (Integer > 0 or blank)
E2	Identification number of a TABLEDi entry for the Modulus of elasticity in lateral direction, also defined as the matrix direction or 2-direction. (Integer > 0 or blank)
NU12	Identification number of a TABLEDi entry for Poisson's ratio. (Integer > 0 or blank)
G12	Identification number of a TABLEDi entry for the in-plane shear modulus. (Integer > 0 or blank)
G1Z	Identification number of a TABLEDi entry for the transverse shear modulus for shear in 1-Z plane. (Integer > 0 or blank)
G2Z	Identification number of a TABLEDi entry for the transverse shear modulus for shear in 2-Z plane. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT8 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT8 then GE table must be blank or 0.

**MAT9** Solid Element (and Shell Element for SOL 600 only) Anisotropic Material Property Definition

Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements (see PSOLID entry description).

Format:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		
	GE11	GE12	GE13	GE14	GE15	GE16	GE22	GE23	
	GE24	GE25	GE26	GE33	GE34	GE35	GE36	GE44	
	GE45	GE46	GE55	GE56	GE66				

Example:

MAT9	17	6.2+3						6.2+3	
					6.2+3				
	5.1+3			5.1+3		5.1+3	3.2	6.5-6	
	6.5-6					125.	.003		

Descriptor	Meaning
MID	Material identification number. (Integer > 0)
Gij	Elements of the 6 x 6 symmetric material property matrix in the material coordinate system. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark 7. (Real or blank)
GE	Structural element damping coefficient. See Remarks 6. and 8. (Real)
GEij	Structural damping matrix. See Remark 9. (Real)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

- MAT9 materials may be made temperature-dependent by use of the MATT9 entry. In nonlinear static analysis (e.g., SOL 106), linear and nonlinear elastic material properties in the residual structure will be updated as prescribed under the TEMPERATURE Case Control command.
- The mass density RHO will be used to compute mass in a structural dynamics problem automatically.
- The third continuation entry is optional.
- The subscripts 1 through 6 refer to x, y, z, xy, yz, and zx of the material coordinate system (see the CORDM field on the PSOLID entry description). The stress-strain relationship is

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = \begin{bmatrix} G11 & G12 & G13 & G14 & G15 & G16 \\ & G22 & G23 & G24 & G25 & G26 \\ & & G33 & G34 & G35 & G36 \\ & & & G44 & G45 & G46 \\ & \text{symmetric} & & & G55 & G56 \\ & & & & & G66 \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} = \begin{Bmatrix} A1 \\ A2 \\ A3 \\ A4 \\ A5 \\ A6 \end{Bmatrix} (T - TREF)$$

- The damping coefficient GE is given by

$$GE = \frac{2.0 \cdot C}{C_0}$$

- TREF is used in two different ways (See Remarks 6. and 7. of TEMPERATURE Case Control entry for details.):
  - Except for SOL106 and SOL400 nonlinear analysis, TREF is used as the reference temperature for the calculation of thermal loads. TEMPERATURE(INITIAL) may be used for this purpose, in which case, TREF is ignored.
  - In SOL106 and SOL400 nonlinear analysis, TREF is used only for the calculation of a temperature-dependent thermal expansion coefficient. The reference temperature for the calculation of thermal loads is obtained from the TEMPERATURE(INITIAL) set selection. See [Figure 9-92](#) in Remark 10 of the MAT1 description.
- If PARAM,W4 is not specified, GE is ignored in transient analysis. See [Parameters](#).
- Defines structural damping matrix data for advanced composites. If the GEij values are present, the Nastran will ignore the GE value given on the third continuation entry field (8) and use the given GEij values. The rationale of these entries is that with composite materials made of a viscoelastic resin system, the damping matrix does not scale with the stiffness matrix. For example, in the directions where you have more fibers, the stiffness is higher because you have more fibers but the damping coefficient is smaller because you have less resin matrix which is the main source of damping. To use this feature in composites, the PCOMPLS MIDi entries must refer to MAT9 entries.

The damping is applied in element material routines by forming the product  $GE_{ij} * G_{ij}$  when forming the K4 damping matrix in EMG.

## MAT9F Solid Element Anisotropic Frequency Dependence

Specifies frequency-dependent material properties on MAT9 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT9F	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66			
							GE		
	GE11	GE12	GE13	GE14	GE15	GE16	GE22	GE23	
	GE24	GE25	GE26	GE33	GE34	GE35	GE36	GE44	
	GE45	GE46	GE55	GE56	GE66				

Example:

MAT9F	101	5						5	
					5				
				5		5			

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
Gij	Identification number of a TABLEDi entry for the material property matrix. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)
GEij	Identification number of a TABLEDi entry for the element structural damping coefficients. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value in a particular field of the MAT9 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT9 entry.
- Table references must be present for each item that is frequency dependent.
- IF GE=0.0 on corresponding MAT9 then GE table must be blank or 0. For any GEij=0.0 on corresponding MAT9, the corresponding GEij table must be blank or 0.

## MAT10 Fluid Material Property Definition

Defines material properties for fluid elements in coupled fluid-structural analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10	MID	BULK	RHO	C	GE	ALPHA			

Example:

MAT10	103	0.656	0.011						
-------	-----	-------	-------	--	--	--	--	--	--

Describer	Meaning
MID	Material identification number. (Integer > 0)
BULK	Bulk modulus. (Real > 0.0)
RHO	Mass density. (Real > 0.0)
C	Speed of sound. (Real > 0.0)
GE	Fluid element damping coefficient. (Real)
ALPHA	Normalized admittance coefficient for porous material. See Remark 7. (Real or blank)

Remarks:

- MAT10 is referenced, with MID, by the PSOLID entry only.
- The material identification numbers must be unique for all MAT1, MAT2, MAT3, MAT9, and MAT10 entries.
- The mass density RHO will be used to compute the mass automatically.
- BULK, RHO, and C are related by

$$\text{BULK} = C^2 \cdot \text{RHO}$$

Two out of the three must be specified, and the other will be calculated according to this equation. If all three are specified and are inconsistent in values, the supplied values of BULK and RHO are used in the computations.

- To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$ , by 2.0.
- If PARAM,W4FL is not specified, GE is ignored in transient analysis. See [Parameters](#).
- If a value of ALPHA is entered, BULK RHO and GE may have negative values.
- The value defined in the ALPHA field always defines the normalized admittance coefficient for porous material but it is differently interpreted depending on the value defined in the FCNT field of the referencing PSOLID entry.

- a. If the MAT10 entry is referenced in a PSOLID entry where FFLUID option is selected, the value defined for ALPHA is considered as the normalized admittance coefficient calculated at unit circular excitation frequency ( $\omega = 1$ ). Its value will be automatically calculated by the program, at each excitation frequency, considering the current circular excitation frequency as scaling factor.
  - b. If the MAT10 entry is referenced in a PSOLID entry where PFLUID option is selected, the value defined for ALPHA has no special meaning but it is only the normalized admittance coefficient calculated by the user at the most appropriate excitation frequency (defined in order to have good results in the frequency range of interest).
9. For Poro-Elastic Material, PEM, user is expected to provide adiabatic BULK modulus for fluid-phase of PEM which is equal to GAMMA, fluid ratio of specific heats (see MATPE1 for detailed), times isothermal BULK modulus. Isothermal BULK modulus is BULK modulus at constant temperature for fluid itself.
  10. When used in conjunction with PSLDN1, only pyramid elements can be associated with the MAT10 fluid property. CHEXA, CPENTA, CTETRA elements using MAT10 are not supported with PSLDN1.
  11. The terms of the fluid mass matrix are not multiplied by the value of PARAM and WTMASS.

## MAT10F Fluid Frequency Dependence

Specifies frequency-dependent material properties on MAT10 entry fields via TABLEDi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MAT10F	MID	BULK	RHO	C	GE	ALPHA			

Example:

MAT10	103	666							
-------	-----	-----	--	--	--	--	--	--	--

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT10 entry. (Integer > 0)
BULK	Identification number of a TABLEDi entry for the Bulk modulus. (Integer > 0 or blank)
RHO	Identification number of a TABLEDi entry for the mass density. Used only in SOL108. (Integer > 0 or blank)
C	Identification number of a TABLEDi entry for the speed of sound. Used only in SOL108. (Integer > 0 or blank)
GE	Identification number of a TABLEDi entry for the element structural damping coefficient. (Integer > 0 or blank)
ALPHA	Identification number of a TABLEDi entry for the normalized admittance coefficient for porous material. See Remark 4.

Remarks:

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT10 entry referenced in field 2. The value in a particular field of the MAT10 entry is replaced by the table referenced in the corresponding field of this entry. Blank or zero entries mean that there is no frequency dependence of the fields on the MAT10 entry. The nominal value of GE defined on MAT10 needs to be non-zero to get frequency dependent damping defined in the GE table on MAT10F.
- Table references must be present for each item that is frequency dependent.
- When ALPHA on MAT10 is specified it is multiplied by the excitation frequency in radians/unit time. If F(ALPHA) is specified then the value supplied by F(ALPHA) will take precedence and ALPHA WILL NOT BE MULTIPLIED by the excitation frequency.



**MATDEUL****General Constitutive Model to be Used for the Eulerian Materials - SOL 700**

Defines a complete constitutive model as a combination of an equation of state, a shear model, a yield model, a failure model, a spall model (PMIN), and corotational frame. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATDEUL	MID	RHO	EID	SID	YID	FID	PID		
	BULKL	BULKQ			BULKTYP				

Example:

MATDEUL	22	3000.	100	109	307	308	402		
---------	----	-------	-----	-----	-----	-----	-----	--	--

Describer	Meaning
MID	Unique material number. (Integer > 0; Required)
RHO	Density. (Real > 0.0; Required)
EID	Number of an EOSxxx entry defining the pressure/density characteristic of the material. (Integer > 0; Required)
	EOSGAM      Gamma Law Gas Equation of State
	EOSIG      Ignition and Growth Equation of State
	EOSJWL      JWL Explosive Equation of State
	EOSMG      Mie-Gruneisen Equation of State
	EOSPOL      Polynomial Equation of State
	EOSTAIT      Tait Equation of State
SID	Number of a SHRxxx entry defining the shear properties of the material. (Integer ≥ 0; Hydrodynamic shear model)
	SHREL      Elastic Shear Model
	SHRPOL      Polynomial Shear Mode
YID	Number of a YLDxxx entry defining the yield model for the material. (Integer ≥ 0; Hydrodynamic yield model)
	YLDHY      Hydrodynamic Yield Model
	YLDJC      Johnson-Cook Yield Model
	YLDMC      Mohr-Coulomb Yield Model
	YLDMSS      Multi-surface yield model for Snow
	YLDPOL      Polynomial Yield Model
	YLDRPL      Rate Power Law Yield Model

Describer	Meaning
	YLDYG Steinberg-Guinan Yield
	YLDTM Tanimura-Mimura Yield Model
	YLDVM von Mises Yield Model
	YLDZA Zerilli-Armstrong Yield Model
FID	Number of a FAILMPS entry defining the failure model for the material. (Integer $\geq 0$ ; no failure)
	FAILMPS Maximum Plastic Strain Failure Model
PID	Number of a PMINC entry defining the spallation characteristics of the material. See Remark 3. (Integer $> 0$ )
BULKL	Linear bulk-viscosity coefficient. (Real $\geq 0.0$ ; 0.0)
BULKQ	Quadratic bulk-viscosity coefficient. (Real $> 0.0$ ; 1.0)
BULKTYP	Bulk viscosity type. (Character, DYNA)
	DYNA Standard DYNA3D model.
	DYTRAN Enhanced DYNA model.

Remarks:

1. If YID is blank or zero, a hydrodynamic yield model is used.
2. If the TYPE field on the PEULER entry is set to HYDRO, then YID is either blank or references a YLDHY entry, and SID is blank. Conversely, if the TYPE field is set to STRENGTH, a nonhydrodynamic yield model is specified.
3. If no PMINC entry is referenced, a minimum pressure of zero is assumed for the Eulerian elements. The PMINC entry will be ignored when a cavitation model through the EOSTAIT entry has been given.

## MATDIGI Material Digimat

Defines material data for the advanced composites with Digimat from e-Xstream engineering (SOL 400 only). For more information about e-Xstream engineering and Digimat, please contact [support@e-xstream.com](mailto:support@e-xstream.com) or consult <http://www.e-xstream.com/>

Format:

1	2	3	4	5	6	7	8	9	10
MATDIGI	MID	UDID					RHO		

Example:

MATDIGI	5	10					3000.		
---------	---	----	--	--	--	--	-------	--	--

Describer	Meaning
MID	Material identification number. (Integer > 0)
UDID	References UDNAME entry (Required, Integer>0)
RHO	Density (Real>0.0 or blank)

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. SOL400 uses only the MID, UDID and RHO entries. All other entries are ignored by SOL400.
3. The entry UDSESV is required input to define the number of state variables which will be used by MATDIGI. The number of state variables (NSTATS) must be larger than 1.
4. License feature NASTRAN\_DIGIMAT must be present to use this feature. For parallel computations, NASTRAN\_DIGIMAT\_DMP must be present.
5. The use of MATDIGI is limited to shell and solid elements.
6. MATDIGI should not be used in SOL 400 perturbation steps.



MATEP

Elasto-Plastic Material - SOL 400

Elasto-plastic material properties for SOL 400 only.

Format: Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD	WKHARD		H	
	“Reffect”	OPTION	RTID	C	P				
	“Aniso”	N/A	R11 or M	R22 or C1	R33 or C2	R12 or C3	R23 or C6	R31	
	“Press”	OPTION	ALPHS	BETA					
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	$\mu$	$\eta$					
	“JhCook”	A	B	N	C	M	Tmelt	Troom	
		$\epsilon 0DOT$							
	“IMPCREEP”	VMISES							

Example:

MATEP	100	TABLE		20		HILL			
	ANISO		1.1	0.9		1.02			

Describer	Meaning
MID	Identification number of MAT1, MAT2, MATORT, MAT8 or MAT9 entry. (Integer > 0).
FORM	Selects a form of stress-plastic strain function to be specified (Character): SLOPE Defines nonzero H. (Default) TABLE Defines the function using TABLES1, TABLEST or TABL3D entries. Units of table are unspecified PERFECT Defines perfectly plastic material (H is zero or blank and FID blank). See Remarks 2.
Y0	Initial yield stress $Y_0$ or hydrostatic stress for Mohr-Coulomb materials. See Remark 2. (Real > 0 or blank)
FID	Identification number of TABLES1 or TABL3D entry. See Remarks 2, 3, and 4. (Integer > 0 or blank)
RYIELD	Enter on of the following yield criteria rules. (Character; Default = VMISES) See Remarks 12.



Describer	Meaning
	VMISES                      von Mises yield criteria (Default)
	HILL                              Hill's 1948 yield criteria
	BARLAT                        Barlat's 1991 yield criteria
	LINMOHR                      Linear Mohr-Coulomb yield criteria
	PBLMOHR                     Parabolic Mohr-Coulomb yield criteria
	IMPCREEP                    Implicit creep model combining both plasticity and creep, von Mises yield criteria.
WKHARD	Selects a hardening rule defined by various work-hardening rules (Character). See Remarks <a href="#">12</a> .
	ISOTROP                        Isotropic hardening. (Default)
	KINEM                            Kinematic hardening.
	COMBINE                      Combination between kinematic and isotropic hardening. (Default kinematic fraction=0.5).
	CHABOCHE                    Caboche formulation - like combined (see MSC Nonlinear User Guide (SOL 400)) for details of this model).
	JHCOOK                        For Johnson-Cook formulation (see MSC Nonlinear User Guide (SOL 400)) for details of this model).
	See Remark <a href="#">5</a> .
H	Workhardening slope. See Remarks <a href="#">2</a> , <a href="#">3</a> , and <a href="#">4</a> . (Real $\geq 0$ ; Default = 0)
"Reffect"	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABL3D is entered and one of the independent variables is strain rate, this field and its options should not be specified.
OPTION	Selects an option for strain-rate dependent yield stress (Character):
	TABLE                            Specifies that TABLES1 input used to define rate-dependency. (Default)
	COWPER                        Specifies the Cowper and Symonds rate-dependency model. See Remark <a href="#">6</a> .
RTID	TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)
"Aniso"	A keyword signifying that the following data (Rij) pertain to the Hill's (Rij) or Barlat's (M, Ci) yield criteria. an isotropic material option. See Remark <a href="#">7</a> . and MSC Nonlinear User Guide (SOL 400) (specify only if RYIELD = HILL or BARLAT).

Describer	Meaning
Rij	Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real > 0; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill) or
M	Barlat M coefficient
C1	Barlat C1 coefficient
C2	Barlat C2 coefficient
C3	Barlat C3 coefficient
C6	Barlat C6 coefficient
“Press”	A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield = LinMohr or PblMohr cracking is to be simulated. See Remarks 8. and 9.
OPTION	Selects an option for pressure-dependent yield criteria. (Character): LIN                                      Linear Mohr-Coulomb model. (Default) PBL                                      Parabolic Mohr-Coulomb model.
ALPHA	Specifies the parameter alpha for linear Mohr-Coulomb model. (Real > 0 or blank)
BETA	Specifies the parameter beta for parabolic Mohr-Coulomb model. (Real > 0 or blank)
“Chaboche”	A keyword specifying the following data pertains to the Chaboche model. (Enter only if WKHARD=CHABOCHE.)
R0	R0 for isotropic hardening (Real)
Rinf	Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization) (Real)
B	Coefficient B for isotropic hardening (Real)
C	C coefficient C for kinematic hardening (Real)
Gam	Gamma coefficient for kinematic hardening (Real)
Kap	Kappa value for viscosity model (Real)
N	Coefficient N for viscosity model (Real)
Qm	Coefficient Qm for isotropic hardening (Real)
μ	Coefficient μ for isotropic hardening (Real)
η	Coefficient η to introduce progressive memory (Real)
“JhCook”	A keyword specifying the following data pertains to the Johnson-Cook model (see the MSC Nonlinear User Guide (SOL 400) for more details. Enter only if WKHARD = JHCOOK).
A	Coefficient A. (Real)
B	Coefficient B. (Real)
N	Exponent n. (Real)

Describer	Meaning
C	Coefficient C. (Real)
M	Exponent m. (Real)
Tmelt	Melting temperature. (Real)
Troom	Ambient temperature. (Real)
$\epsilon_0$ Dot	Reference strain rate. (Real)
“IMPCREEP”	A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used only when RYIELD = IMPCREEP).
VMISES	Equivalent (von Mises) stress. Defines the yield stress for the plasticity element. When used in conjunction with an implicit creep model. If not specified, only the Creep mechanism is active.

Remarks:

1. Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
2. If FORM = SLOPE, the one defines the initial yield stress with  $Y_0$  and a single workhardening slope

is entered as  $H$ .  $H = \frac{\partial \bar{\sigma}}{\partial \bar{\epsilon}^p}$ . In a uniaxial test,  $\epsilon_0$  is the strain at which yielding first occurs is often considered the 0.2% offset strength.

If FORM = TABLE, then the table ID is entered using FID field,  $Y_0$  is not used. The table can be entered as TABLES1 or TABL3D.

When using TABLES1 entry with Type=1, one is defining  $\bar{\sigma} = \bar{\sigma}(\bar{\epsilon})$  as shown in [Figure 9-95](#).

When using TABLES1 entry with Type=2, or TABL3D entry, one is defining  $\bar{\sigma} = \bar{\sigma}(\bar{\epsilon}^p)$  as shown in [Figure 9-96](#).



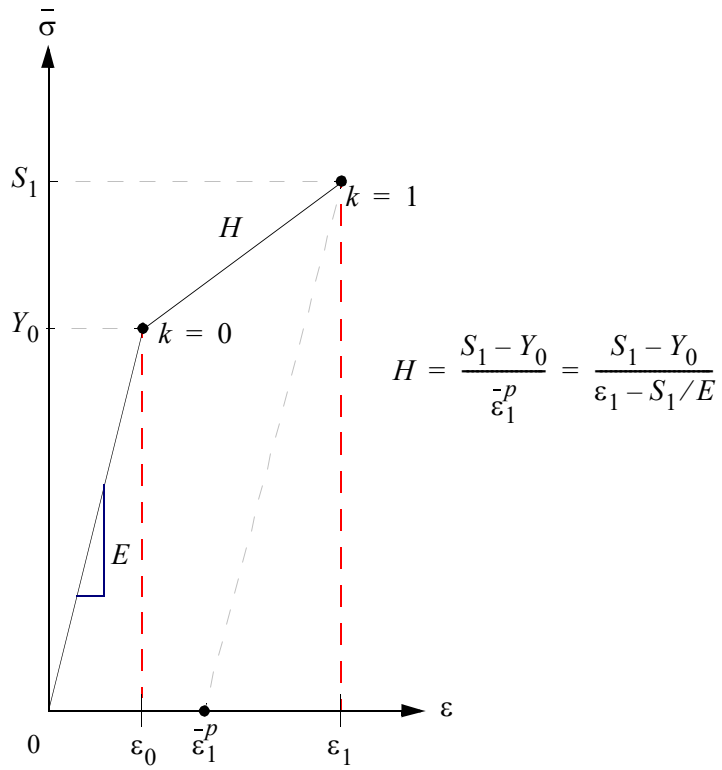


Figure 9-94 Single Workhardening Slope

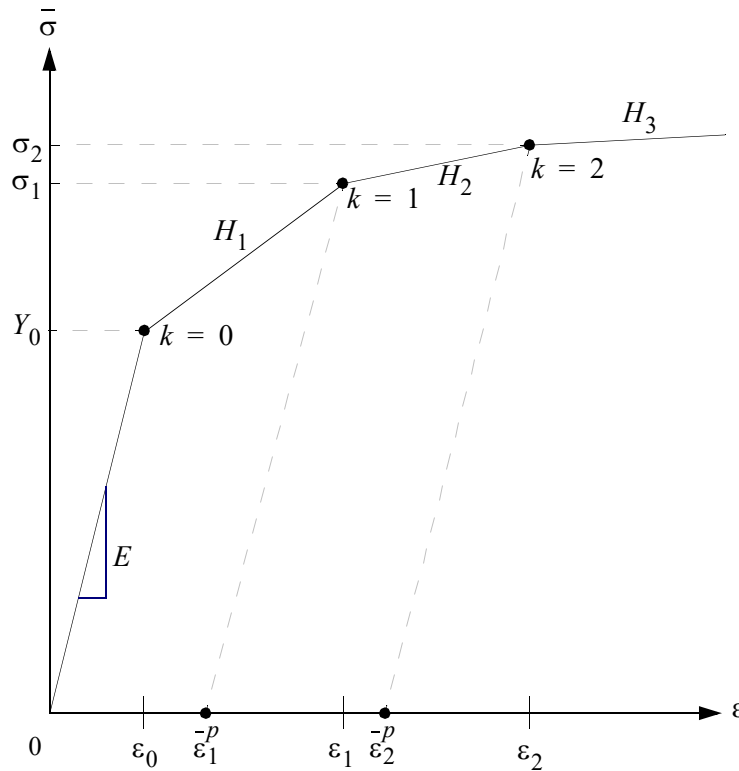


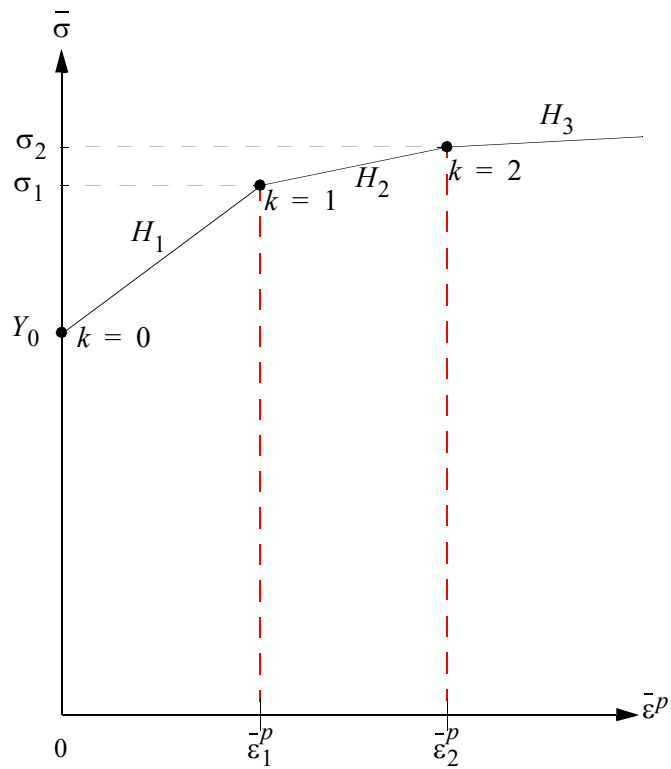
Figure 9-95 Stress vs Total Strain TABLES1(TYPE1)

Figure 9-96 Stress vs Plastic Strain TABLES1(TYPE2) or TABL3D

If FORM = PERFECT,  $Y_0$  is the yield stress and  $H$  and FID should be blank. This is known as elastic-perfectly plastic behavior. If temperature-dependent yield is present, then FORM = TABLE should be used and the temperature-dependent yield entered through TABL3D.

3. For temperature-dependent materials, one can either define the temperature variation through the TABL3D option or include the MATTEP option. The first procedure is recommended.
4. In a large displacement analysis (LGDISP = 1, 2, 11, 12), the Cauchy stress and logarithmic strain are used.
5. The plastic deformation starts when the effective stress ( $\bar{\sigma}$ ) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ( $\bar{\epsilon}^p$ ). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape.



Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

- The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[ 1 + \left( \frac{\dot{\varepsilon}}{C} \right)^{1/P} \right]$$

7. Hill's anisotropic model introduces orthotropic plastic material. The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_y - \sigma_z)^2 + G(\sigma_z - \sigma_x)^2 + H(\sigma_x - \sigma_y)^2 + 2L\tau_{yz}^2 + 2M\tau_{zx}^2 + 2N\tau_{xy}^2}$$

in which the material parameters can be related to the yield stress ratios by

$$F = \frac{1}{2} \left( \frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left( \frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left( \frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$R_{11} = \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the tensile yield stresses measured in material directions x, y and z, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\varepsilon_w}{\varepsilon_t} = \frac{\ln(w_0/w)}{\ln(t_0/t)} = \frac{H + (2N - F - G - 4H)\sin^2\theta \cos^2\theta}{F \sin^2\theta + G \cos^2\theta}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\theta$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F, G, H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left( r_{45} + \frac{1}{2} \right) \left( 1 + \frac{r_0}{r_{90}} \right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45} + 1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

8. The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$aI_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters  $\alpha$  and  $\bar{\sigma}$  (effective stress coinciding with the yield stress) can be related to material constants  $c$  (cohesion) and  $\phi$  (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1 - 12\alpha^2)}}$$

and

$$\sin \phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2} + \sqrt{3}\beta\bar{\sigma}I_1 - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

9. The keywords may appear in any order. However, aniso and press, are mutually exclusive, and cannot coexist.
10. All the alphanumeric fields are recognizable by the first four letters.
11. In SOL 400, MATEP is only supported for nonlinear elements with property extensions. This implies that for such elements, PBARL, PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral). If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data. See remark 9. under the NLMOPTS
12. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE and JHCOOK are only applicable to isotropic materials (MAT1).
13. MATEP can be used to define optional plasticity properties in conjunction with explicit or implicit creep that has been defined through the MATVP card.

- a. For explicit creep, supported yield criteria include RYIELD = VMISES, HILL, BARLAT, LINMOHR and PBLMOHR. All conventional data defined through fields like Y0, WKHARD, etc. are used to define the plasticity properties. Note that in this case, the plasticity is treated implicitly whereas the creep is treated explicitly.
- b. For implicit creep, plasticity properties are defined through special options of the MATEP card - RYIELD field has to be set to IMPCREEP field and the actual yield stress has to be set on the VMISES field under the IMPCREEP option. In this case, both creep and plasticity are treated implicitly. Note - in this case, only Von Mises yield criterion is satisfied. The creep happens immediately whereas the plasticity happens only after the equivalent stress exceeds the VMISES value.

MATEP

Elasto-Plastic Material Properties - SOL 600

Specifies elasto-plastic material properties to be used for large deformation analysis. Used in SOL 600 only.

Format (Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered):

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD	WKHARD	METHOD	H	
	“Reffect”	OPTION	RTID	C	P				
	“Aniso”	N/A	R11 or M	R22 or C1	R33 or C2	R12 or C3	R23 or C6	R31	(Hill/Barlat)
	“ORNL”	OPTION	Yc10	TID	N/A				
	“Press”	OPTION	ALPHA	BETA	CRACKS	SOFTEN	CRUSHS	SRFAC	
	“Gurson”	Q1	Q2	INITIAL	CRITICAL	FAILURE	NUCL	MEAN	
		SDEV	NFRAC						
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	$\mu$	$\eta$					
	“PwrLaw”	A	M	B	N	$\sigma_0 \epsilon_0$			
	“Kumar”	B0	A	B1	B2	B3	T(B0)	T(A)	
		N	B4	B5	B6	T(N)			
	“JhCook”	A	B	N	C	M	Tmelt	Troom	
		$\epsilon_0 \text{DOT}$							
	“YldOpt”	OPTION							
	“Units”	Uopt							
	“VParam”	Nvp	Vp1-1	Vp2-1	Vp1-2	Vp2-2			
	“ImpCreep”	VMISES							

Example:

MATEP	100	table		20		HILL	ADDRAD		
	ANISO		1.1	0.9		1.02			

Describer	Meaning
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. (Integer > 0).
FORM	Selects a form of stress-plastic strain function to be specified (Character):
	SLOPE for defining non-zero H. (Default)
	TABLE for defining the function in TABLES1or TABL3Di units of table are unspecified
	PERFECT for defining perfectly plastic material. (H is zero or blank)



Describer	Meaning
Y0	Initial yield stress $Y_0$ or hydrostatic stress for Mohr-Coulomb materials. See Remark 2. (Real > 0 or blank)
FID	Identification number of TABLES1 or TABL3Di entry. (Integer > 0 or blank)
RYIELD	Enter one of the following yield criteria rules. (Character, Default = vMises if there are no continuation lines. Otherwise, the default will be adjusted to agree with information on the continuation lines. For example, if the second line has PRESS in field 2 and LIN in field 3, the default for RYIELD will be LinMohr.)
ELASTIC	for purely elastic material (implies von Mises for multi-directional stresses). Vmises for von Mises yield criteria.
VMISES	von Mises yield criteria
HILL	for Hill's 1948 yield criteria.
BARLAT	for Barlat's 1991 yield criteria
LINMOHR	for Linear Mohr-Coulomb yield criteria. (See SOL 600 Notes/ Remarks regarding this entry in combination with MATS1. This entry overrides.)
PBLMOHR	for parabolic Mohr-Coulomb yield criteria. (See SOL 600 Notes/Remarks regarding this entry in combination with MATS1. This entry overrides.)
BUYUK	for the Buyukozturk concrete model.
ORNL	for one of the ORNL types. (see "Ornl" below)
GENPLAS	for Generalized Plasticity model
VISCPLAS	for Viscoplastic model using user subroutine UVSCPL
RIGID	for Rigid-Plastic material (no elasticity, von Mises yield)
IMPCREEP	for implicit creep model combining both plasticity and creep, von Mises yield criteria.
WKHARD	Selects a hardening rule defined by various work-hardening rules (Character):
ISOTROP	for isotropic hardening. (Default)
KINEM	for kinematic hardening.
COMBINE	for combination between kinematic and isotropic hardening
CHABOCHE	for Chaboche formulation - like combined (see Marc Vol C documentation for details for this model)
PWRLAW	for Power Law formulation (see Marc Vol C documentation for details of this model, for superplastic forming, this is the only available option)

Describer	Meaning
	<p><b>RPWRLAW</b> for Rate Power Law formulation (see Marc Vol C documentation for details of this model)</p> <p><b>JHCOOK</b> for Johnson-Cook formulation (see Marc Vol C documentation for details of this model)</p> <p><b>KUMAR</b> for Kumar formulation - (see Marc Vol C documentation for details of this model). See Remark 4.</p>
METHOD	Selects a material processing method. See Remark 12. (Character):
	<p><b>ADDMEAN</b> for large strain additive decomposition using the mean normal process. (Default)</p> <p><b>ADDRAD</b> for large strain additive decomposition using the radial return process. (SOL 600 only)</p> <p><b>MULTRAD</b> for multiplicative decomposition (FeFp) using the radial return process</p> <p><b>MULTPLN</b> for plane stress or strain FeFp multiplicative decomposition. (SOL 600)</p> <p><b>SMLMEAN</b> for small strain additive decomposition using the mean normal process. (SOL 600 only)</p> <p><b>SMLRAD</b> for small strain additive decomposition using the radial return process. (SOL 600 only)</p>
H	plasticity modulus, ignored if FID field is specified. H has the same meaning as H in Remark 2 of the MATS1 entry. (Real $\geq 0$ ; Default = 0.0)
“Reffect”	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter if rate-dependent effects are not to be included).
OPTION	Selects an option for strain-rate dependent yield stress (Character):
	<p><b>TABLE</b> for TABLES1 input. (Default)</p> <p><b>COWPER</b> for Cowper and Symonds model. See Remark 6.</p>
RTID	TABLES1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1)
“Aniso”	A keyword signifying that the following data (Rij) pertain to the Hill’s (Rij) or Barlat’s (M, Ci) yield criteria. See Remark 8. and Marc Vol A and C documentation (enter only RYIELD = HILL or BARLAT).
Rij	Stress ratios of initial yield stresses in various material directions to the reference yield stress from FID/Y0 field. (Real $> 0$ ; Default = 1.0 for R11, R22, R33, R12, R23, R31, respectively for Hill), or

Describer	Meaning
M	Barlat M coefficient (see Marc Vol A and C documentation)
C1	Barlat C1 coefficient (see Marc Vol A and C documentation)
C2	Barlat C2 coefficient (see Marc Vol A and C documentation)
C3	Barlat C3 coefficient (see Marc Vol A and C documentation)
C6	Barlat C6 coefficient (see Marc Vol A and C documentation)
“ORNL”	A keyword signifying that the following data pertains to the ORNL’s yield criteria. Only enter if RYIELD = ORNL. See Remark 7.
OPTION	Selects an option for ORNL yield criteria (Character): NORM                           for normal ORNL model for stainless steel. (Default) CRMO                           for ORNL 2-1/4 Cr-Mo steel model. REVP                           for ORNL reversed plasticity model. ARST                           for ORNL full alpha reset model.
Yc10	Equivalent 10th cycle tensile yield stress. (Real > 0)
TID	Identification number of TABLES1 entry for normalized 10th cycle stress-plastic strain curve. (Integer; Default = 0)
“Press”	A keyword signifying that the following data pertain to the pressure-dependent yield criteria. Enter only if Ryield=LinMohr, PblMohr or concrete cracking is to be simulated. See Remarks and 9.
OPTION	Selects an option for pressure-dependent yield criteria. (Character): LIN                               for linear Mohr-Coulomb model. (Default) PBL                               for parabolic Mohr-Coulomb model. CONC                           for Buyukozturk concrete model.
ALPHA	Specifies a parameter alpha for linear Mohr-Coulomb model or concrete model. (Real > 0 or blank)
BETA	Specifies a parameter beta for parabolic Mohr-Coulomb model or Buyukozturk concrete model (not used for linear Mohr-Coulomb model). (Real > 0 or blank)
CRACKS	Critical cracking stress for Buyukozturk concrete model. (Real > 0 or blank)
SOFTEN	Tension-softening modulus for Buyukozturk concrete model. (Real ≥ 0; Default = 0)
CRUSHS	Strain at which material crushes. (Real > 0; Default = 1.E10)
SRFAC	Shear retention factor defining shear stress carrying capacity when crack closes for Buyukozturk concrete model. (0 ≤ Real < 1; Default = 0)
“Gurson”	A keyword signifying that the following data pertain to the modified Gurson model for porous metal plasticity with damage effects. Enter only if the Gurson damage model is to be used. See Remark 9.
Q1	First coefficient for the Gurson yield function. (Real ≥ 0; Default = 1.5)

Describer	Meaning
Q2	Second coefficient for the Gurson yield function. (Real $\geq 0$ ; Default = 1)
INITIAL	Initial void volume fraction. ( $0 \leq \text{Real} < 1$ ; Default = 0)
CRITICAL	Critical void volume fraction at which void coalescence starts. ( $0 < \text{Real} < 1$ ; Default = 0.2)
FAILURE	Failure void volume fraction at which the material loses strength. ( $1./q1 < \text{Real} < 1$ ; Default = 0.733)
NUCL	Select a method of void nucleation (character, Default = strain): NONE for no nucleation. STRAIN for plastic strain controlled nucleation. STRESS for stress controlled nucleation. See Remark 9.
MEAN	Mean strain or stress for void nucleation. (Real $> 0$ ; Default = 0.3)
SDEV	Standard deviation in the assumed normal distribution of nucleation strain or stress. (Real $> 0$ ; Default = 0.01)
NFRAC	Volume fraction of nucleating particles for void nucleation. ( $0 < \text{Real} < 0.5$ ; Default = 0.04)
“Chaboche”	A keyword specifying the following data pertains to the Chaboche model (Enter only if Wkhard=Chaboche)
R0	R0 for isotropic hardening. (Real)
Rinf	Rinfinity for isotropic hardening (Q0 in case of using plastic-strain-range memorization). (Real)
B	b coefficient for isotropic hardening. (Real)
C	C coefficient for kinematic hardening (Real)
Gam	Gamma coefficient for kinematic hardening (Real)
Kap	Kappa value for viscosity model. (Real)
N	n coefficient for viscosity model. (Real)
Qm	Qm coefficient for isotropic hardening. (Real)
$\mu$	coefficient for isotropic hardening. (Real)
$\eta$	coefficient to introduce progressive memory. (Real)
“PwrLaw”	A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C , ISOTROPIC option for more details. Enter only if WKHARD =PWRLAW or RPWRLAW).
A	Coefficient A. (Real)
M	Coefficient m. (Real)
B	Coefficient B. (Real)

Describer	Meaning						
N	Exponent n. (Real)						
$\sigma_0 \epsilon_0$	Enter $\sigma_0$ for rate power law or $\epsilon_0$ for the power law model (Real) See Marc Vol C, ISOTROPIC option if this field is left blank or set to 0.0						
“Kumar”	A keyword specifying the following data pertains to the Kumar model (see Marc Vol C , ISOTROPIC option for more details. Enter only if Wkhard =Kumar).						
B0	Coefficient B0. (Real)						
A	Coefficient A. (Real) If A=0.0, B1,B2, and B3 are used						
B1	Coefficient B1. (Real)						
B2	Coefficient B2. (Real)						
B3	Coefficient B3. (Real)						
T(B0)	TABLEM1 ID for coefficient B0. (Integer)						
T(A)	TABLEM1 ID for coefficient A. (Integer)						
N	Exponent n. (Real) If N=0.0, B4, B5 and B6 are used						
B4	Coefficient B4. (Real)						
B5	Coefficient B5. (Real)						
B6	Coefficient B6. (Real)						
T(N)	TABLEM1 ID for exponent n						
“JhCook”	A keyword specifying the following data pertains to the Johnson-Cook model (see Marc Vol C , ISOTROPIC option for more details. Enter only if WKHARD =JHCOOK).						
A	Coefficient A. (Real)						
B	Coefficient B. (Real)						
N	Exponent n. (Real)						
C	Coefficient C. (Real)						
M	Exponent m. (Real)						
Tmelt	Melting temperature. (Real)						
Troom	Ambient temperature. (Real)						
$\epsilon_0$ Dot	Reference strain rate. (Real)						
“YldOpt”	A keyword specifying one of the following yield models will be used.						
OPTION	Select a yield option from the list below:						
	<table style="width: 100%; border: none;"> <tr> <td style="width: 30%;">GENPL</td> <td>General plasticity viscoplastic model using user subroutine UVSCPL</td> </tr> <tr> <td>RIGID</td> <td>Rigid-plastic material, no elasticity, vonMises yield</td> </tr> <tr> <td>IMCREEP</td> <td>Implicit creep model, both plasticity and creep with vonMises yield criteria</td> </tr> </table>	GENPL	General plasticity viscoplastic model using user subroutine UVSCPL	RIGID	Rigid-plastic material, no elasticity, vonMises yield	IMCREEP	Implicit creep model, both plasticity and creep with vonMises yield criteria
GENPL	General plasticity viscoplastic model using user subroutine UVSCPL						
RIGID	Rigid-plastic material, no elasticity, vonMises yield						
IMCREEP	Implicit creep model, both plasticity and creep with vonMises yield criteria						

Describer	Meaning
“Units”	A keyword specifying if data is read from a database and if so, the units to be used if data is read from a database. (Enter only if necessary).
4	Read from Marc database for flow stress (default units)
5	Use MATILDA database (default units)
6	Data read from input file in SI-mm units
7	Data read from input file in SI-m units
8	Data read from input file in US units
“VParam”	A keyword specifying that the number of viscoplastic parameters. Up to two viscoelastic parameters are to be entered (used when RYIELD=ViscPlas)
Nvp	Number of viscoplastic parameters. (Integer must be 0, 1, or 2)
Vp1-1	First Viscoplastic parameter for Nvp=1. (Real; Default = 0.0)
Vp2-1	Second viscoplastic parameter for Nvp=1. (Real; Default = 0.0)
Vp1-2	First Viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)
Vp2-2	Second viscoplastic parameter for Nvp=2. (Real; Default = 0.0, leave blank if Nvp=0 or Nvp=1)
“ImpCreep”	A keyword specifying that the equivalent (von Mises) tensile yield stress will be entered in the next field (used when RYIELD=ImpCreep).
VMISES	Equivalent (von Mises) tensile yield stress. Overrides any similar entry made for this material elsewhere. (Real, no Default)

Remarks:

1. Unless continuation entry is present specifying various material models, von Mises yield criterion is used as default.
2. If  $Y_0$  is not specified, FID field referring to a stress-plastic strain curve must be provided. If  $Y_0$  is specified without FID field, the material is assumed perfectly plastic or has one slope, H. If both  $Y_0$  and FID fields are specified, FID supersedes the  $Y_0$  field and the first data point in TABLES1 represents  $Y_0$ .

The initial yield point corresponds to the first data point in the function specified on the FID field.

$$Y(\overline{\varepsilon}^p) = Y_0(1 + b \overline{\varepsilon}^p)^n$$

where  $Y_0$  is an initial yield stress, b and n are parameters characterizing the stress-strain relationship.

In case of an anisotropic material, the initial yield point corresponds to the reference yield stress ( $Y_a$  in Remark 7.)

3. For SOL 600, the form of the stress-strain curve is determined by parameters MRTABLS1 and MRTABLS2.
4. The plastic deformation starts when the effective stress ( $\bar{\sigma}$ ) exceeds the yield stress.

The yield stress is initially defined by the initial yield point, which is subsequently modified by the hardening rule to account for strain hardening. Under the isotropic hardening rule, the size of the yield surface expands as a function of effective plastic strain ( $\bar{\epsilon}^p$ ). Under the kinematic hardening rule, the center of the yield surface moves in stress space while keeping the same size and shape. Ziegler's law is used to define the translation of the yield surface. Under the combined hardening, the initial hardening is assumed to be entirely isotropic, but the elastic range attains a constant value (i.e., behaving like kinematic hardening) after some plastic straining. The effective stress for von Mises is expressed as

with

$$\bar{\sigma} = \sqrt{\frac{1}{2}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_z - \sigma_x)^2] + 3(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{zx}^2)}$$

where the stress components are measured from the center of yield surface.

$$R_{11} = \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by r-values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\epsilon_w}{\epsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\alpha$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation, F, G, H and N can be determined by r-values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left( r_{45} + \frac{1}{2} \right) \left( 1 + \frac{r_0}{r_{90}} \right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

5. The Cowper and Symonds model scales the initial yield stress as a function of strain-rate, i.e.,

$$Y(\dot{\varepsilon}) = Y_0 \left[ 1 + \left( \frac{\dot{\varepsilon}}{C} \right)^{1/P} \right]$$

6. Hill's anisotropic model introduces orthotropic plastic material. This option can only be combined with orthotropic or anisotropic elastic material (i.e., with MAT2, MATORT or MAT9). The plastic anisotropy proposed by Hill introduces six parameters to the von Mises yield function, from which an effective stress may be derived as

$$\bar{\sigma} = \sqrt{F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2}$$

in which the material parameters can be related to the yield stress ratios by



$$F = \frac{1}{2} \left( \frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left( \frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left( \frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2R_{23}^2}, M = \frac{3}{2R_{31}^2}, N = \frac{3}{2R_{12}^2}$$

with

$$R_{11} = \frac{Y_1}{Y_a}, R_{22} = \frac{Y_2}{Y_a}, R_{33} = \frac{Y_3}{Y_a}$$

$$R_{12} = \frac{\sqrt{3}T_{12}}{Y_a}, R_{23} = \frac{\sqrt{3}T_{23}}{Y_a}, R_{31} = \frac{\sqrt{3}T_{31}}{Y_a}$$

where  $Y_1$ ,  $Y_2$ , and  $Y_3$  are the initial tensile yield stresses measured in material directions 1, 2 and 3, respectively;  $T_{12}$ ,  $T_{23}$ , and  $T_{31}$  are the shear yield stresses in pure shear; and  $Y_a$  is the reference yield stress which should be an average yield stress in all directions.

In practical applications, however, the initial yield stress cannot be measured in all directions. The plastic anisotropy is pronounced in the sheet metal due to prior rolling process, for which the plastic anisotropy is customarily characterized by  $r$ -values defined by strain ratio measured in the uniaxial tension, i.e.,

$$r_a = \frac{\epsilon_w}{\epsilon_t} = \frac{l_n(w_o/w)}{l_n(t_o/t)} = \frac{H + (2N - F - G - 4H)\sin^2\alpha \cos^2\alpha}{F \sin^2\alpha + G \cos^2\alpha}$$

where  $t$  and  $w$  denote thickness and width, respectively; and  $\alpha$  denotes the angle of orientation (usually measured from the rolling direction). Assuming that the anisotropy parameters stay constant throughout the deformation,  $F$ ,  $G$ ,  $H$  and  $N$  can be determined by  $r$ -values from tensile specimen cut at 0, 45 and 90 degrees to the rolling direction:

$$\frac{H}{G} = r_0 \quad , \quad \frac{H}{F} = r_{90}$$

$$\frac{N}{G} = \left( r_{45} + \frac{1}{2} \right) \left( 1 + \frac{r_0}{r_{90}} \right)$$

The orthotropic plasticity parameters should be calculated from the r-values and the initial yield stress either in 0 or 90 degree direction ( $Y_0$  or  $Y_{90}$ ) from the experiment. The yield stress in the thickness direction can be written as

$$Y_{th} = Y_0 \sqrt{\frac{r_{90}(1+r_0)}{r_0+r_{90}}} = Y_{90} \sqrt{\frac{r_0(1+r_{90})}{r_0+r_{90}}}$$

Similarly, yield stresses in shear may be evaluated by

$$T_{12} = Y_{th} \sqrt{\frac{1}{2r_{45}+1}}$$

and

$$T_{23} = T_{31} = \frac{Y_a}{\sqrt{3}}$$

in which the transverse direction is assumed isotropic.

7. The elasticity constants must be isotropic for ORNL plasticity except for normal ORNL model for stainless steel. The 10th cycle stress-plastic strain curve in TID field should be a normalized function so that the yield stress at zero plastic strain is unity.

The pressure-dependent yielding, based on Drucker-Prager yield criterion, contains three options for frictional materials such as rocks and concrete. The generalized Mohr-Coulomb criterion introduces linear and parabolic models, developed by Drucker and Prager. The linear Mohr-Coulomb model assumes a linear function of hydrostatic stress for a yield function, i.e.,

$$\alpha I_1 + \sqrt{J_2} - \frac{\bar{\sigma}}{\sqrt{3}} = 0 \quad \text{or} \quad \bar{\sigma} = \sqrt{3}\alpha I_1 + \sqrt{3J_2}$$

where

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

and

$$J_2 = \frac{1}{6}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]$$

The parameters  $\alpha$  and  $\bar{\sigma}$  (effective stress coinciding with the yield stress) can be related to material constants  $c$  (cohesion) and  $\phi$  (frictional angle) by

$$c = \frac{\bar{\sigma}}{3\sqrt{(1-12\alpha^2)}}$$

and

$$\sin \phi = \frac{3\alpha}{\sqrt{1 - 3\alpha^2}}$$

The parabolic Mohr-Coulomb model allows a yield envelope to be parabolic in the plane strain case, for which the yield function is expressed as

$$\sqrt{3J_2 + \sqrt{3}\beta\bar{\sigma}I_1} - \bar{\sigma} = 0$$

in which parameters are related to the material constants by

$$\bar{\sigma}^2 = 3\left(c^2 - \frac{\alpha^2}{3}\right)$$

and

$$\beta = \frac{\alpha}{\sqrt{3(3c^2 - \alpha^2)}}$$

8. The Drucker-Prager plasticity models can only be combined with isotropic elasticity. The Buyukozturk concrete plasticity model is a particular form of the generalized Drucker-Prager plasticity model, which is developed specifically for plane stress cases by Buyukozturk. The Buyukozturk yield function is expressed as

$$\beta\sqrt{3}Y I_1 + \gamma I_1^2 + 3J_2 - Y^2 = 0$$

where  $\beta$  is a user-specified constant,  $\gamma$  is an internal parameter (set to 0.2) with no user's access, and  $Y$  is the yield stress.

The Buyukozturk concrete plasticity model is coupled with crack and crush capability, which is designed for a low-tension material. The low-tension material develops a crack in the perpendicular direction to the maximum principal stress when it exceeds a critical value. The tension softening modulus can be specified (in absolute value) by the user to process the cracking process gradually. The default value (0.) is intended for a sudden cracking with a complete loss of the stiffness upon cracking. After the initial crack, a second crack can initiate in the perpendicular direction to the first crack. Likewise, the third crack can be formed in 3D solid elements. The loading may reverse the direction after the crack is formed. In this case, the crack will close and some load carrying capacity is resumed. The compression capability is fully resumed and the shear stresses may be transmitted over the crack surface with a reduced stiffness by a factor specified as shear retention factor. The material may fail in compression by crushing. The input value for the crush strain is positive, which implies an absolute value of a compressive strain. The material loses its integrity for good upon crushing. The reinforcement bars may be simulated by adding REBAR elements.

9. The Gurson model for porous metal plasticity may be used only with isotropic hardening rule. All other hardening rules will be ignored if Gurson model is selected. The Gurson model modified by Tvergaard and Needleman is designed for porous metal plasticity with damage effects in the ductile material. The material is assumed to form voids under loading, which grow, coalesce, then leads to crack formation and eventually failure. This process is a function of hydrostatic stress and the void volume fraction  $f_v$ . The yield function is established as follows:

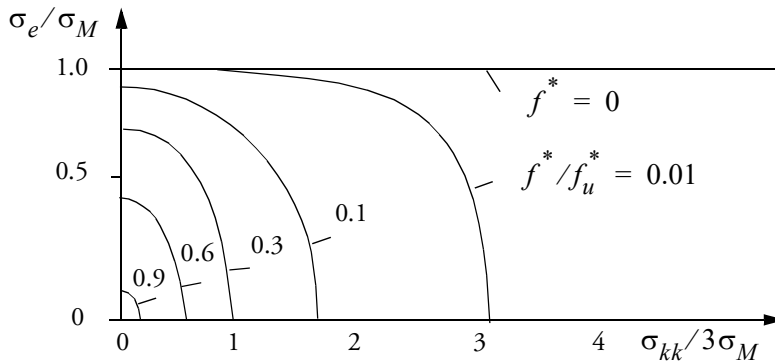
$$\frac{3J_2}{Y^2} + 2q_1 f_v^* \cosh\left(\frac{q_2 I_1}{2Y}\right) - [1 - (q_1 f_v^*)^2] = 0$$

in which  $Y$  denotes a yield stress of the fully dense matrix material,  $I_1$  denotes the first invariant of stresses, and the modified void volume fraction  $f_v^*$  is computed by

$$f_v^* = f_v \quad \text{if } f_v \leq f_v^c$$

$$f_v^* = f_v^c + \left( \frac{f_v^u - f_v^c}{f_v^f - f_v^c} \right) (f_v - f_v^c) \quad \text{if } f_v > f_v^c$$

where  $f_v^c$  is the critical void volume fraction,  $f_v^f$  is the void volume at failure and  $f_v^u = 1/q_1$ . The solid loses all stress carrying capability when the void volume fraction reaches  $f_v^f$ .



The evolution of damage as measured by void volume fraction is due to void nucleation and void growth. Void nucleation occurs by debonding of the second phase particles. The strain for nucleation depends on the particle sizes. Assuming a normal distribution of particle sizes, the void nucleation itself is modeled as a normal distribution in strains if nucleation is strain-controlled. If the void nucleation is assumed to be stress controlled in the matrix, a normal distribution is assumed in stresses. The void volume fraction changes due to the growth of existing voids and nucleation of new voids, i.e.,

$$\dot{f}_v = \dot{f}_{growth} + \dot{f}_{nucleation}$$

in which the void growth can be determined based on the compressibility of the material

$$\dot{f}_{growth} = (1 - f_v) \dot{\epsilon}_{kk}^p$$

and the nucleation can be defined either as strain or stress-controlled with a normal distribution about the mean value. In case of strain-controlled nucleation, the rate is expressed as

$$\dot{f}_{nucleation} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[ -\frac{1}{2} \left( \frac{\bar{\epsilon}_m^p - \epsilon_n}{S} \right)^2 \right] \dot{\epsilon}_m^p$$

where  $f_v^n$  is the volume fraction of void forming particles,  $\bar{\epsilon}_m^p$  denotes the effective plastic strain in the matrix material, and the void nucleation strain is assumed normally distributed with a mean value of  $\epsilon_n$  and a standard deviation of  $S$ . In case of stress-controlled nucleation, the rate is expressed as

$$\dot{f}_{nucleation} = \frac{f_v^n}{S \sqrt{2\pi}} \text{Exp} \left[ -\frac{1}{2} \left( \frac{\bar{\sigma} + \frac{1}{3} \sigma_{kk} - \sigma_n}{S} \right)^2 \right] \left( \dot{\sigma} + \frac{1}{3} \dot{\sigma}_{kk} \right)$$

If the size of the second phase particles are widely dispersed, the standard deviation would be larger than more uniform cases. A typical values for an engineering alloy as suggested by numerical experiments are set as default values for  $\epsilon_n$ ,  $S$ , and  $f_v^n$ .

10. The keywords may appear in any order. However, aniso, ORNL, press, and Gurson are mutually exclusive, and cannot coexist.
11. All the alphanumeric fields are recognizable by the first four letters.
12. For SOL 600, METHOD (first line, field 8) corresponds to Marc's PLASTICITY options as follows. METHOD must be the same for all MATEP entries in the SOL 600 input but this is not enforced. The value of METHOD on MATEP with the smallest MID governs who the PLASTICITY option is set. If METHOD is left blank, it may alternatively be set by PARAM,MARCPLAS.

Name	Marc PLASTICITY parameter value
Addmean	3
Addrad	4
Multrad	5
Multpln	6
Smlmean	1
Smlrad	2

13. The Bulk Data fields denoted by N/A are different from blank fields. Those fields with N/A are not used currently, but the space is reserved in case of future additions. On the other hand, the blank fields which have no specifications are neither used nor reserved (any memory space) for future use. The Method field determines the options under Marc parameter PLASTICITY.

The initial yield stress should be extracted from the table provided in the FID field. The yield stress-plastic strain function specified in TABLES1 under FID field is not a normalized function.

The anisotropic material parameters  $R_{ij}$  are equivalent to Marc input data as follows:

R11 = YRDIR1  
 R22 = YRDIR2  
 R33 = YRDIR3  
 R12 = YRSHR1  
 R23 = YRSHR2  
 R31 = YRSHR3

The crack/crush capability in Marc may be combined with other isotropic material options.

14. The strain effect on the yield stress (Reffect) is specified under the model definition option STRAIN RATE in Marc.
15. This Bulk Data entry accommodates Marc's input data under the model definition options ISOTROPIC, ORTHOTROPIC, ANISOTROPIC, RATE EFFECTS, WORK HARD, DAMAGE and CRACK DATA as well as the parameter PLASTICITY.
16. The fields without default values can be left blank only if they are specifically permitted to be blank.
17. RYIELD equates to Marc's ISOTROPIC (3,2) field.
18. WKHARD equates to Marc's ISOTROPIC (3,3) field.
19. RYIELD = LINMOHR, PBLMOHR and WKHARD = CHABOCHE, PWRLAW, RPWRLAW, JHCOOK, and KUMAR are only applicable to isotropic materials (MAT1).

MATEP

Elasto-Plastic Material - SOL 700

Elasto-plastic material properties for SOL 700 only.

Format:

Note that the primary entry is required. All other continuation lines are required only for certain options and only one such option may be entered.

1	2	3	4	5	6	7	8	9	10
MATEP	MID	FORM	Y0	FID	RYIELD		TYPE	H	
	"Reffect"	OPTION1	RTID	C	P				
	"JhCook"	A	B	N	C	M	Tmelt	Troom	
		$\epsilon_0$ DOT	CP						
	"PwrLaw"	A	M	B	N	C			
	"Poly"	A	B	C	D	E	F	YMAX	
	"TMmodel"	A	B	C	D	M	$\epsilon_{SM}$	Tmelt	
		Troom	YCR	E	K	$\epsilon_0$ DOT	CP		
	"ZAmodel"	A	B	N	C	M	$\epsilon_0$ DOT	D	
		CP							
	"OrthoCr"	E	NU	RELV	TYPE	TIDXX	TIDYY	TIDZZ	
		TIDXY	TIDYZ	TIDZX	TIDSR				
	"Soil"	TABLE	TYPE	VALUE	CUTOFF	A0	A1	A2	
		YIELD	YSTYP						

Example:

MATEP	7	SLOPE	1.2E6		VMISES			1.0E3	
	"Reffect"	COWPER		2.0	4.0				

Describer	Meaning
MID	Identification number of MAT1 or MATORT entry. MATORT is only acceptable for RYIELD=ORTHOOCR and MAT1 is acceptable for all other yield criteria except ORTHOOCR. (Integer > 0; required.)
FORM	Selects a form of stress-plastic strain function to be specified for RYIELD=VMISES. (Character; default=SLOPE):
	SLOPE Defines non-zero H.
	TABLE Defines the function using TABLED1 entry. Units of table are unspecified.
	PERFECT Defines perfectly plastic material (H is zero or blank and FID is blank).

Describer	Meaning
Y0	Initial yield stress for RYIELD=VMISES and ORTHOCR. (Real > 0.0; default=blank)
FID	Identification number of TABLED1 for RYIELD=VMISES. (Integer > 0; default=blank)
RYIELD	Enter on of the following yield criteria rules. (Character; Default = VMISES)
VMISES	von Mises yield criteria. It supports “Reffect” option. See Remarks 1.-8.
JHCOOK	For Johnson-Cook formulation. It supports “JhCook” option. See Remarks 9. and 10.
RPWRLAW	Rate power law. It supports “PwrLaw” option. See Remark 11.
POLY	Polynomial Yield Model. It supports “Poly” option. See Remark 12.
TMMODEL	Tanimura-Mimura Yield Model. It supports “TMmodel” option. See Remark 13.
ZAMODEL	Zerilli-Armstrong Yield Model. It supports “ZAmodeL” option. See Remarks 14.
ORTHOCR	Orthotropic Crushable Model. It supports “OrthoCr” option. It is only available for solid elements with MATORT. See Remarks 15.-18.
SOIL	Elastoplastic Crushable Model. It supports “Soil” option. It is only available for solid elements. See Remarks 19.-22.
TYPE	The type of stress and strain defined in FID. (Character; default=TRUE)
ENG	Engineering stress and strain.
TRUE	True stress and strain.
PLAST	True stress and plastic strain.
PMOD	Plastic modulus and true stress.
H	Plasticity modulus. (Real > 0; Default = 0).
“Reffect”	A keyword signifying that the following data pertains to the rate-dependent material properties (do not enter rate-dependent effects if they are not to be included). If TABLED1 is entered and one of the independent variables is strain rate, this field and its options should not be specified.
OPTION1	Selects an option for strain-rate dependent yield stress (Character; default=TABLE):
TABLE	Specifies that RTID input used to define rate dependency.
COWPER	Specifies the Cowper and Symonds rate-dependency model.
RTID	TABLED1 ID for strain-rate effects on yield stress, i.e., the table defines yield stress as a function of strain-rate starting from zero strain-rate. The yield stresses in this table should comprise the initial yield stress specified on Y0 or FID field at zero strain-rate.



Describer	Meaning
C	Specifies the constant C for Cowper and Symonds model. (Real; Default = 1.0)
P	Specifies the constant P for Cowper and Symonds model. (Real; Default = 1.0)
“Jhcook”	Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature.
A	Static yield stress. (Real > 0.0; required.)
B	Hardening parameter. (Real; default=0.0)
N	Hardening exponent. (Real; default=1.0)
C	Strain-rate parameter. (Real; default=0.0)
M	Temperature exponent. (Real; default=1.0)
TMELT	Melt temperature. (Real; default=1.E20)
TROOM	Room temperature. (Real; default=293.0)
ε0DOT	Reference strain rate. (Real > 0.0; default=1.0)
CP	Specific heat. (Real; default=1.E20)
“PwrLaw”	Rate power law
A	Hardening parameter. (Real; default=0.0)
M	Hardening exponent. (Real; default=1.0)
B	Initial yield parameter. (Real > 0.0; required)
N	Strain rate exponent. (Real; default=1.0)
C	Minimum yield stress. (Real; default=1.E20)
“Poly”	Polynomial Yield Model
A	Initial yield parameter. (Real > 0.0; required)
B	Coefficient B. (Real; default=0.0)
C	Coefficient C. (Real; default=0.0)
D	Coefficient D. (Real; default=0.0)
E	Coefficient E. (Real; default=0.0)
F	Coefficient F. (Real; default=0.0)
YMAX	Maximum yield stress. (Real; default=1.E20)
“SGmodel”	Steinberg-Guinan Yield Model
A1-A4	Yield parameters. (Real > 0.0; required.)
H, B	Yield parameters. (Real; default=0.0)
TMELT	Melt temperature. (Real; default=1.E20)
TROOM	Room temperature. (Real; default=293.0)
CP	Specific heat. (Real; default=1.E20)

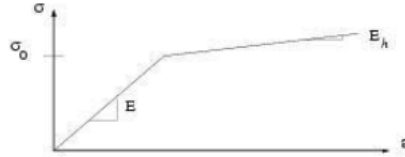


Describer	Meaning
“TMmodel”	Tanimura-Mimura Yield Model
A	Static yield parameter. (Real > 0.0; Required)
B	Hardening parameter. (Real; default=0.0)
C	Strain rate parameter. (Real; default=0.0)
D	Strain rate parameter. (Real; default=0.0)
M	Temperature exponent. (Real; default=1.0)
$\epsilon SM$	Quasi-static strain rate. (Real > 0.0; default=1.0)
TMELT	Melt temperature. (Real; default=1.E20)
TROOM	Room temperature. (Real; default=293.0)
YCR	Critical yield stress. (Real > 0.0; default=1.E20)
E	Strain rate parameter. (Real; default=0.0)
K	Strain rate exponent. (Real; default=1.0)
$\epsilon 0DOT$	Reference strain rate. (Real > 0.0; default=1.0)
CP	Specific heat. (Real; default=1.E20)
“ZAModel”	Zerilli-Armstrong Yield Model.
A	Static yield parameter. (Real > 0.0; required)
B	Hardening parameter. (Real > 0 0; default=0.0)
N	Hardening exponent. (Real > 0.0; default=1.0)
C	Strain rate parameter. (Real; default=0.0)
M	Temperature exponent. (Real; default=1.0)
$\epsilon 0DOT$	Reference strain rate. (Real > 0.0; default=1.0)
D	Bcc parameter. (Real; default=0.0)
	= 0            Fcc metal
	≠ 0           Bcc metal
CP	Specific heat. (Real; default=1.E20)
“OrthoCr”	Orthotropic Crushable Model.
E	Young’s modulus for the fully compacted material. (Real > 0.0; required)
NU	Poisson’s ratio for the fully compacted material. (-1.0 < Real < 0.5; required)
RELV	Relative volume at which the material is fully compacted. (0.0 < Real < 1.0; required)
TYPE	The type of data defined as the x-value in TIDXX~TIDSR. (Character; default=CRUSH)
	CRUSH                    Crush factor (1-relative volume)

Describer	Meaning
	RELVOL                      Relative volume $V/V_0$
TIDXX	TABLED1 ID defining the variation of the (local) xx-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDYY	TABLED1 ID defining the variation of the (local) yy-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDZZ	TABLED1 ID defining the variation of the (local) zz-stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDXY	TABLED1 ID defining the variation of the (local) xy-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDYZ	TABLED1 ID defining the variation of the (local) yz-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDZX	TABLED1 ID defining the variation of the (local) zx-shear stress (y-value) with relative volume or crush (x-value). (Integer > 0; required)
TIDSR	TABLED1 ID defining the variation of a yield factor (y-value) with the deviatoric strain rate (x-value). (Integer > 0; default=not used)
“Soil”	Elastoplastic Crushable Model.
TABLE	TABLED1 ID defining the variation of pressure (y-value) with crush factor or volumetric strain (x- value). (Integer > 0; required)
TYPE	The type of data defined as the x-value in the table. (Real > 0.0; default=1.0)
	1.0                      Crush factor. (= 1-relative volume)
	2.0                      Volumetric (true) strain
VALUE	The value for cut-off stress. (Real < 0.0; See Remark 20.)
CUTOFF	Cut-off stress. See Remark 21. (Real > 0.0; default=2.0)
	1.0                      Pressure for total tensile failure
	2.0                      Pressure for tensile failure
	3.0                      Minimum pressure
A0, A1, A2	Yield function constants. (Real; default=0.0)
YIELD	Surface description. See Remark 22. (Real > 0.0; default=1.0)
	1.0                      The yield surface is defined as a function of p and $J_2$ .
	2.0                      The yield surface is defined as a function of p and $s_y$ .
YSTYP	Type of YSURF Yield Surface description. See Remark 22. (Real > 0.0; default=1.0)
	1.0                      DYNA definition
	2.0                      Dytran additional definition

Remarks:

1. A bilinear stress-strain characteristic is used by specifying FORM=SLOPE, Y0 and H:



where the equivalent stress is given by 
$$\sigma = \sigma_0 + \frac{E \cdot E_h}{E - E_h} \epsilon_p$$

where

$\sigma_0$	= yield stress specified in the Y0 field
$E$	= Young's modulus
$E_h$	= hardening modulus specified in the H field
$\epsilon_p$	= equivalent plastic strain
$\sigma$	= equivalent stress

2. A piecewise linear, stress-strain characteristic is used by specifying FID and TYPE (beams and shells only)

$$\sigma = [(\sigma_i - \sigma_{i-1})(\epsilon - \epsilon_{i-1}) / (\epsilon_i - \epsilon_{i-1})] + \sigma_{i-1}$$

The stress-strain characteristic used internally in Nastran SOL700 is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

True stress/true strain	(TYPE = TRUE)
Engineering stress/engineering strain	(TYPE = ENG)
True stress/plastic strain	(TYPE = PLAST)
Plastic modulus/true stress	(TYPE = PMOD)

3. With solid elements, only an elastic perfectly plastic yield model is currently used. Only the Y0 field is used.
4. If FID is blank or zero, a bilinear stress-strain curve is assumed. If FID has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If FID is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If RTID is blank or zero and C and P are blank or zero, the yield stress does not vary with strain rate. If RTID has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. (C and P must be blank or zero.)

If RTID is blank or zero and C and P are defined, the enhancement of the yield stress with strain rate

is calculated as 
$$\frac{\sigma_d}{\sigma_y} = 1 + \left(\frac{\dot{\epsilon}_p}{C}\right)^{1/p}$$

where  $\sigma_d$  is the dynamic stress,  $\sigma_y$  is the static yield stress, and  $\dot{\epsilon}_p$  is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined, it overrides the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see FID and RTID fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. The Johnson Cook yield stress is computed from

$$\sigma_y = (A + B\epsilon_p^N) \left[ 1 + C \ln\left(\frac{\dot{\epsilon}}{\dot{\epsilon}_0}\right) \right] (1 - T^{*M})$$

where

$\epsilon_p$	= effective plastic strain
--------------	----------------------------

$$T^* = \frac{T - TROOM}{TMELT - TROOM}$$

$\dot{\epsilon}$   
 $\dot{\epsilon}_0$  = reference strain rate

$\epsilon_0$  = reference strain rate

T = temperature

TROOM = room temperature

TMELT = melt temperature

A, B, N, C, and M are input constants.

10. The reference strain rate for Johnson Cook yield model is per unit time.
11. The rate power law yield stress is computed from

$$\sigma_y = MAX(C, B + A\epsilon_p^{M \cdot N} \dot{\epsilon})$$

where

$\epsilon_p$	= effective plastic strain
--------------	----------------------------

$\dot{\epsilon}$  = reference strain rate

A, B, M, N and C are constants.

12. The polynomial yield stress is computed from

$$\sigma_y = \text{MIN}(YMAX, A + B\varepsilon_p + C\varepsilon_p^2 + D\varepsilon_p^3 + E\varepsilon_p^4 + F\varepsilon_p^5)$$

where

$\varepsilon_p$	= effective plastic strain
-----------------	----------------------------

YMAX = maximum yield stress

A, B, C, D, E and F are constants.

13. The Tanimura-Mimura yield stress is computed from

$$\sigma_y = \left[ A + B\varepsilon_p + (C + D\varepsilon_p) \left( 1 - \frac{A + B\varepsilon_p}{\sigma_{cr}} \right) \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_s} \right) \right] (1 - T^{*M}) + E \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right)^K$$

where

$\varepsilon_p$	= effective plastic strain
-----------------	----------------------------

$\sigma_{cr}$  = critical yield stress

$\dot{\varepsilon}$  = effective strain rate

$\dot{\varepsilon}_s$  = quasi-static strain rate

$\dot{\varepsilon}_0$  = reference strain rate

$$T^* = \frac{T - TROOM}{TMELT - TROOM}$$

T = temperature

TROOM = room temperature

TMELT = melt temperature

A, B, D, M, E and K are constants

14. The Zerilli-Armstrong yield stress is computed from

$$\sigma_y = \begin{cases} (A + B\varepsilon_p^N)e^{\left[-MT + CT \ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} & \text{For FCC Metals} \\ (A + B\varepsilon_p^N) + De^{\left[-MT + CT \ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right]} & \text{For BCC Metals} \end{cases}$$

where

$\varepsilon_p$	= effective plastic strain
$\dot{\varepsilon}$	= effective strain rate
$\dot{\varepsilon}_0$	= reference strain rate

$T$  = temperature

A, B, N,C,M and D are constants.

- If the initial Poisson's ratios in MATORT are not supplied, the default is set to zero. Therefore, the behavior of the material during compaction is uncoupled. This means that straining in the (local) x-direction produces stresses only in the (local) x-direction, and not in the (local) y- or z-direction. The tables define the variation of the stress in a particular direction with the relative volume or the crush. The relative volume is defined as (current volume)/(initial volume) and varies from 1.0 (uncompressed) to 0.0 (zero volume). Crush is defined as one minus the relative volume and varies from 0.0 to 1.0. Since the tables should be defined with increasing x-values, it is convenient to use the default value for type, which is CRUSH. When defining the curves, care should be taken that the extrapolated values do not lead to negative yield stresses.
- The elastic moduli (and the initial Poisson's ratios only if they are supplied) in MATORT vary linearly with the relative volume from their initial uncompact values to the fully compacted ones.
- When the material is fully compacted, its behavior becomes isotropic with an elastic perfectly plastic material characteristic.
- If the TIDSR option is used, you can supply a table including strain-rate effects. Strain rate is defined here as the Euclidean norm of the deviatoric strain-rate tensor; i.e.,  $\dot{\varepsilon} = \sqrt{\varepsilon_{ij}^{dev} \cdot \varepsilon_{ij}^{dev}}$   
 The y values in this table are factors with which the stresses in the other tables are multiplied to incorporate strain-rate effects.

19. The pressure-volume characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain and is defined as  $(1 - \frac{V}{V_0})$ , with  $\frac{V}{V_0}$  as the relative volume; or in terms of the volumetric (true) strain which is defined as:  $\int_{t_0}^t \frac{dV}{V}$  or  $\ln\left(\frac{V}{V_0}\right)$

The crush factor must be between 0 and 1. The volumetric strain must always be negative.

20. If the field for the value of CUTOFF is left blank, then this value is calculated from the yield function defined by the constants A0, A1, and A2. In case of a Mohr-Coulomb yield model, the cut-off pressure is calculated as the root of the pressure-yield stress curve. If the YIELD=1.0 is used, the cut-off pressure is calculated as the intersection point of the yield surface with the hydrostat (if only A0 is nonzero, then the cut-off pressure is set to -100K, where K is the bulk modulus). The cut-off pressure must be negative.
21. Either a minimum pressure (CUTOFF=3.0) or a failure pressure (CUTOFF=1.0 or 2.0) can be specified. In the first case, this corresponds to a tensile cutoff, where the pressure cannot fall below the minimum value. In the second case, if the pressure falls below the failure pressure the element fails and cannot carry tensile loading for the remainder of the analysis. Thus, the pressure can never become negative again. If CUTOFF=1.0 is used, the elements can physically fail, which means that the stresses are set to zero, but also the failure flag is used as in normal MATF models. If CUTOFF=2.0 is used, only the stresses are set to zero.
22. If YIELD=1.0 is used, the yield stress is determined by a Mohr-Coulomb model:  $\sigma_y = \text{MIN}(A_0 + A_1 p, A_2)$

If YIELD=2.0 is used, the yield surface in three-dimensional space is defined by  $\phi_s=0$  here

$$\Phi_s = \frac{1}{2} S_{ij} S_{ij} - (B_0 + B_1 p + B_2 p^2) = J_2 - (B_0 + B_1 p + B_2 p^2)$$

where  $s_{ij}$  are the deviatoric stresses and  $J_2$  is the second invariant of the stress deviation. The coefficients  $B_0$ ,  $B_1$  and  $B_2$  can be related to the coefficients  $A_0$ ,  $A_1$  and  $A_2$ . The relation between the coefficients depends on the YSTYP field as shown below.

If YSTYP=2.0, then

$$B_0 = A_0$$

$$B_1 = A_1$$

$$B_2 = A_2$$

Thus, the yield stress is defined as  $\sigma_y = \sqrt{3(A_0 + A_1 p + A_2 p^2)}$



$$B_0 = \frac{1}{3}A_0^2$$

If  $Y_{STYP}=1.0$ , then  $B_1 = \frac{2}{3}A_0A_1$

$$B_2 = \frac{1}{3}A_1^2$$

and  $A_2$  is ignored.

Thus, the yield stress is defined as  $\sigma_y = A_0 + A_1 p$

**MATF**

**Material Failure Model**

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis. Up to three criteria may be specified for each material. For progressive failure (ITYPE=2) only the first criterion will contribute to failure. The other two, if specified, will only provide output of failure index. SOL 400 (only with extended property entries), and all linear solution sequences between SOL101 and SOL112 (only with CPYRAM or CHEXA/PCOMPLS) are supported.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	ITYPE							
	"CRI"	Criterion	$V_1^1$	$V_2^1$	$V_3^1$	$V_4^1$	$V_5^1$	$V_6^1$	1st
	$V_7^1$	$V_8^1$	$V_9^1$	Find <sup>1</sup>	$V_{10}^1$	$V_{11}^1$	$V_{12}^1$	$W_1^1$	
	$W_2^1$	$W_3^1$	$W_4^1$	$W_5^1$	$W_6^1$	$W_7^1$	$W_8^1$	$W_9^1$	
	"PF"	A1	A2	A3	A4	A5			
	"CRI"	Criterion	$V_1^2$	$V_2^2$	$V_3^2$	$V_4^2$	$V_5^2$	$V_6^2$	2nd
	$V_7^2$	$V_8^2$	$V_9^2$	Find <sup>2</sup>	$V_{10}^2$	$V_{11}^2$	$V_{12}^2$	$W_1^2$	
	$W_2^2$	$W_3^2$	$W_4^2$	$W_5^2$	$W_6^2$	$W_7^2$	$W_8^2$	$W_9^2$	
	"CRI"	Criterion	$V_1^3$	$V_2^3$	$V_3^3$	$V_4^3$	$V_5^3$	$V_6^3$	3rd
	$V_7^3$	$V_8^3$	$V_9^3$	Find <sup>3</sup>	$V_{10}^3$	$V_{11}^3$	$V_{12}^3$	$W_1^3$	
	$W_2^3$	$W_3^3$	$W_4^3$	$W_5^3$	$W_6^3$	$W_7^3$	$W_8^3$	$W_9^3$	

Example 1 (3 Criteria, no progressive failure):

MATF	100	0							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
		4500.	4500.	4500.					
+									
	CRI	7	2500.	3000.	1500.	2000.			2nd

+	1000.	1000.							
	.11	.06	.1	.05	.075	.03	.03	.03	
	CRI	4	2500.	4000.	2500.	4000.	2000.	3000.	3rd
	4500.	4500.	4500.						

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criteria has been omitted.)

Describer	Meaning
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default)
ITYPE	Flag to invoke progressive failure. (Integer; Default = 0) 0 No progressive failure, compute failure indices only. 2 Gradual selective stiffness degradation. 3 Immediate selective stiffness degradation.
“CRI”	Enter the character string “CRI” to start input data for a failure criterion.
CRI67	Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows: $1000000*ITYPE+10000*C3+100*C2+C1$ where C1, C2, C3 are integer values for the various Criteria listed.
Criterion	Select an integer corresponding to the failure criterion to be applied. (Integer; no Default) Up to three failure criteria may be specified for each MID. 1 For maximum stress criterion. (See Remark 3.) 2 For maximum strain criterion. (See Remark 4.) 3 For Hill failure criterion. (See Remark 5.) 4 For Hoffman failure criterion. (See Remark 6.) 5 Tsai-Wu failure criterion. (See Remark 7.) 7 Hashin failure criterion. (See Remark 8.) 8 Puck failure criterion. (See Remark 9.) 10 Hashin-Tape (See Remark ) 11 Hashin-Fabric (See Remark ) 13 User-defined criterion with user subroutine UFAIL See Remark 11.
$V_i^j$	See the respective remark for the meaning of material data. Stress limit $i$ for criterion $j$ . See Remarks for meaning and default.
$W_i^j$	Strain limit $i$ for criterion $j$ . See Remarks for meaning and default.

Describer	Meaning
Find <sup>j</sup>	Failure index scale factor for criterion j. Used for criteria 3, 4, and 5. (Real > 0.0, or blank, Default = 1.0)
"PF"	Enter the character string "PF" to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line may be omitted. Only the first criterion uses the PF line. (Character; no Default)
A1	Residual stiffness fraction. For ITYPE=2, the stiffness is not reduced beyond this fraction of the initial stiffness. For ITYPE=3, this is the fraction of initial stiffness upon failure. (Real > 0.0, Default = 0.01)
A2	<p>Leave blank except for Criterion values listed below.</p> <p>Criterion=7 (Hashin) A2 is the factor for E2 reduction due to matrix compression failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix compression failure.</p> <p>Criterion=8 (Puck) - Same as for Criterion=7</p> <p>Criterion=10 (Hashin Tape) - Same as for Criterion=7</p>
A3	<p>Leave blank except for the Criterion values listed below.</p> <p>Criterion=7 (Hashin) A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction.</p> <p>Criterion=8 (Puck) - Same as for Criterion=7</p> <p>Criterion=10 (Hashin Tape) - Same as for Criterion=7</p>
A4	<p>Leave blank except for the Criterion values listed below</p> <p>Criterion=7 (Hashin) A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criterion=8 (Puck) - Same as for Criterion=7</p> <p>Criterion=10 (Hashin Tape)- Same as for Criterion=7.</p>
A5	<p>Leave blank except for the Criterion values listed below</p> <p>Criterion=7 (Hashin) A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0, It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criterion=8 (Puck) - Same as for Criterion=7</p> <p>Criterion=10 (Hashin Tape) - Same as for Criterion=7</p>

Remarks:

1. The MATF Bulk Data entry contains supplementary data for failure index calculation and failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate a progressive failure process.
2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied; that is, the calculated failure index exceeds 1.0. Upon failure, the elastic moduli are reduced. This is done differently depending on failure criterion. The behavior up to the failure point is linear elastic. After failure the behavior is still elastic but with a different stiffness. The option should not be combined with other material nonlinearities like plasticity.
3. According to the Maximum Stress Criterion, six failure indices  $F_1$  are calculated as follows:

$$F_1 = \begin{cases} \frac{\sigma_x}{V_1} & \text{for } \sigma_x > 0 \\ \frac{-\sigma_x}{V_2} & \text{for } \sigma_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\sigma_y}{V_3} & \text{for } \sigma_y > 0 \\ \frac{-\sigma_y}{V_4} & \text{for } \sigma_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\sigma_z}{V_5} & \text{for } \sigma_z > 0 \\ \frac{-\sigma_z}{V_6} & \text{for } \sigma_z < 0 \end{cases}$$

$$F_4 = \frac{\tau_{xy}}{V_7}$$

$$F_5 = \frac{\tau_{yz}}{V_8}$$

$$F_6 = \frac{\tau_{zx}}{V_9}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

$V_2$  : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default  $V_1$

$V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

$V_4$  : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default  $V_3$

$V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

$V_6$  : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default  $V_5$

$V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default

$V_8$  : maximum allowable yz shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable zx shear stress. Real > 0.0 or blank; Default  $V_7$

4. According to the Maximum Strain Criterion, 6 failure indices  $F_i$  are calculated as follows:

$$F_1 = \begin{cases} \frac{\varepsilon_x}{W_1} & \text{for } \varepsilon_x > 0 \\ \frac{-\varepsilon_x}{W_2} & \text{for } \varepsilon_x < 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{\varepsilon_y}{W_3} & \text{for } \varepsilon_y > 0 \\ \frac{-\varepsilon_y}{W_4} & \text{for } \varepsilon_y < 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{\varepsilon_z}{W_5} & \text{for } \varepsilon_z > 0 \\ \frac{-\varepsilon_z}{W_6} & \text{for } \varepsilon_z < 0 \end{cases}$$

$$F_4 = \frac{\varepsilon_{xy}}{W_7}$$

$$F_5 = \frac{\varepsilon_{yz}}{W_8}$$

$$F_6 = \frac{\varepsilon_{zx}}{W_9}$$

where the indices (x, y, z) denote material coordinate direction.

$W_1$  : maximum allowable strain in the x-direction in tension. Real > 0.0, no Default.

$W_2$  : maximum allowable strain in the x-direction in compression. Real > 0.0 or blank; Default  $W_1$

$W_3$  : maximum allowable strain in the y-direction in tension. Real > 0.0, no Default.

$W_4$  : maximum allowable strain in the y-direction in compression. Real > 0.0 or blank; Default  $W_3$

$W_5$  : maximum allowable strain in the z-direction in tension. Real > 0.0; no Default

$W_6$  : maximum allowable strain in the z-direction in compression. Real > 0.0 or blank; Default  $W_5$

$W_7$  : maximum allowable shear strain in the xy plane. Real > 0.0 no Default

$W_8$  : maximum allowable shear strain in the yz plane. Real > 0.0 or blank; Default  $W_7$

$W_9$  : maximum allowable shear strain in the zx plane. Real > 0.0 or blank; Default  $W_7$

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. A single failure index is calculated as:

$$F = \left[ \frac{\sigma_x^2}{V_1^2} + \frac{\sigma_y^2}{V_3^2} + \frac{\sigma_z^2}{V_5^2} - \left( \frac{1}{V_1^2} + \frac{1}{V_3^2} - \frac{1}{V_5^2} \right) \sigma_x \sigma_y - \left( \frac{1}{V_2^2} + \frac{1}{V_5^2} - \frac{1}{V_3^2} \right) \sigma_x \sigma_z - \left( \frac{1}{V_3^2} + \frac{1}{V_5^2} - \frac{1}{V_1^2} \right) \sigma_y \sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} \right] / \text{Find}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.

$V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.

$V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default

$V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default

$V_8$  : maximum allowable yz shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable zx shear stress. Real > 0.0 or blank; Default  $V_7$

6. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion. A single failure index F is calculated as:

$$F = \left( C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left( \frac{1}{V_1} - \frac{1}{V_2} \right) \sigma_x + \left( \frac{1}{V_3} - \frac{1}{V_4} \right) \sigma_y + \left( \frac{1}{V_5} - \frac{1}{V_6} \right) \sigma_z + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} \right) / \text{Find}$$

with

$$C_x = \frac{1}{2} \left( \frac{1}{V_1 V_2} + \frac{1}{V_3 V_4} - \frac{1}{V_5 V_6} \right)$$

$$C_y = \frac{1}{2} \left( \frac{1}{V_3 V_4} + \frac{1}{V_5 V_6} - \frac{1}{V_1 V_2} \right)$$

$$C_z = \frac{1}{2} \left( \frac{1}{V_5 V_6} + \frac{1}{V_1 V_2} - \frac{1}{V_3 V_4} \right)$$



where the indices (x, y, z) denote material coordinate direction.

- $V_1$  : maximum allowable stress in the x-direction in tension (Real > 0.0, no Default).
- $V_2$  : maximum allowable stress in the x-direction in compression (Real > 0.0 or blank; Default  $V_1$  )
- $V_3$  : maximum allowable stress in the y-direction in tension (Real > 0.0, no Default).
- $V_4$  : maximum allowable stress in the y-direction in compression (Real > 0.0 or blank; Default  $V_3$  )
- $V_5$  : maximum allowable stress in the z-direction in tension (Real > 0.0, no Default).
- $V_6$  : maximum allowable stress in the z-direction in compression (Real > 0.0 or blank; Default  $V_5$  )
- $V_7$  : maximum allowable in-plane shear stress (Real > 0.0, no Default).
- $V_8$  : maximum allowable yz shear stress (Real > 0.0, no Default  $V_7$ ).
- $V_9$  : maximum allowable zx shear stress (Real > 0.0, no Default  $V_7$ ).

7. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion. A single failure index F is calculated as:

$$F = \left[ \left( \frac{1}{V_1} - \frac{1}{V_2} \right) \sigma_x + \left( \frac{1}{V_3} - \frac{1}{V_4} \right) \sigma_y + \left( \frac{1}{V_5} - \frac{1}{V_6} \right) \sigma_z + \frac{\sigma_x^2}{V_1 V_2} + \frac{\sigma_y^2}{V_3 V_4} + \frac{\sigma_z^2}{V_5 V_6} + \frac{\tau_{xy}^2}{V_7^2} + \frac{\tau_{yz}^2}{V_8^2} + \frac{\tau_{zx}^2}{V_9^2} + 2V_{10} \sigma_x \sigma_y + 2V_{11} \sigma_y \sigma_z + 2V_{12} \sigma_x \sigma_z \right] / \text{Find}^j$$

where the indices (x, y, z) denote material coordinate direction.

- $V_1$  : maximum allowable stress in the x-direction in tension. Real > 0.0, no Default.
- $V_2$  : maximum allowable stress in the x-direction in compression. Real > 0.0 or blank; Default  $V_1$
- $V_3$  : maximum allowable stress in the y-direction in tension. Real > 0.0, no Default.
- $V_4$  : maximum allowable stress in the y-direction in compression. Real > 0.0 or blank; Default  $V_3$
- $V_5$  : maximum allowable stress in the z-direction in tension. Real > 0.0; no Default
- $V_6$  : maximum allowable stress in the z-direction in compression. Real > 0.0 or blank; Default  $V_5$
- $V_7$  : maximum allowable in-plane shear stress. Real > 0.0 no Default
- $V_8$  : maximum allowable yz shear stress. Real > 0.0 or blank; Default  $V_7$

$V_9$  : maximum allowable zx shear stress. Real > 0.0 or blank; Default  $V_7$

$V_{10}$  : interactive strength constant for the xy plane. Real; no Default

$V_{11}$  : interactive strength constant for the yz plane. Real; no Default

$V_{12}$  : interactive strength constant for the zx plane. Real; no Default

The following restrictions apply to  $V_{10}$ ,  $V_{11}$  and  $V_{12}$  :

$$V_{10}^2 < \frac{1}{V_1 V_2 V_3 V_4} \quad V_{11}^2 < \frac{1}{V_3 V_4 V_5 V_6} \quad V_{12}^2 < \frac{1}{V_1 V_2 V_5 V_6}$$

8. For the Hashin criterion four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \frac{1}{V_{10}^2}(\tau_{xy}^2 + \tau_{xz}^2) & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \frac{|\sigma_x|}{V_2} & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 + \frac{1}{V_{11}^2}(\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2}(\sigma_{xy}^2 + \sigma_{xz}^2) & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \leq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\frac{1}{V_4} \left( \left( \frac{V_4}{2V_{11}} \right)^2 - 1 \right) (\sigma_y + \sigma_z) + \frac{1}{4V_{11}^2} (\sigma_y + \sigma_z)^2 + \frac{1}{V_{11}^2} (\sigma_{yz}^2 - \sigma_y \sigma_z) + \frac{1}{V_{10}^2} (\sigma_{xy}^2 + \sigma_{xz}^2) \right) & \sigma_y + \sigma_z < 0 \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

where the indices (x, y, z) denote material coordinate direction.

$V_1$  : maximum fiber tensile stress. Real > 0.0, no Default.

$V_2$  : maximum fiber compressive stress. Real > 0.0 or blank; Default  $V_1$

$V_3$  : maximum matrix tensile stress. Real > 0.0, no Default.

$V_4$  : maximum matrix compressive stress. Real > 0.0 or blank; Default  $V_3$  .

$V_{10}$  : maximum in plane shear stress. Real > 0.0 no Default

$V_{11}$  : maximum transverse shear stress; Default  $V_{10}$

9. For the Puck failure criterion five failure indices are calculated for the plane stress case and four for the general 3-D case.

The first two failure indices are related to fiber failure and apply to both plane stress and 3-D case:

Tensile fiber mode, for  $\sigma_x > 0$

$$F_1 = \frac{\sigma_x}{V_1}$$

Compressive fiber mode, for  $\sigma_x < 0$

$$F_2 = \frac{|\sigma_x|}{V_1}$$

For plane stress case, the next three failure indices are related to the matrix failure and are given as follows:

$$F_3 = \sqrt{\left(\frac{\sigma_{xy}}{V_{10}}\right)^2 + \left(1 - W_2 \frac{V_3}{V_{10}}\right)^2 \left(\frac{\sigma_y}{V_3}\right)^2} + W_2 \frac{\sigma_y}{V_{10}} \quad \text{for } \sigma_y > 0$$

$$F_4 = \frac{1}{V_{10}} \left( \sqrt{\sigma_{xy}^2 + (W_1 \sigma_y)^2} + W_1 \sigma_y \right) \quad \text{for } \sigma_y < 0 \text{ and } \left( 0 \leq \left| \frac{\sigma_y}{\sigma_{xy}} \right| \leq \frac{R_1}{R_2} \right)$$

$$F_5 = \left( \frac{\sigma_{xy}}{2(1 + W_3 V_{10})} \right)^2 + \left( \frac{\sigma_y}{V_4} \right)^2 \frac{V_4}{|\sigma_y|} \quad \text{for } \sigma_y < 0 \text{ and } \left( 0 \leq \left| \frac{\sigma_{xy}}{\sigma_y} \right| \leq \frac{R_2}{R_1} \right)$$

where the parameters,  $R_1$  and  $R_2$  are as follows:

$$R_1 = \frac{V_{10}}{2(1 + W_1)} \left( \sqrt{1 + 2W_1 \frac{V_4}{V_{10}}} - 1 \right)$$

$$R_2 = V_{10} \sqrt{1 + 2W_3}$$

Failure index for 3-D case is calculated based on stress,  $\sigma_n(\theta)$ ,  $\sigma_{nt}(\theta)$  and  $\sigma_{n1}(\theta)$  on an arbitrary plane with inclination angle  $\theta$ . The stresses are given by:

$$\sigma_n = \sigma_y \cos^2 \theta + \sigma_z \sin^2 \theta + 2\sigma_{yz} \sin \theta \cos \theta$$

$$\sigma_{nt} = (\sigma_z - \sigma_y) \sin \theta \cos \theta + \sigma_{yz} (\cos^2 \theta - \sin^2 \theta)$$

$$\sigma_{n1} = \sigma_{zx} \sin \theta + \sigma_{xy} \cos \theta$$

For 3-D case, the next two failure indices related to matrix failure are given by:

$$F_3 = \sqrt{\left(\frac{1}{V_3} - P_1\right)^2 \sigma_n^2 + \left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2} + P_1 \sigma_n \quad \text{for } \sigma_n \geq 0$$

$$F_4 = \sqrt{\left(\frac{\sigma_{nt}}{R_1}\right)^2 + \left(\frac{\sigma_{n1}}{V_{10}}\right)^2 + (P_2 \sigma_n)^2} + P_2 \sigma_n \quad \text{for } \sigma_n \geq 0$$

where  $P_1$  and  $P_2$  are given as:

$$P_1 = \left(\frac{W_4}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_2}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

$$P_2 = \left(\frac{W_3}{R_1}\right) \frac{\sigma_{nt}^2}{\sigma_{nt}^2 + \sigma_{n1}^2} + \left(\frac{W_1}{V_{10}}\right) \frac{\sigma_{n1}^2}{\sigma_{nt}^2 + \sigma_{n1}^2}$$

The following material parameters are used:

$V_1$  = Maximum fiber tensile stress (Real > 0.0; no Default)

$V_2$  = Maximum fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )

$V_3$  = Maximum matrix tensile stress (Real > 0.0; no Default)

$V_4$  = Maximum matrix compressive stress (Real > 0.0 or blank; Default  $V_3$ )

$V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)

$W_1$  = p12c, slope 1 of failure envelope (Real > 0.0; no Default)

$W_2$  = p12t, slope 2 of failure envelope (Real > 0.0; Default  $W_1$ )

$W_3$  = p23c, slope 3 of failure envelope (Real > 0.0 or blank; Default; calculated internally; this is recommended for plane stress)

$W_4$  = p23t, slope 4 of failure envelope (Real > 0.0; Default  $W_3$ , not used for plane stress)

The Hashin-Tape criterion is a variant of the Hashin criterion adapted for tape type of materials. Four failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \frac{1}{V_3^2}(\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y + \sigma_z > 0 \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left( \left( \frac{V_4}{2V_{11}} \right)^2 - 1 \right) \frac{(\sigma_y + \sigma_z)}{V_4} + \frac{1}{4V_{11}^2} (\sigma_y + \sigma_z)^2 - \frac{\sigma_y \sigma_z}{V_{11}^2} + \left( \left( \frac{\tau_{xy}}{V_{10}} \right)^2 + \left( \frac{\tau_{xz}}{V_{12}} \right)^2 + \left( \frac{\tau_{yz}}{V_{11}} \right)^2 + V_5 \left( \frac{\sigma_x}{V_6} \right)^2 \right) & (\sigma_y + \sigma_z < 0) \\ 0 & \sigma_y + \sigma_z \geq 0 \end{cases}$$

$V_1$  = Maximum tape fiber tensile stress (Real > 0.0; no Default)

$V_2$  = Maximum tape fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )

$V_3$  = Maximum tape cross-fiber tensile stress (Real > 0.0; no Default)

$V_4$  = Maximum tape cross-fiber compressive stress (Real > 0.0 or blank; Default  $V_3$ )

$V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)

$V_{11}$  = Maximum transverse shear stress. (Real > 0.0 or blank; Default  $V_{10}$ )

$V_{12}$  = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default  $V_{11}$ )

$V_5$  = Maximum fiber tensile stress for matrix compression (Real > 0.0 or blank; required if

$V_6 = 1.0$ , otherwise not used)

$V_6$  = Contribution factor for  $V_5$  (Real 0.0 or 1.0; Default = 0.0)

The Hashin-Fabric criterion is a variant of the Hashin criterion adapted for fabric type of materials. Six failure indices are calculated:

$$F_1 = \begin{cases} \left(\frac{\sigma_x}{V_1}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x > 0 \\ 0 & \sigma_x \leq 0 \end{cases}$$

$$F_2 = \begin{cases} \left(\frac{\sigma_x}{V_2}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 & \sigma_x < 0 \\ 0 & \sigma_x \geq 0 \end{cases}$$

$$F_3 = \begin{cases} \left(\frac{\sigma_y}{V_3}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_4 = \begin{cases} \left(\frac{\sigma_y}{V_4}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{yz}}{V_{12}}\right)^2 & \sigma_y < 0 \\ 0 & \sigma_y \geq 0 \end{cases}$$

$$F_5 = \begin{cases} \left(\frac{\sigma_z}{V_5}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

$$F_6 = \begin{cases} \left(\frac{\sigma_z}{V_6}\right)^2 + \left(\frac{\tau_{xy}}{V_{10}}\right)^2 + \left(\frac{\tau_{xz}}{V_{12}}\right)^2 + \left(\frac{\tau_{yz}}{V_{11}}\right)^2 & \sigma_z < 0 \\ 0 & \sigma_z \geq 0 \end{cases}$$

- $V_1$  = Maximum first fiber tensile stress (Real > 0.0; no Default)
- $V_2$  = Maximum first fiber compressive stress (Real > 0.0 or blank; Default  $V_1$ )
- $V_3$  = Maximum second cross-fiber tensile stress (Real > 0.0; no Default)
- $V_4$  = Maximum second cross-fiber compressive stress (Real > 0.0 or blank; Default  $V_3$ )
- $V_5$  = Maximum thickness tensile stress (Real > 0.0; no Default)
- $V_6$  = Maximum thickness compressive stress ((Real > 0.0 or blank; Default =  $V_5$ )
- $V_{10}$  = Maximum in-plane shear stress. (Real > 0.0; no Default)
- $V_{11}$  = Maximum transverse shear stress. (Real > 0.0 or blank; Default  $V_{10}$ )
- $V_{12}$  = Maximum z-x transverse shear stress. (Real > 0.0 or blank; Default  $V_{11}$ )

10. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.
11. User subroutine UFAIL is used along with option MATUDS.
12. In SOL 400, MATF is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATF data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.  
For non-composite material with MATF, only the first failure criteria is calculated and its failure index is outputted in NLOUT.
13. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.
14. Stress limits such as ST, SC, SS,  $X_t$ ,  $X_c$ ,  $Y_t$ ,  $Y_c$  in the MAT1, MAT2 and MAT8 entries are not used in SOL 400.
15. Stress and/or strain allowables in all directions should be defined if the default is not appropriate. Be sure not to set some of these values to low numbers.
16. If MAT8 with Hashin is used in SOL 400 run with PSLDN1 then Hashin FT will be ignored with an appropriate warning message to include MATF instead.

**MATF**

**Material Failure Model - SOL 600**

Specifies failure model properties for linear elastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600. Up to three criterion may be specified for each material if failure indices are desired (ITYPE=0). For progressive failure (ITYPE=2) only one criterion should normally be specified. Even if failure indices (no progressive failure) is the objective of the analysis, it is recommended that only one criterion per material be specified to simplify post processing.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID	ITYPE	SB	UFAIL					
	"CRI"	Criteria	V <sub>1</sub>	V <sub>2</sub>	V <sub>3</sub>	V <sub>4</sub>	V <sub>5</sub>	V <sub>6</sub>	1st
	V <sub>7</sub>	V <sub>8</sub>	V <sub>9</sub>	Find	V <sub>10</sub>	V <sub>11</sub>	V <sub>12</sub>	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	
	"PF"	A1	A2	A3	A4	A5	IC1	IC2	
	IC3	IC4	IC5	IC6	CI7	IC8	IC9		
	"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	2nd
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	
	"CRI"	Criteria	Xt	Xc	Yt	Yc	Zt	Zc	3rd
	Sxy	Syz	Szx	Find	Fxy	Fyz	Fzx	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Example 1 (3 Criteria, no progressive failure):

MATF	100	0							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
		4500.	4500.	4500.					
+									
	CRI	2							2nd
+									
	.11	.06	.1	.05	.075	.03	.03	.03	
	CRI	4	2500.	4000.	2500.	4000.	2000.	3000.	3rd
		4500.	4500.	4500.	0.90				

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)



Example 2 (with progressive failure):

MATF	100	2							
	CRI	1	2500.	4000.	2500.	4000.	2000.	3000.	1st
	4500.	4500.	4500.						
+									
	PF	.001					1	1	
+	1	1							

Alternate Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
MATF	MID	CRI67	V <sub>1</sub>	V <sub>2</sub>	V <sub>3</sub>	V <sub>4</sub>	V <sub>5</sub>	V <sub>6</sub>	
	V <sub>7</sub>	V <sub>8</sub>	V <sub>8</sub>	Find	V <sub>10</sub>	V <sub>11</sub>	V <sub>12</sub>	Ext	
	Exc	Eyt	Eyc	Ezt	Ezc	Gxy	Gyz	Gzx	

Alternate Example Format:

MATF	100	1	2500.	4000.	2500.	4000.	2000.	3000.	
	4500.	4500.	4500.						

Describer	Meaning
MID	Identification number of a MAT1, MAT2, MAT8, MATORT or MAT9 entry. See Remarks 1. and 2. (Integer > 0; no Default)
ITYPE	Flag to invoke progressive failure. (Integer; Default = 0)
	0 No progressive failure compute failure indices only. (Default)
	1 Standard (original) Marc method.
	2 Gradual selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)
	3 Immediate selective stiffness degradation. (MSC Nastran 2005 and subsequent releases)
SB	Allowable shear stress of bonding material between layers (composites only). To use SB in the stiffness formulation of composites, include PARAM,MINCLDSB,1 in the bulk data. (Real, no Default)
UFAIL	Enter the string "UFAIL" if the ufail.f user subroutine is to be used to define your own failure conditions. If UFAIL is entered, leave fields 3 and 4 blank and do not enter any continuation lines. UFAIL is not available using the alternate format. (Character; Default is blank,). The file ufail.f must be in the same directory as the Nastran input file and must be in lower case.
"CRI"	For the primary failure model, warning may lead to slow convergence.



Descriptor	Meaning
CRI67	Used with the alternate format only. (Integer; no Default; Required) It is highly recommended that only one criterion be used. However, up to three criteria from the list under Criteria below can be specified in a packed list as follows: 1000000*ITYPE+10000*C3+100*C2+C1 where C1, C2, C3 are integer values for the various Criteria are listed.
Criteria	Select an integer corresponding to the failure criterion to be applied. (Integer; no Default) Up to three failure criterion may be specified for each MID for SOL 600.
	1 for maximum stress criterion. (See Remark 3.)
	2 for maximum strain criterion. (See Remark 4.)
	3 for Hill failure criterion. (See Remark 5.)
	4 for Hoffman failure criterion. (See Remark a.)
	5 Tsai-Wu failure criterion. (See Remark b.)
	7 Hashin failure criterion. (See Remark 8.)
	8 Puck failure criterion (See Remark 9.)
	9 User-defined failure criterion. (See Remark 8.)
	10 Hashin-Tape (See Remark )
	11 Hashin-Fabric (See Remark )
	For Hashin models or Puck model, see Remarks 9, 8, , for meaning of material data.
$V_1 = X_t$	Maximum tensile stress in x-direction. (Real, 0.0, or blank, no Default)
$V_2 = X_c$	Maximum compressive stress (absolute value) in x-direction (Real, 0.0, or blank, Default = $X_t$ )
$V_3 = Y_t$	Maximum tensile stress in y-direction. (Real, 0.0, or blank, no Default)
$V_4 = Y_c$	Maximum compressive stress (absolute value) in y-direction. (Real, 0.0, or blank, Default = $Y_t$ )
$V_5 = Z_t$	Maximum tensile stress in z-direction. (Real, 0.0, or blank, no Default)
$V_6 = Z_c$	Maximum compressive stress (absolute value) in z-direction. (Real, 0.0, or blank, Default = $Z_t$ )
$V_7 = S_{xy}$	Maximum shear stress in xy-plane. (Real, 0.0, or blank, no Default)
$V_8 = S_{yz}$	Maximum shear stress in yz-plane. (Real, 0.0, or blank, Default = $S_{xy}$ for criterion 5)
$V_9 = S_{zx}$	Maximum shear stress in zx-plane (Real, 0.0, or blank, Default = $S_{xy}$ for criterion 5)
Find	Failure index. See Remarks 5.-a. (Real, 0.0, or blank, Default = 1.0)
$V_{10} = F_{xy}$	Interactive strength constant for xy-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2}\sqrt{\frac{1}{X_t X_c} \frac{1}{Y_t Y_c}}$ for criterion 5)

Describer	Meaning
$V_{11} = F_{yz}$	Interactive strength constant for yz-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2}\sqrt{\frac{1}{Y_t Y_c} \frac{1}{Z_t Z_c}}$ for criteria 5)
$V_{12} = F_{zx}$	Interactive strength constant for zx-plane. (Real, 0.0, or blank, Default = $-\frac{1}{2}\sqrt{\frac{1}{Z_t Z_c} \frac{1}{X_t X_c}}$ for criterion 5).
Ext	Maximum tensile strain in x-direction. (Real, 0.0, or blank, no Default)
Exc	Maximum compressive strain (absolute value) in x-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Eyt	Maximum tensile strain in y-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Eyc	Maximum compressive strain (absolute value) in y-direction. (Real, 0.0, or blank, Default = Eyt for criterion 2)
Ezt	Maximum tensile strain in z-direction. (Real, 0.0, or blank, Default = Ext for criterion 2)
Ezc	Maximum compressive strain (absolute value) in z-direction. (Real, 0.0, or blank, Default = Ezt for criteria 2)
Gxy	Maximum shear strain in xy-plane. (Real, 0.0, or blank, no Default)
Gyz	Maximum shear strain in yz-plane. (Real, 0.0, or blank, Default = Gxy for criterion 2)
Gzx	Maximum shear strain in zx-plane. (Real, 0.0, or blank, Default = Gxy for criterion 2)
“PF”	Enter the character string “PF” to start progressive failure input data if ITYPE is 2 or 3. If the defaults are to be taken, the PF line and the line following the PF line may be omitted (Character, no Default)
A1	Residual stiffness fraction. For Criteria=3, this is the fraction of initial stiffness upon failure. For Criteria=2, the stiffness is not reduced more than this fraction (Real, Default = 0.01)
A2	Must be 0.0 or blank except for the Criteria values listed below
Criteria=7 (Hashin)	A2 is the factor for E2 reduction due to matrix compression failure. Takes values between 0.0 and 1.0 and defaults to 0.0 where E2 is reduced in the same way as for matrix tension. A value of 1.0 leads to no E2 reduction due to matrix compression failure.
Criteria=10 (Hashin Tape)	Same as for Criteria=7
Criteria=8 (Puck)	Same as for Criteria=7

Describer	Meaning
A3	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin)      A3 is the factor for G12 reduction relative to E2 reduction. It takes values between 0.0 and 1.0 and defaults to 0.0 where G12 is reduced in the same way as E2. A value of 1.0 leads to no G12 reduction</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p>
A4	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin)      A4 is the factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=10 (Hashin Tape)      Factor for E3 reduction due to fiber failure. It takes values between 0.0 and 1.0 and defaults to 0.0 where E3 is reduced in the same way as E1. A value of 1.0 leads to an E3 reduction due to E2 only. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure.</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p>
A5	<p>Must be 0.0 or blank except for the Criteria values listed below</p> <p>Criteria=7 (Hashin)      A5 is the factor for G12 reduction from fiber failure and takes values between 0.0 and 1.0. It defaults to 0.0 where G12 reduces to matrix failure. A value of 1.0 leads to G12 reduction due to only fiber failure. Values between 0.0 and 1.0 lead to a mixture of degradation from matrix and fiber failure</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p>
IC1	<p>For all Criteria values except those listed below, IC1 is set to 1 if failure in the positive x-direction is critical (leads to element deactivation).</p> <p>Criteria=7 (Hashin)      Set IC1=1 if fiber tension is critical (leads to element deactivation).</p> <p>Criteria=10 (Hashin Tape)      Same as for Criteria=7</p> <p>Criteria=8 (Puck)      Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>

Describer	Meaning
IC2	<p>For all Criteria values except those listed below, IC2 is set to 1 if failure in the negative x-direction is critical</p> <p>Criteria=7 (Hashin)                      Set IC2=1 if fiber compression is critical</p> <p>Criteria=10 (Hashin Tape)              Same as for Criteria=7</p> <p>Criteria=8 (Puck)                         Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>
IC3	<p>For all Criteria values except those listed below, IC3 is set to 1 if failure in the positive y-direction is critical</p> <p>Criteria=7 (Hashin)                      Set IC3=1 if matrix tension is critical</p> <p>Criteria=10 (Hashin Tape)              Same as for Criteria=7</p> <p>Criteria=8 (Puck)                         Same as for Criteria=7</p> <p>(Integer, Default = 0)</p>
IC4	<p>For all Criteria values except those listed below, IC4 is set to 1 if failure in the negative y--direction is critical</p> <p>Criteria=7 (Hashin)                      Set IC4=1 if matrix compression is critical</p> <p>Criteria=10 (Hashin Tape)              Same as for Criteria=7</p> <p>Criteria=8 (Puck)                         Set IC4=1 if matrix compression mode B is critical</p> <p>(Integer, Default = 0)</p>
IC5	<p>For all Criteria values except those listed below, IC5 is set to 1 if failure in the positive z-direction is critical</p> <p>Criteria=7 (Hashin)                      Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)              Not used leave blank</p> <p>Criteria=8 (Puck)                         Set IC5=1 if matrix compression mode C is critical</p> <p>(Integer, Default = 0)</p>
IC6	<p>For all Criteria values except those listed below, IC6 is set to 1 if failure in the negative z--direction is critical</p> <p>Criteria=7 (Hashin)                      Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)              Not used leave blank</p> <p>Criteria=8 (Puck)                         Not used leave blank</p> <p>(Integer, Default = 0)</p>

Describer	Meaning
IC7	<p>For all Criteria values except those listed below, IC7 is set to 1 if failure in the xy plane is critical</p> <p>Criteria=7 (Hashin)            Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)    Not used leave blank</p> <p>Criteria=8 (Puck)              Not used leave blank</p> <p>(Integer, Default = 0)</p>
IC8	<p>For all Criteria values except those listed below, IC8 is set to 1 if failure in the yz plane is critical</p> <p>Criteria=7 (Hashin)            Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)    Not used leave blank</p> <p>Criteria=8 (Puck)              Not used leave blank</p> <p>(Integer, Default = 0)</p>
IC9	<p>For all Criteria values except those listed below, IC9 is set to 1 if failure in the zx plane is critical</p> <p>Criteria=7 (Hashin)            Not used, leave blank</p> <p>Criteria=10 (Hashin Tape)    Not used leave blank</p> <p>Criteria=8 (Puck)              Not used leave blank</p> <p>(Integer, Default = 0)</p>

Remarks:

1. The MATF Bulk Data entry contains supplementary data for failure prediction of the elastic materials with the same MID. If this capability is used in nonlinear analysis, MATF can activate progressive failure process.
2. Progressive failure behavior for various materials can be simulated using the MATF Bulk Data entry. Failure occurs when any one of the specified failure criterion is satisfied. Upon failure, the elastic modulus reduces to 10% of the original modulus if there is only one value of modulus as in isotropic material or in a beam or truss element. If it pertains to an orthotropic material, all of the material moduli at the integration point are reduced to the lowest modulus specified. The behavior up to the failure point is linear elastic even if an elasto-plastic material is specified, which is followed by a nonlinear behavior for the post-failure analysis. If the initial yield stress is less than the allowable maximum stress, the failure criterion will be ignored.
3. According to the Maximum Stress Criterion, the material fails when any of the stress components (9 components including 6 normal stress components in tension and compression, and three shear stress components) exceeds the maximum allowable stress:

$$|\sigma_{ij}| > Xt, Xc, Yt, \dots, S_{xy}, \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.



4. According to the Maximum Strain Criterion, the material fails when any of the strain components (9 components including 6 normal strain components in tension and compression, and three shear strain components) exceeds the maximum allowable strain:

$$|\varepsilon_{ij}| > Ext, Exc, Ext, \dots, Gxy, \dots \text{etc.}$$

where the indices (x, y, z or i, j) denote material coordinate direction.

5. According to the Hill Failure Criterion, there is no distinction between tensile and compressive behavior. The failure is determined based on

$$\frac{\sigma_x^2}{X^2} + \frac{\sigma_y^2}{Y^2} + \frac{\sigma_z^2}{Z^2} - \left(\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2}\right)\sigma_x\sigma_y - \left(\frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2}\right)\sigma_y\sigma_z - \left(\frac{1}{Z^2} + \frac{1}{X^2} - \frac{1}{Y^2}\right)\sigma_z\sigma_x + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{ind}$$

in which  $X, Y, Z, S_{xy}, S_{yz}, S_{zx}$  are maximum allowable stresses and  $F_{ind}$  is the failure index prescribed by the user.

- a. The Hoffman Failure Criterion introduces distinction between tensile and compressive stresses to generalize the Hill Failure Criterion, i.e.,

$$C_x(\sigma_x - \sigma_y)^2 + C_y(\sigma_y - \sigma_z)^2 + C_z(\sigma_z - \sigma_x)^2 + \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_x + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\sigma_z + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} > F_{ind}$$

with

$$C_x = \frac{1}{2} \left( \frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c} \right)$$

$$C_y = \frac{1}{2} \left( \frac{1}{Y_t Y_c} + \frac{1}{Z_t Z_c} - \frac{1}{X_t X_c} \right)$$

$$C_z = \frac{1}{2} \left( \frac{1}{Z_t Z_c} + \frac{1}{X_t X_c} - \frac{1}{Y_t Y_c} \right)$$

in which  $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$  are maximum allowable stresses and  $F_{ind}$  is the failure index, prescribed by the user.

- b. The Tsai-Wu Failure Criterion is another generalization of the Hill Failure Criterion:

$$\begin{aligned} & \left(\frac{1}{X_t} - \frac{1}{X_c}\right)\sigma_x + \left(\frac{1}{Y_t} - \frac{1}{Y_c}\right)\sigma_y + \left(\frac{1}{Z_t} - \frac{1}{Z_c}\right)\sigma_z + \frac{\sigma_x^2}{X_t X_c} + \frac{\sigma_y^2}{Y_t Y_c} + \frac{\sigma_z^2}{Z_t Z_c} \\ & + \frac{\tau_{xy}^2}{S_{xy}^2} + \frac{\tau_{yz}^2}{S_{yz}^2} + \frac{\tau_{zx}^2}{S_{zx}^2} + 2F_{xy}\sigma_x\sigma_y + 2F_{yz}\sigma_y\sigma_z + 2F_{zx}\sigma_x\sigma_z > F_{ind} \end{aligned}$$

in which  $X_t, X_c, Y_t, Y_c, Z_t, Z_c, S_{xy}, S_{yz}, S_{zx}$  are maximum allowable stresses,  $F_{xy}, F_{yz}, F_{zx}$  are interactive strength constants, and  $F_{ind}$  is the failure index, prescribed by the user.

6. For the Hashin criterion the inputs are used:

$V_1$  = Maximum fiber tensile stress  
 $V_2$  = Maximum fiber compressive stress  
 $V_3$  = Maximum matrix tensile stress  
 $V_4$  = Maximum matrix compressive stress  
 $V_7$  = Maximum in-plane shear stress  
 $V_8$  = Maximum transverse shear stress  
 All other variables should be set to zero

7. For the Puck failure criterion the following inputs are used:

$V_1$  = Maximum fiber tensile stress  
 $V_2$  = Maximum fiber compressive stress  
 $V_3$  = Maximum matrix tensile stress  
 $V_4$  = Maximum matrix compressive stress  
 $V_7$  = Maximum in-plane shear stress  
 Ext = p12c, slope 1 of failure envelope  
 Exc = p12t, slope 2 of failure envelope  
 Eyt = p23c, slope 3 of failure envelope  
 Eyc = p23t, slope 4 of failure envelope

8. For the user-defined failure criterion, leave all fields blank except fields 1-3 of the primary entry (continuation entries are not required).  
 9. For the Hashin-Tape criterion the following inputs are used:



- $V_1$  = Maximum tape fiber tensile stress
- $V_2$  = Maximum tape fiber compressive stress
- $V_3$  = Maximum tape cross-fiber tensile stress
- $V_4$  = Maximum tape cross-fiber compressive stress
- $V_7$  = Maximum in-plane shear stress
- $V_8$  = Maximum transverse shear stress
- $V_9$  = Maximum z-x transverse shear stress
- $V_5$  = Maximum fiber tensile stress for matrix compression
- $V_6$  = Contribution factor for  $Z_t$  (Real 0.0 or 1.0; Default = 0.0)

All other variables should be set to zero

For the Hashin-Fabric criterion the following inputs are used:

- $V_1$  = Maximum first fiber tensile stress
- $V_2$  = Maximum first fiber compressive stress
- $V_3$  = Maximum second cross-fiber tensile stress
- $V_4$  = Maximum second cross-fiber compressive stress
- $V_5$  = Maximum thickness tensile stress
- $V_6$  = Maximum thickness compressive stress
- $V_7$  = Maximum in-plane shear stress
- $V_8$  = Maximum transverse shear stress
- $V_9$  = Maximum z-x transverse shear stress

All other variables should be set to zero

10. A MATTF entry with the same MID as MATF may be used to specify the temperature variation of the failure criterion values.

**Notes:**

1. This Bulk Data entry accommodates Marc's input data under the model definition option FAIL DATA.
2. For the primary format, if only one criterion is needed and no data is required on the 3rd line, it can be omitted. If more than one criterion is needed, all 3 lines are required except for the last one. If the third line of the last one is not required, it may be omitted.
3. For the alternate format the third line may be omitted if no data is needed.

4. For Nastran versions prior to MSC 2007, the alternate format was the only available option. These older versions used ITYPE=1 and a packed list for CRI67 with the format  $100*C3+10*C2+C1$ . ITYPE=1 was an older progressive failure method that is no longer recommended. If these jobs are re-run using MSC 2007, the CRI67 field should be changed as described above or the new primary format used.
5. Stress limits such as ST, SC, SS,  $X_t$ ,  $X_c$ ,  $Y_t$ ,  $Y_c$  in the MAT1, MAT2 and MAT8 entries are not used in SOL 600.

**MATF**

**Material Failure Model - SOL 700**

Specifies failure model properties. Use SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATF	MID								
	“CRI”	Criterion (=101)	MES	DT					
	“CRI”	Criterion (=102)	D1	D2	D3	D4	D5	ε0DOT	
			TROOM	TMELT	CP	MTH			
	“CRI”	Criterion (=103)	MPS	MPS-C	VOLF	DT			
	“CRI”	Criterion (=104)	PRS						
	“CRI”	Criterion (=105)	ALPHA	THETA	GAMMA	BETA	R	D	
			W	X0	CBAR	N	TYPE	ITER	
			TOFF						

Example:

MATF	7								
	CRI	101	2.0	1.0E20					

Describer	Meaning
MID	Identification number of MAT1 or MATORT entry. (Integer > 0; required.)
“CRI”	Enter the character string “CRI” to start input data for a failure criterion.
Criterion	Select an integer corresponding to the failure criterion to be applied. Only one failure criterion is allowed. (Integer; no Default)
101	Maximum Equivalent Stress Failure Model. It is only available for solid elements.
102	Johnson-Cook Failure Model. It is only available for solid elements. See Remarks 1.
103	Maximum Plastic Strain Failure Model. It is only available for Hugh-Liu beams, shells and solid elements.
104	Maximum Pressure Failure Model. It is only available for solid elements.
105	CAP Failure Model. It is only acceptable for solid elements. See Remark 2.-4.

Describer	Meaning
MES	Maximum equivalent stress that causes failure on the deviatoric part of the stress tensor. (Real > 0.0; required)
DT	Minimum time step that causes total failure. (Real > 0.0; default=not used.)
D1-D5	Damage coefficients. (Real; default=0.0)
ε0DOT	Reference plastic strain rate. (Real; default=1.0)
TROOM	Room temperature. (Real; default=0.0)
TMELT	Melt temperature. (Real; default=1.0E20)
CP	Heat capacity. (Real; default=1.0E20)
MTH	Specifies how failure is applied. (Character; default=CONT) CONT      Continuous failure DISC      Discrete failure
MPS	Maximum plastic strain that causes failure. (Real > 0.0; required)
MPS-C	Maximum plastic strain when material is under compression that causes failure. (Real > 0.0; default=MPS-C)
VOLF	Minimum volumetric failure criteria. It is only available for solid elements. (Real > 0.0; default=1.0E-12)
DT	Minimum time step that causes total failure. This option is only acceptable in solid elements. (Real > 0.0; default=not used)
PRS	Maximum pressure that causes failure. (Real > 0.0; required)
ALPHA	Failure envelope parameters. (Real ≥ 0.0; required)
THETA	Failure envelope linear coefficient. (Real ≥ 0.0; required)
GAMMA	Failure envelope exponential coefficient. See Remark 2. (Real ≥ 0.0; required)
BETA	Failure envelope exponent. (Real ≥ 0.0; required)
R	Cap surface axis ratio. (Real ≥ 0.0; required)
D	Hardening law exponent. (Real ≥ 0.0; required)
W	Hardening law coefficient. (Real ≥ 0.0; required)
X0	Hardening law exponent. (Real ≥ 0.0; default=0.0)
CBAR	Kinematic hardening coefficient. (Real ≥ 0.0; default=0.0)
N	Kinematic hardening parameter. (Real ≥ 0.0; default=0.0)
TYPE	Formulation type. (Character; default=SOIL) SOIL      Soil or concrete (cap surface may contract) ROCK      Rock (cap surface does not contract)
ITER	Iteration scheme. (Character; default=VEC) VEC      Fixed number of iterations (vectorized)

Describer	Meaning
	FULL Fully iterative
TOFF	Tension cut off (positive in compression). See Remark 3. (Real < 0.0; default=blank)

**Remarks:**

1. Defines the properties of a failure model where failure is determined by a damage model. The damage model is given by:

$$D = \sum_{time} \frac{\Delta \epsilon_p}{\epsilon^{frac}}$$

$$\epsilon^{frac} = (D_1 + D_2 \exp(D_3 \sigma^*)) \left( 1 + D_4 \ln \frac{\dot{\epsilon}_{pl}}{\dot{\epsilon}_{pl}^0} \right) (1 + D_5 T^*)$$

$$T = SIE / CP$$

$$T^* = \frac{T - T_{room}}{T_{melt} - T_{room}}$$

$$\sigma^* = \frac{\sigma_m}{\bar{\sigma}}$$

The summation is performed over all past time increments. The variable D measures the damage; T is the temperature,  $\sigma_m$  is the mean stress,  $\bar{\sigma}$  is the von Mises equivalent stress, and  $\epsilon^{frac}$  is the fracture strain.

The fracture strain depends on a nondimensional plastic strain rate,  $\dot{\epsilon}_{pl} / \dot{\epsilon}_{pl}^0$ . If D exceeds one it set equal to one.

There are two methods to determine when elements fail:

- Continuous failure: The yield stress is reduced by a factor (1-D). When D exceeds 1 the yield stress equals zero and the element fails.
- Discrete failure: the element fails when D equals one.

This failure model applies to high-strain rate deformation of metals. It is less suitable for quasi-static problems.

2. For a physically meaningful model, the value of the failure envelope exponential coefficient should be less than the failure envelope parameter (ALPHA < GAMMA).
3. The tension cut off value (TOFF) can be defined on the entry and must be less than zero. If the tension cut off is left blank, Nastran SOL700 calculates the tension cut off as the intersection point of the failure envelope surface with the J1-axis as described in Remark 4.

4. The cap material model can be used for geomechanical problems with materials like soil, concrete and rock.

The cap model is characterized by the following constitutive equations:

$$\begin{aligned}\varepsilon &= \varepsilon^e + \varepsilon^p \\ \sigma &= C(\varepsilon - \varepsilon^p)\end{aligned}$$

where  $\varepsilon$ ,  $\varepsilon^e$ , and  $\varepsilon^p$  are the total, elastic and plastic strain tensor,  $C$  is the elasticity matrix and  $\sigma$

is the stress tensor. The flow rule is given by:

$$\dot{\varepsilon}^p = \sum_{k=1}^e \dot{\lambda}_k \frac{\partial f_k}{\partial \sigma}$$

where the sum is over the active yield surfaces  $f^k$ , i.e., the failure envelope ( $k=1$ ), the hardening cap surface ( $k=2$ ), and the fixed tension cutoff surface ( $k=3$ ). The yield conditions are defined by:

$$\begin{aligned}f_1(\sigma) &\leq 0 \\ f_2(\sigma, \kappa) &\leq 0 \\ f_3(\sigma) &\leq 0\end{aligned}$$

The hardening parameter,  $\kappa$ , for the cap model is related to the plastic volume change by a hardening law.

The cap model is a plasticity model described by a yield surface that is defined by means of a failure envelope, a hardening cap and a tension cut off. Figure 1 shows the typical yield surfaces in a cap model.

The failure envelope surface is denoted by  $f_1 = \sqrt{J_{2D}} - \text{MIN}(F_e(J_1), T_{mises})$

and the cap by  $f_2 = \sqrt{J_{2D}} - F_c(J_1, \kappa)$  for  $L(\kappa) \geq J_1 \geq X(\kappa)$

where  $J_1$  is the first invariant (trace) of the stress tensor,  $J_{2D}$  is the second invariant of the stress deviator,  $\kappa$  is an internal state variable that measures hardening as a function of the history of plastic volumetric strain and  $L(\kappa)$  and  $X(\kappa)$  define the range of the cap. Note that  $J_1$  is chosen as negative in tension.

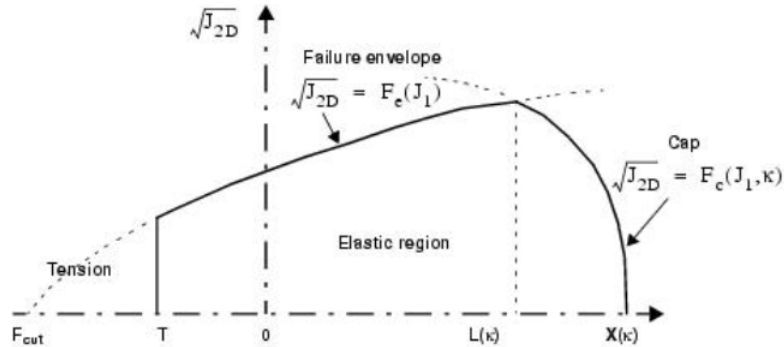


Figure 9-97 Typical Yield Surfaces in a Cap Model

The functions  $F_e$  and  $F_c$  are given by (see References 1. and 2.):

$$F_e(J_1) = a - \gamma \exp(-\beta J_1) + \theta J_1$$

$$F_c(J_1, \kappa) = \frac{1}{R} \sqrt{[X(\kappa) - L(\kappa)]^2 - [J_1 - L(\kappa)]^2}$$

The von Mises type transition failure surface is defined by the

following:  $T_{mises} = \frac{1}{R} |X(\kappa) - L(\kappa)|$

The intersection of the cap with the (hydrostatic)  $J_1$  axis is given by:  $X(\kappa) = \kappa + R F_e(\kappa)$

and  $L(\kappa)$  is defined by:  $L(\kappa) = \kappa = \begin{cases} \kappa & \text{if } \kappa > 0 \\ 0 & \text{if } \kappa \leq 0 \end{cases}$

The hardening parameter,  $\kappa$ , is related to the actual plastic volume

change:  $\epsilon_v^p(X) = W \{ 1 - \exp[-D(X(\kappa) - X_0)] \}$

The tension cut off surface is given by the function:  $f_3 = T - J_1$

where  $T$  is the maximum hydrostatic tension sustainable by the material.

Kinematic work hardening for the failure envelope surface is based on the approach of Isenberg et. Al. [1978]. It is switched on by specifying  $N$ . The failure envelope surface is replaced by a family of envelope surfaces that are bounded by an initial yield surface and by a limiting failure envelope surface. Which member of the family is taken, is implemented by replacing in all yield relations the stress tensor  $\sigma$  by  $\sigma - \zeta$  where  $\zeta$  is a deviatoric tensor that accumulates in time. This tensor  $\zeta$  is called

the “back stress tensor” and is defined by  $\frac{d\zeta}{dt} = \bar{c}\bar{F}(\sigma, \zeta)\frac{d\varepsilon^P}{dt}$

$$F = \text{MAX}\left(0, 1 - \frac{(\sigma - \zeta) \cdot \zeta}{2NF_e(J_1)}\right)$$

Here  $\varepsilon^P$  is the deviatoric plastic strain tensor,  $N$  denotes the size of the yield surface and represents the radial distance between the outside of the initial yield surface and the inside of the limit surface. After each increment of  $\zeta$ , it is checked whether its second invariant exceeds  $N$ . In that case,  $\zeta$  is scaled by a scalar such that its second invariant equals  $N$ . For consistency between the limit surface of the kinematic hardening cap model and the failure envelope of the standard cap model, the parameter  $\alpha$  is placed by  $\alpha - N$ .

### References:

1. Sandler, I. S. and Rubin, D., “An Algorithm and a Modular Subroutine for the Cap Model,” International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 3, 173-186 (1979)
2. Simo, J. C., Ju, J.W., Pister, K. S., and Taylor, R.L., “Assessment of Cap Model: Consistent Return Algorithms and Rate-dependent Extension,” Journal of Engineering Mechanics, Vol. 114, No. 2, February 1988.
3. Isenberg, Jeremy, D K. Vaughan, and I Sandler. Nonlinear Soil-Structure Interaction. Palo Alto, Calif: The Institute, 1978. Print.



## MATF1 Frequency Dependent Isotropic Material Definition

Defines the frequency dependent properties for an isotropic poroelastic material.

Format:

1	2	3	4	5	6	7	8	9	10
MATF1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example:

MATF1	1	30			130				
-------	---	----	--	--	-----	--	--	--	--

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer>0; Required)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer≥0 or Blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer≥0 or Blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer≥0 or Blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer≥0 or Blank)
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. (Integer≥0 or Blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer≥0 or Blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer≥0 or Blank)
T(SC)	Identification number of a TABLEMi entry for the compression stress limit. (Integer≥0 or Blank)
T(SS)	Identification number of a TABLEMi entry for the shear stress limit. (Integer≥0 or Blank)

Remarks:

1. Remarks 1 to 4 under [MATT1](#) are applicable with MATF1..
2. MATF1 must reside under Case Control 'BEGIN BULK TRMC' and is applicable to that trim component only. It may only be referenced by MATPE1 entry for a poroelastic element.

**MATFAB**

**Woven Fabric Material- SOL700**

Defines the properties of a bi-directional woven fabric material for shell elements. Used in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATFAB	MID	RHO	ECOAT	NUCOAT	GCOAT	DAMPCOAT	COMPCOAT	PERC	
+	E1L	E1Q		THETA1	XWARP	YWARP	ZWARP		
+	E2L	E2Q		THETA2	XWEFT	YWEFT	ZWEFT		
+	SCOF	G12	DAMPFIB	COMPFIB	LOCKA NG1	LOCKANG2			

Example:

MATFAB	3	850	5.52E6	0.33					
+	21.6E7				1.0	0.0	0.0		
+	21.6E7				0.0	1.0	0.0		

Describer	Meaning
MID	Unique material number. (Integer > 0; required)
RHO	Density. (Real > 0.0; required)
ECOAT	Young's modulus of coating material. See Remark 1. (Real > 0.0; default=blank)
NUCOAT	Poisson's ratio of coating material. See Remark 1. (Real > 0.0; default=blank)
GCOAT	Shear modulus of coating material. See Remark 1. (Real > 0.0; default=blank)
DAMPCOAT	Damping is applied to the coating stresses. See Remark 2. (Real > 0.0; default=0.1)
	$d\sigma_{ij} = DAMPCOAT \cdot ECOAT \cdot \varepsilon_{ij} dt_{elm}$
COMPCOAT	Scale factor for coating compression stresses. See Remark 3. (0.0? Real?1.0; default=1.0)
PERC	Thickness percentage of coating material. (0.0? Real?100.0; default=0.0=no coating)
E1L	Young's modulus of fabric in warp direction, linear coefficient. (Real > 0.0; required)
E1Q	Young's modulus of fabric in warp direction, quadratic coefficient. (Real > 0.0; default=0.0)
THETA1	Orientation angle between the element coordinate system and the warp ends. See Remark 4. (Real; default=blank)
XWARP	X component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0)
YWARP	Y component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)

Describer	Meaning
ZWARP	Z component of vector indicating the warp direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
E2L	Young's modulus of fabric in weft direction, linear coefficient. (Real > 0.0; required)
E2Q	Young's modulus of fabric in weft direction, quadratic coefficient. (Real ? 0.0; default=0.0)
THETA2	Orientation angle between the element coordinate system and the weft ends. See Remark 4. (Real; default=blank)
XWEFT	X component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
YWEFT	Y component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=1.0)
ZWEFT	Z component of vector indicating the weft direction of the fabric material. The vector is with respect to the basic coordinate system. See Remark 4. (Real; default=0.0)
SCOF	Shear coefficient of friction. See Remark 6. (Real; default=0.0)
G12	Shear modulus of fabric material. See Remark 7. (Real; default=blank)
DAMPFIB	Damping is applied to the fiber stresses. See Remark 2. (Real ≥ 0.0; default=0.1)
	$d\sigma_{ij} = DAMPCOA \bullet E\varepsilon_{ij}dt_{elm}$
COMPFIB	Scale factor for fiber compression stresses. See Remark 3. (0.0 ≤ Real ≤ 1.0; default=1.0)
LOCKANG1	Locking angle 1 for change in fiber cross-over angle. See Remark 10. (Real ≥ 0.0; default=10.0)
LOCKANG2	Locking angle 2 for change in fiber cross-over angle. See Remark 10. (Real ≥ 0.0; default=15.0)

#### Remarks:

1. When a coating is defined (PERC>0), two out of three values need to be specified for ECOAT, NUCOAT, and GCOAT.
2. For air bag modeling the following values of DAMPCOAT and DAMPFIB are suggested:  
DAMPCOAT= 0.05  
DAMPFIB= 0.05
3. The compressive stresses in the fibers are scaled with the value of COMPFIB. Putting COMPFIB = 0.0 results in a tension only fiber model.

The compressive stresses in the coating are scaled with the value of COMPCOAT. Putting COMPCOAT = 0.0 results in a tension only coating model.

The compressive stresses are scaled in the direction of the principal stresses.

When PERC = 100%, and the coating of this fabric model is used to simulate an isotropic air bag material, it is best to scale down the compressive stresses of the coating. A suggested value is COMPCOAT = 0.1

4. Since this is a model which tracks warp and weft directions and uses total warp/weft strain as a state variable, the initial warp and weft directions must be specified. There are two ways to indicate the initial warp and weft directions:
  - a. THETA1 and THETA2  
Orientation angles between the element coordinate system and the warp/weft ends. If no orientation angle is specified, vectors will be used to indicate the warp/weft directions of the fabric material with respect to the basic coordinate system.
  - b. XWARP, YWARP, ZWARP and XWEFT, YWEFT, ZWEFT  
Vectors indicating the warp/weft directions of the fabric material with respect to the basic coordinate system. The projection of these vectors on the surface of each element is used to determine the angle between the element and the material coordinate system. If the orientation angles are defined, these vectors are ignored.
5. For shell element properties (PSHELL1), when the material is MATFAB, the material angle THETA is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.  
For layered composite element properties (PCOMP), when the material of a ply is MATFAB, the angle THETA<sub>i</sub> is ignored. The orientation of the fabric fibers is defined completely on the MATFAB entry.
6. The maximum shear stress is given by a friction coefficient of the fabric (SCOF) times the RMS value of the direct fiber stresses.
7. If the field G12 is left blank, the shear modulus is computed from the RMS value of the two linear stiffness coefficients.
8. When MATFAB material is referenced by shell elements, the Spin Rate method (SPIN) is applied automatically when no stress-rotation correction is specified on SPINCOR option. See PSHELL1 entry for the details on SPINCOR option.
9. There are a number of specific output sublayer variables useful for this material:
  - Q1AF Direction cosines/sines between the element coordinate
  - Q2AFIB System and the warp ends
  - Q1BFIB Direction cosines/sines between the element coordinate
  - Q2BFIB System and the weft picks
  - SGMA Direct stress in fabric parallel to the warp ends
  - SGMB Direct stress in fabric parallel to the weft picks
  - SGFRIC Stress due only to shear in the weave of the fabric
  - EPSFA Strain in fabric parallel to the warp ends
  - EPSFB Strain in fabric parallel to the weft picks
  - ANGLE Crossover angle between warp ends and weft picks

10. When a fabric is being sheared, the angle between the fibers changes. At a certain moment, the fibers will reach a locking angle, after which a further change in the fiber angle is no longer possible.

The simulation models this behavior as follows:

- a.  $\text{Change in Fiber Crossover Angle} < \text{LockAng1}$

The shear stress between the fibers is cut off based on the friction coefficient SCOF

- b.  $\text{LockAng1} < \text{Change in Fiber Crossover Angle} < \text{LockAng2}$

The shear stress between the fibers is linearly increased.

- c.  $\text{Change in Fiber Crossover Angle} > \text{LockAng2}$

The shear stress between the fibers is no longer cut off.

This situation is equal to an infinite friction coefficient SCOF.

**MATFTG**

**Fatigue Material Properties**

Defines fatigue material properties for time domain based SOLs 101, 103, 112 and frequency domain based SOLs 108 and 111.

Format (SOL 101, 103, 112):

1	2	3	4	5	6	7	8	9	10
MATFTG	MID	CNVRT							
	"STATIC"	YS	UTS	CODE	TYPE	RR	SE	mp	
		RA							
	"SN"	SRI1	b1	Nc1	b2	Nfc	SE	BTHRESH	
	"SNS1"	M1	M2	M3	M4	MSS	RTHICK	nTHICK	
	"SNS2"								
	"SNBR1"	SF-FXY	DE-FXY	TE-FXY	SF-MXY	DE-MXY	TE-MXY		
	"SNBR0"	SF-FZ	DE-FZ	TE-FZ	SF-MZ	DE-MZ	TE-MZ		
	"TABLE"	VALUE1	TID1	VALUE2	TID2	VALUE3	TID3		
		VALUE4	TID4	...	...	-etc.-			
	"BASTEN"	A	B	c	Eb	Sd			
	"EN"	Sf	b	c	Ef	n'	K'	Nc	
		SEe	SEp	SEc	Ne	FSN	S		
	"MATID"	E	NU						
		MID1	MID2	MID3	MID4	MID5	MID6	MID7	
		-etc.-							

Format (SOL 108, 111):

1	2	3	4	5	6	7	8	9	10
MATFTG	MID	CNVRT	MATOUT						
	"STATIC"	YS	UTS		TYPE				
	"SN"	SRI1	b1	Nc1	b2	Nfc			
	"TABLE"	VALUE1	TID1						
	"MMPDS"	A1	A2	A3	A4				
	"EN"	Sf	b	c	Ef	n'	K'	GAMMA	
		SEe	SEp	SEc	Ne			TOLER	
	"MATID"	E	NU						
		MID1	MID2	MID3	MID4	MID5	MID6	MID7	
		-etc.-							

Examples:

MATFTG	9								
	STATIC	430	682	99					

MATFTG	9								
	STATIC	430	682				0.1		
	SN	3095	-0.1339	1.e8	0.0	1.e8			

MATFTG	9								
	STATIC	430	682				0.1		
	MATID	2.1e5	0.3						
		9	THRU	15	18				

Describer	Meaning
MID	Unique material ID that matches the identification of a MAT1 entry unless the MATID line is supplied (Integer>0). See Remark 1.
CNVRT	Conversion factor. See Remark 2. regarding units.
MATOUT	SOL 108, 111 only. This is a debug option for writing the fully derived material curve to a file (Integer 0 or 1; Default = 0).
STATIC	Required flag indicating that yield and/or ultimate tensile strengths and other common parameters are supplied (Character=STATIC). See Remark 3. and 4.
YS	Yield strength (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 3000$ ). See Remark 2. regarding units.
UTS	Ultimate tensile strength (Valid range equivalent in MPa: $100.0 \leq \text{Real} \leq 4000$ ). See Remark 2. regarding units.
CODE	SOL 101, 103, 112 only. Material code used in automatically generating S-N or $\epsilon$ -N data curves and for surface finish correction. See Remark 3.
TYPE	SOL 101, 103, 112: Specification of the type of S-N curves defined using the TABLE or BASTEN flag (no Default). See Remark 4.
	SOL 108, 111: Specification as to whether material definitions are specified as RANGE or AMPLitude. See Remark 4.
RR	SOL 101, 103, 112 only. R-ratio of test used to create S-N or $\epsilon$ -N curve. ( $-1.0e30 \leq \text{Real} \leq 1.0$ , Default=-1.0, fully reversed loading).
SE	SOL 101, 103, 112 only. Standard Error of Log(N). ( $0.0 \leq \text{Real} \leq 10.0$ , Default=0.1).
mp	SOL 101, 103, 112 only. Plastic Poisson's Ratio. ( $0.45 \leq \text{Real} \leq 0.55$ , Default=0.5).

Describer	Meaning
RA	<b>SOL 101, 103, 112</b> only. Percentage reduction in area used for deriving $\epsilon$ -N data for metals of type “other” (TYPE field = 200-207 or >300). (0.0<Real≤100.0; no Default).
SN SNS1/SNS2 SNBR1/SNBR0	Flag indicating the definition of an S-N curve(s) follow (Character = SN, optional). See Remark 5. and 7. and 8.
SRI1	Stress range intercept. (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 2.5e4$ , no Default). See Remark 2 regarding units.
b1	First fatigue strength exponent. ( $-1.0 < \text{Real} < -0.02$ , no Default)
Nc1	In 1-segment S-N curve, the cycles limit of endurance. In 2-segment S-N curve, this is the fatigue transition point. Both are defined in cycles. ( $1.0 \leq \text{Real} \leq 1.0E25$ ; no Default).
b2	Second fatigue strength exponent. It is zero when defining 1-segment S-N curve; ( $-0.5 < \text{Real} \leq 0.0$ , Default=0.0).
Nfc	Fatigue cutoff. ( $1.e-9 \leq \text{Real} \leq 1.0e30$ ; Default=1.0e30).
SE	<b>SOL 101, 103, 112</b> only. Standard Error of Log(N). ( $0.0 \leq \text{Real} \leq 10.0$ , Default=0.1). SE from “STATIC” line will be used if left blank.
BTHRESH	<b>SOL 101, 103, 112</b> only. Threshold value of the bending (r) ratio used in interpolation between “stiff” and “flexible” SN curves for fatigue analysis of seam welds. ( $0.0 \leq \text{Real} \leq 0.999$ ; Default=0.5)
M1 - M4	<b>SOL 101, 103, 112</b> only. Mean stress slope parameters M1 through M4 representing sensitivity to mean stress in four (4) regimes of R-ratio as plotted on a constant life Haigh diagram and used in FKM mean stress correction. ( $-0.99 \leq \text{Real} \leq 0.0$ ; Default = blank). See Remark 6.
MSS	<b>SOL 101, 103, 112</b> only. Mean stress sensitivity factor, used only for fatigue analysis of spot welds with CORR=SIMPLE on the FTGPARM entry. ( $0.0 \leq \text{Real} \leq 1.0$ , Default=blank). This mean stress correction method is a simplified version of the FKM method where $M1=M2=M3=M4=-MSS$ .
RTHICK	<b>SOL 101, 103, 112</b> only. Reference thickness/threshold (in consistent model length units) for sheet thickness correction used in fatigue analysis of seam welds. Ignored if THICK=0 on the FTGPARM entry for “SEAMW”. Must be supplied if THICK=1. ( $\text{Real} \geq 1.0e-9$ ; Default=1.0)
nTHICK	<b>SOL 101, 103, 112</b> only. Sheet thickness correction exponent used in the fatigue analysis of seam welds. Ignored if THICK=0 on the FTGPARM entry for “SEAMW”. Must be supplied if THICK=1. (Real; Default=0.16667)
SF-FXY SF-MXY SF-FZ SF-MZ	<b>SOL 101, 103, 112</b> only. Scale factor for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.



Describer	Meaning
DE-FXY DE-MXY DE-FZ DE-MZ	<b>SOL 101, 103, 112</b> only. Diameter exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.
TE-FXY TE-MXY TE-FZ TE-MZ	<b>SOL 101, 103, 112</b> only. Thickness exponent for stress due to FX or FY (shear forces), MX or MY (bending moments), FZ (axial force), and MZ (torsion), respectively (Real, Default = blank) used in fatigue analysis of spot welds only. See Remark 9.
TABLE	Flag indicating the definition of S-N curves as a number of tables follows (Character=TABLE; optional). See Remark 5.
VALUEi	<b>SOL 101, 103, 112:</b> The constant mean stress, R-ratio, or life (in cycles) of this particular S-N curve. (Real; no Default). See Remark 2 regarding units.  <b>SOL 108, 111:</b> This should be specified as 1.0 as only one curve is supported and should be defined as zero mean.
TIDi	<b>SOL 101, 103, 112:</b> A TABLEM1 ID defining the S-N curve of stress range (y) vs. life (x) (in cycles) for this particular S-N curve, or mean stress (x) vs. stress amplitude (y) for constant life (Haigh diagram) curves (Integer>0). See Remark 2. regarding units.  <b>SOL 108, 111:</b> A TABLEM1 (may cause unintended recalculation of eigenvalues if used in RESTART job) or TABRND1 ID defining the S-N curve of stress (y) vs. life (x) (in cycles). Only one curve is allowed and should be defined at zero mean. See Remark 2. regarding units.
MMPDS	<b>SOL 108, 111</b> only. Flag indicating the definition of an S-N curve based on the MMPDS S-N approach (Character=MMPDS; optional). See Remark 5.
A1 A2 A3 A4	Coefficients or exponent of the standard MMPDS-01 material equation (must be defined in units-KSI based on amplitude). (Real; typical ranges $5 \leq A1 \leq 110$ ; $-47 \leq A2 \leq -1.0$ , $0.1 \leq A3 \leq 1.0$ ; $0.0 \leq A4 \leq 114$ ; no Default). CNVRT is not applied to these values at all - it is ignored!
BASTEN	<b>SOL 101, 103, 112</b> only. Flag indicating the definition of an S-N curve based on the Bastenaire S-N approach (Character=BASTEN; optional). See Remark 5.
A	Bastenaire coefficient - a parameter positioning the curve along the life axis. (Valid range equivalent in MPa: $1.0e4 \leq \text{Real} \leq 1.0e10$ ). See Remark 2. regarding units.
B	Scale Factor parameter. (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$ ). See Remark 2. regarding units.
c	Bastenaire exponent. ( $0.01 \leq \text{Real} \leq 1.0e10$ )
Eb	Bastenaire fatigue limit (Valid range equivalent in MPa: $1.0 \leq \text{Real} \leq 1.0e10$ ). See Remark 2 regarding units.

Describer	Meaning
Sd	Bastenaire scatter factor. (Valid range equivalent in MPa: $0.0 \leq \text{Real} \leq 1.0e25$ ). See Remark 2 regarding units.
EN	Flag indicating the definition of an $\varepsilon$ -N curve is to follow. (Character=EN; optional). See Remark 5..
Sf	Fatigue strength coefficient (Valid range equivalent in MPa: $50.0 \leq \text{Real} \leq 4000.0$ ; no Default). See Remark 2. regarding units.
b	Fatigue strength exponent ( $-0.5 \leq \text{Real} \leq -2.0e-3$ ; no Default).
c	Fatigue ductility exponent. ( $-1.2 \leq \text{Real} \leq -0.15$ ; no Default).
Ef	Fatigue ductility coefficient, ( $0.001 \leq \text{Real} \leq 10.0$ ; no Default).
n'	Cyclic strain-hardening exponent, ( $5.0e-3 \leq \text{Real} \leq 0.5$ ; no Default).
K'	Cyclic strength coefficient, (Valid range equivalent in MPa: $50 \leq \text{Real} \leq 1.2e4$ ; no Default). See Remark 5. regarding units.
Nc	<b>SOL 101, 103, 112</b> only. Cut-off (reversals), ( $1.0e5 \leq \text{Real} \leq 1.0e24$ ; Default=2.0e8).
GAMMA	<b>SOL 108, 111</b> only. Additional coefficient needed for the Walker equation, ( $0.0 \leq \text{Real} \leq 1.0$ ; Default=0.0).
SEe	<b>SOL 101, 103, 112</b> only. Standard Error of Log(e) (elastic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0).
SEp	<b>SOL 101, 103, 112</b> only. Standard Error of Log(e) (plastic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0 ).
SEc	<b>SOL 101, 103, 112</b> only. Standard Error of Log(e) (cyclic), ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.0).
Ne	<b>SOL 101, 103, 112</b> only. Endurance Limit (Reversals), ( $1.0 e-9 \leq \text{Real} \leq 1.0e25$ ; Default=2.0e8).
FSN	<b>SOL 101, 103, 112</b> only. Fatemi-Socie Parameter, ( $0.0 \leq \text{Real} \leq 10.0$ ; Default=0.6)
S	<b>SOL 101, 103, 112</b> only. Wang Brown Parameter, ( $-10.0 \leq \text{Real} \leq 10.0$ ; Default=1.0)
TOLER	<b>SOL 108, 111</b> only. Parameter used for frequency based strain life analysis with Neuber notch correction in vibration fatigue only. ( $\text{Real} \geq 1e-10$ , Default = 1e-5)
MATID	Flag indicating that a list of MAT1 IDs is to follow. See Remark 1.
E	Young's modulus, (Valid range equivalent in MPa: $\text{Real} > 0.0$ , no Default).
NU	Elastic Poisson's ratio. ( $0.25 \leq \text{Real} \leq 0.35$ , Default = 0.3)
MIDi	IDs of existing MAT1 entries to which this MATFTG entry will be linked. At least one ID must be supplied. The "THRU" keyword is supported.

Remarks:

1. Element properties must reference MAT1 entries in order to be linked to a MATFTG entry as only metal fatigue analysis of isotropic materials is supported. The MID must match that of an existing MAT1 entry called out by the property entry (e.g. PSHELL) unless the "MATID" line is provided, in which case all the MIDi referenced on the "MATID" line are then linked to the MATFTG entry. When the "MATID" line is used, E (required) and NU (optional) must be provided on this line as the code has no way of knowing from which referenced MAT1 this data should be extracted. Young's Modulus (E) and elastic Poisson's ratio (NU) are extracted from the corresponding MAT1 entry of the same ID when no "MATID" line is present. Usage of E and NU in the fatigue analysis itself is dependent on the type of fatigue analysis requested and fatigue material properties provided. Young's Modulus (E) is mainly used for deriving S-N or  $\epsilon$ -N data when none is provided. If the "MATID" line is used, and a MAT1 of the same ID also exists, that MAT1 ID must be in the MIDi list or it will be ignored in the analysis. In other words, when the "MATID" line is used, only the supplied MIDi IDs are used in the fatigue analysis.
2. The CNVRT field is only used if fatigue material stress based parameters are directly input using the "STATIC," "SN," "EN," "TABLE," or "BASTEN" methods. It is used to allow the user to input the fatigue material stress related parameters, (YS, UTS, SRI1, VALUEi, TIDi, A, B, Eb, K) in different units other than the model's consistent units. Example: model is producing stresses in PSI units, fatigue material parameters input in MPa, the CNVRT factor should be 145.0377 to convert MPa to PSI. Note that the y-values of any referenced TABLEM1 entries are also converted for the S-N method or both the x- and y-values when TYPE=LIFE (Haigh curves). It is also necessary to use the DTI,UNITS for defining the model's stress units. See DTI,UNITS. (Real; Default=1.0)
3. For SOLs 101, 103, and 112, if only STATIC is supplied, then for standard S-N or  $\epsilon$ -N analysis, the S-N or  $\epsilon$ -N curve is derived using the UTS and a material CODE. For fatigue analysis of spot or seam welds, a standard material set is used based on CODE, and UTS can be set to any valid number as the UTS from the standard material set will be used instead (only ferrous and aluminum codes are valid for spot/seam welds). Valid codes are listed in Table 9-22 below. At a minimum UTS must be supplied along with the material CODE and E on MAT1, or an error will be issued. If either flag (SN or EN) is present, then the automatic generation is suppressed if all the data necessary to define S-N or  $\epsilon$ -N curves are given. For SN, these fields are SRI1, b1, Nc1. For EN, these fields are Sf, b, c, Ef, n', K'. Either all of them are present, in which case they are used directly, or all of them are omitted, in which case the parameters are derived as mentioned above. The determination as to whether S-N or  $\epsilon$ -N curves are generated is determined by the TYPE field set on the FTGPARAM entry. If surface finish corrections are to be applied, CODE is also required (see PFTG entry). When curves are derived, the specified CODE gets internally converted to a generic code *ferrous*=99 for CODE<100, *aluminum*=100 for 100 ≤ CODE < 200, *other*=0 for 200 ≤ CODE < 300, or *titanium*=300 for 300 ≤ CODE < 400. When SN or EN flags exist, the generic code is set to *other*, but the CODE is used as given.

For SOLs 108, and 111, if only STATIC is supplied, see Remark 5.

4. For SOLs 101, 103, and 111: TYPE can be set to AMPL, RANGE, or MAX for amplitude, range, or maximum stress, respectively, when using the "BASTEN" flag. Or TYPE can be set to MEAN (or AMEAN), RRATIO (or ARRATIO), or LIFE (or ALIFE) for constant mean, R-ratio, or life (Haigh diagrams), respectively, when using the "TABLE" flag. When using multiple S-N curves for mean

stress correction, CORR=INTERP on FTGPARM must be used. In this case, for RRATIO, a curve at  $R=-1$  is required and for MEAN, a curve at zero (0) mean is required. AMPL, RANGE, and MAX are used to define stress types of Bastenaire models. MEAN vs. AMEAN, or RRATIO vs. ARRATIO, or LIFE vs. ALIFE indicates the stress type of the curves is range or amplitude, respectively.

**For SOLs 108 and 111:** TYPE can be set to AMPL or RANGE to indicate that the S-N curve is defined as amplitude or range and is applicable for the SN and TABLE lines.

- For SOLs 101, 103, and 111:** If an S-N analysis is specified (TYPE field on FTGPARM entry) but no S-N curve is supplied, an error will be issued. If an  $\varepsilon$ -N analysis is specified but no  $\varepsilon$ -N curve is supplied, an error will be issued. The STATIC data is required if SN or EN flags are specified on the MATFTG entry. The SN, BASTEN, and TABLE flags are generally mutually exclusive. When both SN and TABLE lines are defined, if TYPE is set, the SN parameters (SRI1, b1, Nc1, b2) will be ignored. EN flag may be present with SN, BASTEN, or TABLE flags, but one or the other will be ignored based on the type of analysis as set using the TYPE field on the FTGPARM entry. See Remark 7. and 8. for other “SNS1/SNS2/SNBR1/SNBR2” definitions.

**For SOLs 108 and 111:** The STATIC line is required and at least one of SN, TABLE, MMPDS, or EN lines. If no SN, TABLE, MMPDS, or EN lines are present, then the fatigue analysis is skipped and only the random vibration analysis is performed<sup>1</sup>. The use of the TYPE field in the FTGPARM entry has limited affect. Please note: If only SN data is present, it is used by default. If only EN data is present, it is used by default. The SN, TABLE, and MMPDS entries are mutually exclusive and if more are present for any particular MID, then the following hierarchical rule is implement to choose which to use: MMPDS, TABLE, SN. If both SN and EN are defined for a particular MID, EN is selected by default unless TYPE on the FTGPARM is set, in which case the corresponding SN or EN definition is used. However, TYPE is ignored if the corresponding material definition is not present, and the material definition used is then based on the hierarchy just defined.

- For SOLs 101, 103, and 111:** Mean stress sensitivity: M1 for  $R>1$ ; M2 for  $-\infty < R < 0$ ; M3 for  $0 < R < 0.5$ ; M4 for  $0.5 < R < 1$ . If  $M_{1-4}$  are undefined, and the material type (CODE) is given, all the parameters will be estimated using empirically defined rules for the FKM mean stress correction method. If only  $M_2$  is defined, then  $M_1$  and  $M_4$  will be set to zero and  $M_3$  to  $M_2/3$ .
- For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of spot welds, “SNS1” and “SNS2” are used to define the SN curves of the top and bottom sheets to which the spot weld is connected and “SN” is used to define the SN curve of the weld nugget. If no “SN”, “SNS1”, or “SNS2” flags are present, generic SN parameters are used for steel or aluminum per the material CODE supplied. See Table 9-23 for default values used in this case. If CODE is blank, steel is assumed. If only one of the flags is provided, then “SN” = “SNS1” = “SNS2”. If “SNS1” or “SNS2” is missing but the other is present, then “SNS1” = “SNS2”. If “SNS1” and “SNS2” are defined, but not “SN”, then “SN” = “SNS1”.
- For SOLs 101, 103, and 111:** The “SN” line is used for the definition of standard SN parameters for any SN analysis. In the case of fatigue analysis of seam welds, “SNBR1” and “SNBR0” are used to define the flexible (bending ratio=1.0) and stiff (bending ratio=0.0) SN curves. If no “SN”, “SNBR1”, or “SNBR0” flags are present, generic SN parameters are used for steel or aluminum per the material

<sup>1</sup>If the EN line is supplied with only n’ and K’ defined, then the Neuber notch correction is also applied in the random only analysis affecting the resultant output stress PSD.

CODE supplied. See [Table 9-23](#) for default values used in this case. If CODE is blank, steel is assumed. If only “SN” is supplied, then the “SN” parameters are used for both “SNBR1” and SNBR0”. If “SNBR1” or “SNBR0” is missing but the other is present, then “SNBR1” = “SNBR0”. The 3rd and 4th lines of data for these keywords should not be entered as they are applicable to fatigue analysis of spot welds only, and not for seam welds.

9. For SOLs 101, 103, and 111: If SF, DE, & TE values are not supplied, default values are used based on material CODE for steel or aluminum. See [Table 9-24](#) and [Table 9-25](#). These should only be entered for fatigue analysis of spot welds and not for anything else.

Table 9-22 Table of Material CODEs

CODE	Description
1	Flake cast iron (FCI)
2	Ferritic cast iron with compacted graphite (FCICG)
3	Pearlitic cast iron with compacted graphite (PCICG)
4	Bainitic cast iron with compacted graphite (BCICG)
5	Ferritic cast iron with spheroidal graphite (FCISG)
6	Ferrite/pearlite cast iron with spheroidal graphite (FPCISG)
7	Pearlitic cast iron with spheroidal graphite (PCISG)
8	Bainitic cast iron with spheroidal graphite (BCISG)
9	Cast steel with less than 0.2% carbon (CSL2C)
10	Normalized cast steel with 0.2-0.4% carbon (NCS24C)
11	Quenched & tempered cast steel with 0.2-0.4% carbon (QTCS24)
12	Normalized cast steel with 0.4-0.7% carbon (NCS47)
13	Plain carbon wrought steel with < 0.2% carbon (PCWS)
14	Hot rolled/normalized plain carbon wrought steel, 0.2-0.4% carbon (HNPCWS24)
15	Quenched & tempered cast steel with 0.4-0.7% carbon (QTCS47)
16	Quenched & tempered plain carbon wrought steel, 0.2-0.4% carbon (QTPCWS24)
17	Hot rolled/normalized plain carbon wrought steel, 0.4-0.7% carbon (HNPCWS47)
18	Quenched & tempered plain carbon wrought steel, 0.4-0.7% carbon (QTPCWS47)
19	Normalized low alloy wrought steel (NLAWS)
20	Quenched & tempered low alloy wrought steel (QTHSLAWS)
21	Normalized Ni/Cr/Mo wrought steel (NNCMWS)
22	Quenched & tempered Ni/Cr/Mo wrought steel (QTNCMWS)
23	Austenitic stainless steel (ASS)
24	Ferritic stainless steel (FSS)
25	Martensitic stainless steel (MSS)
26	Annealed plain carbon wrought steel, 0.2-0.4% carbon (APCWS24)
27	Annealed plain carbon wrought steel, 0.4-0.7% carbon (APCWS47)
28	Normalized carbon/manganese steel (MCMS)
29	Quenched and tempered carbon/manganese steel (QTCMS)
30	Hardened chromium steel (HCS)
31	Quenched and tempered chromium steel (QTCS)

CODE	Description
99	Steel of unknown heat treatment (STEEL)
100	Wrought aluminum (WA)
101	Wrought aluminum-copper alloy (WACA)
102	Wrought aluminum-manganese alloy (WAMNA)
103	Wrought aluminum-magnesium alloy (WAMGA)
104	Wrought aluminum-magnesium-silicon alloy (WAMGSA)
105	Wrought aluminum-zinc alloy (WAZA)
106	Cast aluminum alloy (CAA)
107	Wrought complex special purpose aluminum alloys (WCSPAA)
200	Wrought copper (WCU)
201	Wrought brass (WBR)
202	Wrought aluminum bronze (WABR)
203	Cupronickel (CUPNI)
204	Nickel silver (NIAG)
205	Wrought phosphor bronze (WPHBR)
206	Wrought copper beryllium (WCUBE)
207	Cast copper alloys (CCUA)
300	Titanium alloy (TA)
400	Wrought magnesium alloys (WMGA)
401	Cast magnesium alloys (CMGA)
500	Fusible alloys, solders (FUSSOL)
600	Cast zinc alloys (CZINCA)
700	Wrought nickel alloys (WNIA)
701	Cast nickel alloys (CNIA)
800	Precious metals (PRECMET)
900	Clad materials (CLADMAT)
1000	Thermoplastics (THERPLAS)
1001	Thermosetting plastics (TSETPLAS)

Table 9-23 Spot and Seam Weld Fatigue Analysis SN Curve Defaults

Property	Generic Nugget Spot Weld		Generic Top/Bottom Sheet Spot Weld		Generic Flexible Seam Weld (r=1)		Generic Stiff Seam Weld (r=0)	
	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199
CODE	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199
UTS (MPa)*	4000	4000	4000	4000	1.8E4	4000	1.8E4	4000
YS (MPa)	355	150	355	150	Blank	Blank	Blank	Blank
E (MPa)	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4
SRI1 (MPa)	2100	2462	2900	2462	3.6E4	3140	1.8E4	1275
b1	-0.1667	-0.2	-0.1667	-0.2	-0.3333	-0.1734	-0.3333	-0.1625
Nc1	1E6	1E7	1E6	1E7	1E7	1E7	1E7	1E7
b2	-0.09091	-0.1111	-0.09091	-0.1111	-0.3333	-0.1734	-0.3333	-0.1625
SE	0.334	0.33	0.33	0.33	0.4	0.4	0.4	0.4
RR	0.0	0.0	0.0	0.0	-1.0	-1.0	-1.0	-1.0
Nfc	1E30	1E30	1E30	1E30	1E30	1E30	1E30	1E30
MSS	0.1	0.2	0.1	0.2	Blank	Blank	Blank	Blank
M1	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
M2	-0.1	-0.2	-0.1	-0.2	-0.25	-0.25	-0.25	-0.25
M3	-0.1	-0.2	-0.1	-0.2	-0.1	Blank	-0.1	Blank
M4	Blank	Blank	Blank	Blank	-0.1	Blank	-0.1	Blank
BTHRESH	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
RTHICK	Blank	Blank	Blank	Blank	1 (mm)	1 (mm)	1 (mm)	1 (mm)
nTHICK	Blank	Blank	Blank	Blank	0.16667	0.16667	0.16667	0.16667

\*Both Spot Weld and Seam Weld fatigue analyses use structural stresses, which do not relate to UTS. Corrections based on UTS do no apply to welds.



Table 9-24 Steel Scale Factor, Diameter & Thickness Exponent Defaults for Fatigue Analysis of Spot Welds

Steel (CODE < 100)	Scale Factor	Diameter Exponent	Thickness Exponent
FX,FY	SF-FXY = 1.0	DE-FXY = 0.0	TE-FXY = 0.0
MX,MY	SF-MXY = 0.6	DE-MXY = 0.0	TE-MXY = 0.5
FZ	SF-FZ = 0.6	DE-FZ = 0.0	TE-FZ = 0.5
MZ	SF-MZ = 0.0	DE-MZ = 0.0	TE-MZ = 0.0

Table 9-25 Aluminum Scale Factor, Diameter & Thickness Exponent Defaults for Fatigue Analysis of Spot Welds

Aluminum (100 ≤ CODE ≤ 200)	Scale Factor	Diameter Exponent	Thickness Exponent
FX,FY	SF-FXY = 0.4	DE-FXY = 0.5	TE-FXY = -0.25
MX,MY	SF-MXY = 0.4	DE-MXY = 0.5	TE-MXY = -0.25
FZ	SF-FZ = 1.0	DE-FZ = 0.0	TE-FZ = 1.0
MZ	SF-MZ = 0.0	DE-MZ = 0.0	TE-MZ = 0.0

## MATG

## Gasket Material Properties

Specifies gasket material properties to be used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATG	MID	IDMEM	BEHAV	TABLD	TABLU1	TABLU2	TABLU3	TABLU4	
	TABLU5	TABLU6	TABLU7	TABLU8	TABLU9	TABLU10	YPRS	EPL	
	GPL	GAP	TABYPRS	TABEPL	TABGPL	TABGAP	N/A	N/A	

Example:

MATG	100	10	0	1001	1002	1003			
							100.	2500.	
	950.	0.0							

Descriptor	Meaning
MID	Material ID number. (Integer)
IDMEM	ID of MAT1 providing material behavior for membrane behavior. See Remarks. (Integer)
BEHAV	Behavior type (presently only type 0 is supported). (Integer)
TABLD	ID of a TABLES1 table providing loading path of the gasket (pressure versus displacement). See Remarks 1. and 3. (Integer)
TABLUi	ID of TABLES1 table providing unloading path(s) of the gasket (pressure versus displacement) can range from 1 to 10. If there is no unloading, no unloading tables need be entered. Leave fields blank for tables that are not required. See Remarks. (Integer)
YPRESS	Yield pressure. See Remark 4. (Real)
EPL	Tensile modulus (pressure per unit length). (Real)
GPL	Transverse shear modulus (force per unit area). (Real)
GAP	Initial gap (if present). (Real)
TABYPRS	ID of TABLES1 table associated with yield pressure (not presently used). (Integer)
TABEPL	ID of TABLES1 table associated with tensile modulus (not presently used). (Integer)
TABGPL	ID of TABLES1 table associated with transverse shear modulus (not presently used). (Integer)
TABGAP	ID of TABLES1 table associated with initial gap (not presently used). (Integer)

Remarks:

1. MATG defines nonlinear properties in the thickness direction for compression only, designed for gasket-like materials. MATG has anisotropy only in the thickness direction, which is called normal anisotropy.
2. The MATG option can only be used with continuum composite elements, use PLCOMP.
3. The MATG entry defines the compressive behavior in thickness. The thickness direction is the principal direction (3) in 3-dimensional solids and (2) for 2-dimensional solids (plane strain and axisymmetric elements). Since MATG material allows only normal anisotropy, linear properties in MAT1 are required for in-plane behavior.
4. The initial yield pressure should match a point in table TABLD.
5. The loading path for the gasket is always in compression. However, it starts from the origin to initial yield pressure (nonlinear elastic range) and continues with strain hardening slope into the plastic region. All the data points are specified in the first quadrant.
6. As many as 10 unloading paths may be defined in the thickness direction using TABLS1 in pressure vs. gasket closure distance as in the loading path. All the unloading paths must start from zero pressure and end at the loading path in the plastic region. Unloading behavior at undefined paths will be interpolated between two adjacent unloading paths. The last point of the last specified unloading path signifies full compression, which does not allow any further closure beyond the point.

Element Thickness Direction for MATG	Runs Parallel from Face	To Face
Element T direction	G1-G2-G3-G4	G5-G6-G7-G8

7. If creep analysis is required, Bulk Data entry, MPCREEP, must also be entered to activate Marc's CREEP parameter.

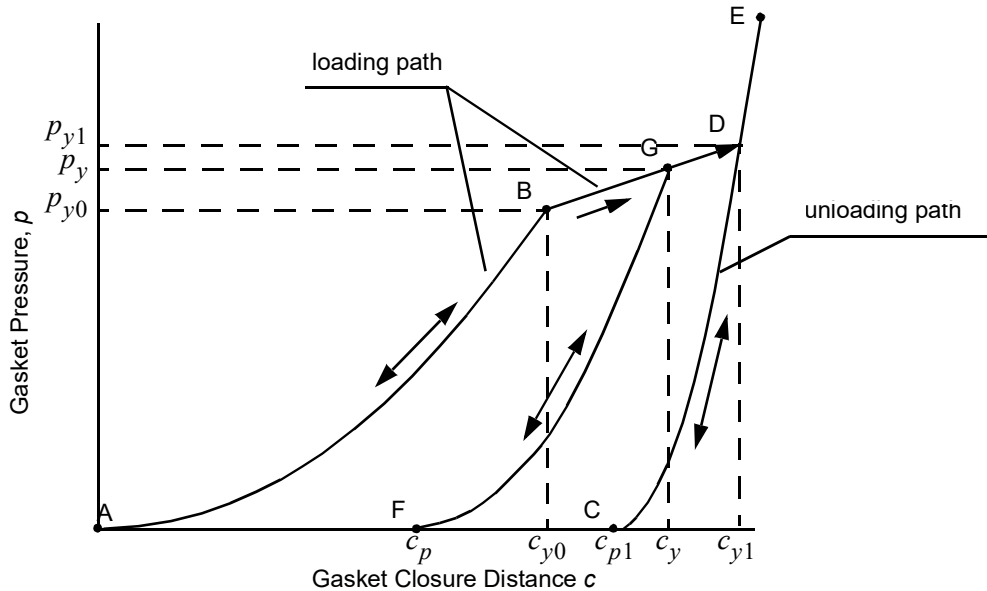


Figure 9-98 Pressure-closure Relation of a Gasket

8. See associated MATTG entry for temperature variation of these properties.
9. All continuation cards must be entered.
10. MID, IDMEM, BEHAV, TABLD, TABLU1, YPRS, EPL and GPL must be non-zero.
11. Each unloading curve must begin with gasket pressure of 0.0. Subsequent unloading curves must start with larger closure distances (when gasket pressure is 0.0) than previous unloading curves.
12. Points on loading and unloading curves must be defined in order of increasing gasket pressure.
13. MATG may be referenced by “solid” composite elements only, via PCOMPLS linked to PSOLID and PSHLN2 linked to PLPLANE.
14. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
15. Result data recovery is not supported in SOL 400 when ANALSYS=MODES, MFREQ or DFREQ.

**MATHE**

**Hyperelastic Material Properties**

Specifies hyperelastic (rubber-like) material properties for nonlinear (large strain and large rotation) analysis in SOL 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin Model (Model = Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	
	C10	C01	D1	TAB1	TAB2	TAB3	TAB4	TABD	
	C20	C11	C02	D2	NA	ND			
	C30	C21	C12	C03	D3				
	C40	C31	C22	C13	C04	D4			
	C50	C41	C32	C23	C14	C05	D5		

Format 2: (Model = Ogden or Foam)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model	NOT	K	RHO	Texp	Tref	GE	
	Mu1	Alpha1	Beta1						
	Mu2	Alpha2	Beta2	Mu3	Alpha3	Beta3			
	Mu4	Alpha4	Beta4	Mu5	Alpha5	Beta5			
	D1	D2	D3	D4	D5				

Note that Foam does not support D1~D5.

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	
	NKT	N/E	Im						
	D1	D2	D3	D4	D5				

Format 4: User Strain Energy Function (Model = GHEMi) (SOL 400 only)

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO	Texp	Tref	GE	

Example - Format 1:

MATHE	2	Foam		3.30E+07	0.00035	4.00E-06	75	0.02	
	2.00E+04	1.5	1.15						
	1.50E+04								

Example - Format 2:

MATHE	1	Ogden	2	200000					
	16	2	1.15						
	-4	-2							

Example - Format 3:

MATHE	2	Gent		3.30E+07	0.00035	4.00E-06	75	0.02	
	2.00E+04	1250							

Example - Format 4:

MATHE	2	GHEM1		3.30E+07	0.00035	2.50E-06	70.5	0.015	
-------	---	-------	--	----------	---------	----------	------	-------	--

Describer	Meaning
MID	Identification number of a MATHE entry. (Integer > 0; no Default)
Model	Select hyperelastic material model from (Character; Default = Mooney):
Mooney	for generalized Mooney-Rivlin hyperelastic model. See Remark 1.
Ogden	for Ogden hyperelastic model. See Remark 2.
Foam	for foam model. See Remark 3.
Aboyce	for Arruda-Boyce strain energy model. See Remarks 4. and 17.
Gent	for Gent strain energy model. See Remarks 5. and 17.
GHEM1	Invariant-base user subroutine (foam models only) $W = W(I_1, I_2, I_3)$
GHEM2	Principal-stretch based user subroutine (foam models only) $W = W(\lambda_1, \lambda_2, \lambda_3)$
GHEM3	Invariant-based with volumetric and deviatoric split user subroutine (foam models only) $W = W_{dev}(I_1, I_2) + U(J)$
GHEM4	Principal-stretch based with volumetric and deviatoric split user subroutine (foam models only) $W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)$
GHEM5	Invariant-based with deviatoric part only (rubber models only) $W = W_{dev}(I_1, I_2)$

Describer	Meaning
GHEM6	<p>Principal-stretch based with deviatoric part only (rubber models only)  <math>W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)</math></p> <p>where <math>I_1, I_2, I_3</math> (<math>\lambda_1, \lambda_2,</math> and <math>\lambda_3</math>) are strain invariants (principal stretches), and <math>\bar{I}_1</math> and <math>\bar{I}_2</math> (<math>\bar{\lambda}_1, \bar{\lambda}_2,</math> and <math>\bar{\lambda}_3</math>) are their deviatoric parts, defined by <math>\bar{I}_1 = J^{-2/3} I_1</math> and <math>\bar{I}_2 = J^{-4/3} I_2, \bar{\lambda}_i = J^{-1/3} \lambda_i</math> (<math>i = 1, 2, 3</math>); <math>J</math> is the determinant of the deformation gradient.            For GHEMi models, see Remarks 13., 14., and 15.</p>
NOT	Number of terms to be used in Ogden and Foam model. ( $0 < \text{Integer} \leq 5$ ; Default = 1)
K	Specifies a bulk modulus. See Remarks 7. and 16. (Real > 0 or -1.0; Default = automatically set for nearly incompressible condition. Enter -1.0 if nonlinear volumetric strain energy function defined by Di will be used.
RHO	Mass density in original configuration. (Real; Default = 0.0)
Texp	Coefficient of thermal expansion. See Remark 8. (Real; Default = 0.0)
Tref	Reference temperature at which the thermal expansion coefficient is measured. Tref is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)
GE	<p>If GE is real it represents the structural damping coefficient. See Remark 12. (Real; Default = 0.0).            For SOL 600 only, if GE is integer, it represents the form of the user subroutine uelastomer.f as follows:</p> <ol style="list-style-type: none"> <li>1 Foam Material - Invariant-based model  <math>W = W(I_1, I_2, I_3)</math></li> <li>2 Foam Material - Principal stretch based  <math>W = W(\lambda_1, \lambda_2, \lambda_3)</math></li> <li>3 Foam Material -Invariant-based with volumetric and deviatoric split  <math>W = W_{dev}(\bar{I}_1, \bar{I}_2) + U(J)</math></li> <li>4 Foam Material - Principal stretch with volumetric and deviatoric split  <math>W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3) + U(J)</math></li> <li>5 Rubber Material (Mooney, Arruda-Boyce or Gent) - Invariant based  <math>W = W_{dev}(\bar{I}_1, \bar{I}_2)</math></li> <li>6 Rubber Material (Ogden) - Principal stretch based)  <math>W = W_{dev}(\bar{\lambda}_1, \bar{\lambda}_2, \bar{\lambda}_3)</math></li> </ol>

Describer	Meaning
	<p>where <math>I_1, I_2,</math> and <math>I_3</math> (<math>\lambda_1, \lambda_2,</math> and <math>\lambda_3</math>) are strain invariants (principal stretches), and <math>\bar{I}_1</math> and <math>\bar{I}_2</math> (<math>\bar{\lambda}_1, \bar{\lambda}_2,</math> and <math>\bar{\lambda}_3</math>) are their deviatoric parts, defined by <math>\bar{I}_1 = J^{-2/3} I_1</math> and <math>\bar{I}_2 = J^{-4/3} I_2, \bar{\lambda}_i = J^{-1/3} \lambda_i</math> (<math>i = 1, 2, 3</math>); <math>J</math> is the determinant of the deformation gradient. If GE is integer, Bulk Data entry USRSUB6 is also required (see <i>Marc Volume D</i> for more details.)</p>
NA	Order of the distortional strain energy polynomial function. See Remark 6. ( $0 < \text{Integer} < \text{or} = 5$ ; Default = 2; Default = 5 for SOL 600)
ND	Order of the Volumetric strain energy polynomial function. See Remark 6. ( $0 < \text{Integer} < \text{or} = 5$ ; Default = 1; Default = 5 for SOL 600)
Cij	Material constants related to distortional deformation for generalized Mooney-Rivlin model. (Real; Default = 0.0) SOL 600 uses only five constants (C10, C01, C20, C11, and C30) and ignores others.
Di	Material constants related to volumetric deformation. (Real $\geq 0.0$ ) Needed only if $K = -1.0$ . Not used for foam model. See Remark 7. For SOL 600, only Mooney model supports nonlinear volumetric strain energy function.
TAB1	Table identification number of a TABLES1 entry that contains simple tension-compression data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios $l/l_0$ and y-values must be values of the engineering stress $F/A_0$ . $l_0$ is the initial length and $A_0$ is the initial cross-sectional area. See Remark 9. (Integer $> 0$ or blank)
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be stretch ratios $l/l_0$ and y-values must be values of the engineering stress $F/A_0$ . $l_0$ is the initial length and $A_0$ is the initial cross-sectional area. See Remark 9. (Integer $> 0$ or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants Cij. The x-values in the TABLES1 entry must be values of the shear strain and y-values must be values of the engineering shear stress. (Integer $> 0$ or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants Cij. The x and y values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and the values of the nominal stress $F/A_0$ . $l_0$ and $A_0$ are the initial length and cross-sectional area, respectively, in the l-direction. See Remark 9. (Integer $> 0$ or blank)



Describer	Meaning
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constant Di. The x-values in the TABLES1 entry must be values of the volume ration $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; y-values must be values of the pressure, assumed positive in compression. See Remark 9. (Integer > 0 or blank)
Muk	Coefficients $\mu_k$ of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)
Alphak	Coefficients $\alpha_k$ of the strain energy function for Ogden or foam material. See Remarks 2. and 3. (Real; Default = 0)
Betak	Coefficients $\beta_k$ of the strain energy function for foam material. These fields should be left blank for Ogden mode. See Remarks 2. and 3. (Real; Default = 0)
NTK	Material constant for Arruda-Boyce strain energy model. (Real > 0; Default = 1)
N/E	Material constant representing the number (N) of statistical links of the chain for Arruda-Boyce model; or tensile modulus (E) for Gent strain energy model. (Real > 0; Default = 1)
Im	Maximum first invariant for Gent strain energy model. (Real > 0; Default = 0)

Remarks:

- The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$W(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^3 C_{ij}(\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + 4.5K(J^{1/3} - 1)^2$$

with

$$\bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$$

$$\bar{I}_2 = \bar{\lambda}_1^2 \bar{\lambda}_2^2 + \bar{\lambda}_2^2 \bar{\lambda}_3^2 + \bar{\lambda}_3^2 \bar{\lambda}_1^2$$

where  $K$  and  $J$  are bulk modulus and volume ratio, respectively.

For small strains, the shear modulus  $G$  is related to the Mooney-Rivlin constants by

$$G = 2(C_{10} + C_{01}) \text{ with } C_{01} \approx 0.25 C_{10}$$

The model reduces to a Mooney-Rivlin material with only two constants ( $C_{10}$  and  $C_{01}$ ), and to a Neo-Hookean material with one constant ( $C_{10}$ ). The third order Mooney-Rivlin model in SOL 600 uses only five distortional constants ( $C_{10}$ ,  $C_{01}$ ,  $C_{11}$ ,  $C_{20}$ ,  $C_{30}$ ) and the bulk modulus  $K$  for volumetric deformation. Instead of MATHE, the hyperelastic material can be specified using MATHP Bulk Data entry in SOLs 106, 129, and 600.

2. For the Ogden material model, the strain energy function is

$$W = \sum_{k=1}^5 \frac{\mu_k}{\alpha_k} (\bar{\lambda}_1^{\alpha_k} + \bar{\lambda}_2^{\alpha_k} + \bar{\lambda}_3^{\alpha_k} - 3) + 4.5K(J^{1/3} - 1)^2$$

where  $\alpha_k$  and  $\mu_k$  are the moduli and exponent constants, while  $\bar{\lambda}_i$  is the deviatoric stretch ratio defined as:

$$\bar{\lambda}_i = J^{-\frac{1}{3}} \lambda_i$$

and  $J$  and  $K$  are the determinant of the deformation gradient and the bulk modulus, respectively. A two-term Ogden model is equivalent to a simple Mooney-Rivlin model

$$\begin{aligned} \mu_1 &= 2C_{10} \quad \text{and} \quad \mu_2 = 2C_{01} \\ \text{with} \quad \alpha_1 &= 2. \quad \text{and} \quad \alpha_2 = 2. \end{aligned}$$

3. For foam material model, the distortional strain energy function is the same as Ogden. The volumetric energy function of foam model is defined by a polynomial function with coefficients  $\beta_i$
4. For the Arruda-Boyce model, the strain energy function is

$$W = N_{KT} \left[ \frac{1}{2} (\bar{I}_1 - 3) + \frac{1}{20N} (\bar{I}_1^2 - 9) + \frac{1}{1050N^2} (\bar{I}_1^3 - 27) + \frac{19}{7000N} (\bar{I}_1^4 - 81) + \frac{519}{67375N^4} (\bar{I}_1^5 - 243) \right] + 4.5K(J^{1/3} - 1)^2$$

$$\text{with } \bar{I}_1 = \bar{\lambda}_1^2 + \bar{\lambda}_2^2 + \bar{\lambda}_3^2$$

where  $N_{KT}$  is a material constant and  $N$  is a material parameter representing the number of statistical links of the material chain.

If the material test data are available from multiple experiments such as uniaxial and equi-biaxial tests, the Ogden model is more accurate in fitting experimental results. If only uniaxial tension data is available, the Arruda-Boyce model provides more accurate data fitting for multiple modes of deformation.

5. For the Gent model, the strain energy function is

$$W = -\frac{1}{6}EI_m \log \left[ \frac{I_m}{I_m - \bar{I}_1 + 3} \right]$$

where  $E$  and  $I_m$  are tensile modulus and maximum first invariant, respectively.

6. The curve fitting is activated if any of the TAB1, TAB2, TAB3, TAB4 and/or TABD are specified. The NA and ND fields are used in case of the curve fitting for Mooney from the experimental data.
7. Although the conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible, SOL 600 provides a compressible rubber model. Nearly incompressible material may be simulated with a large value of  $K$ . The default value for the Mooney-Rivlin model represents a nearly incompressible condition, which is  $K = 10^4 (C10 + C01)$ . In case of Ogden model, the default is  $2500 \cdot \sum \mu_k \alpha_k$ . If any of the  $D_i$  are non-zero, then the entered value of the Bulk Modulus must be -1.0 and the nonlinear volumetric strain energy function will be used. See Remark 16.
8. The thermal expansion coefficient is a secant value measured with respect to a temperature,  $T_{ref}$ .

$$\text{The thermal strain is computed by } \varepsilon_{th} = \bar{\alpha}(T - T_0)$$

where  $T_0$  is an initial temperature. The secant coefficient of thermal expansion is related to the instantaneous coefficient of thermal expansion by

$$\alpha = \frac{d\varepsilon_{th}}{dT} = \bar{\alpha} + \frac{d\bar{\alpha}}{dT}(T - T_0)$$

The choice of secant or instantaneous coefficient (CTE) of thermal expansion is determined by PARAM, MARALPHA in SOL 600. All CTE's in SOL 600 models must be either secant or instantaneous.

9. All the material constants may be obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties. The bulk modulus  $K$  may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 fields are blank, the material constants must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supersedes the manual input of the parameters. It is recommended that the GUI be used to determine the materials constants, so one can visualize the correlation between the numerical representation and the experimental data. It is recommended that the experimental data span the range of anticipated strains to reduce the chances of adverse numerical calculations.
10. All the alphanumeric fields are recognizable by the first four letters.
11. Enter NKT and N/E for Aboyce (leave  $I_m$  blank). Enter N/E and  $I_m$  for Gent (leave NKT blank).
12. For SOL 600, Bulk Data entry USRSUB6 is also required.

13. The GHEMi models using the user subroutine are only available if the updated Lagrange formulation is used for the hyperelastic material. The user subroutine MD UELASTOMER defined with the GHEMi must be used along with the Bulk Data entry MATUDS.
14. The GHEMi are not available for plane stress, shell, membrane, and beam elements.
15. For the GHEM3 and GHEM4 models, the user subroutine is called twice, once with either a 3 or 4 to evaluate the deviatoric strain energy and once with a 7 to evaluate the volumetric strain energy.
16. The series representation of the volumetric strain energy is only available for the updated Lagrange formulation.
17. The Gent and Arruda-Boyce models are not available for plane stress, shell, membrane, and beam elements.
18. The Ogden, Foam, Arruda-Boyce and Gent models are not available in SOLs 106 and 129.
19. In SOL 400, MATHE is only supported for nonlinear elements with property extensions. This implies that for such elements, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for incompressible rubber models, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral). Note that only PSOLID with PSLDN1 should be used for 3D solid nonlinear elements and PLSOLID cannot be used for MATHE. Note also that, prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
20. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

#### Notes (SOL 600 only):

1. The structural damping constant GE is specified in Marc under the option DAMPING as a numerical damping  $\gamma$ , i.e.,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE;  $\alpha$  and  $\beta$  are equivalent to user parameters ALPHA1 and ALPHA2.

2. Material curve-fitting from experimental data is not supported in SOL 400. Direct material property input is required.
3. This Bulk Data entry accommodates Marc's input data under the model definition options MOONEY, OGDEN, and FOAM as well as the parameter ELASTICITY. It also accommodates MATHP input data in Nastran.

**MATHE** Hyperelastic Material Properties - SOL 700

Specifies hyperelastic (rubber-like) material properties for nonlinear analysis. Use SOL700 only.

**Format 1 (default): Mooney-Rivlin Model (Model=Mooney1)**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model			RHO				
	A	B	NU						

**Format 2: Ogden Model (Model=Ogden1)**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	MODEL			RHO				
	MU1	ALPHA1	NU						
	MU2	ALPHA2		MU3	ALPHA3				
	MU4	ALPHA4		MU5	ALPHA5				
	MU6	ALPHA6		MU7	ALPHA7				
	MU8	ALPHA8							

**Format 3: Foam Model (Model=FOAM1)**

1	2	3	4	5	6	7	8	9	10
MATHE	MID	Model		K	RHO				
	TYPE	VALUE	CUTOFF	TABLE	TABY				
	ALPHA	UNLOAD							

**Example –Format 1:**

MATHE	7	MOONEY1			100.				
	0.34	0.27	0.495						

**Example –Format 2:**

MATHE	7	OGDEN1			100.				
	6.3	1.3	0.495						
	0.12	5.0		-0.1	-2.0				

**Example –Format 3:**

MATHE	7	FOAM1		3.	100.				
	1.0	-100.0	1.0	111	112				
	0.4	2.0							

Descriptor	Contents
MID	Material identification number. (Integer > 0; required.)
Model	Select hyperelastic material model from (Character; Default = Mooney1) Mooney1 Mooney-Rivlin hyperelastic model. It is only available for solid elements. See Remark 1. Ogden1 Ogden hyperelastic model. It is only available for solid elements. See Remark 2. FOAM1 Isotropic elastic foam material. It is only available for solid elements. See Remarks 3.-8.
RHO	Density. (Real > 0; required)
A	Strain-energy density function constant. (Real; required)
B	Strain-energy density function constant. (Real; required)
NU	Poisson's ratio. ( $0.0 \leq R < 0.5$ ; required)
MU1-MU8	Strain energy density function coefficients. MU1 is required. (Real; default=0.0)
ALPHA1-ALPHA8	Strain energy density function coefficients. ALPHA1 is required. (Real; default=0.0)
TYPE	The type of data defined as the x-value in the table. (Real > 0.0; default=1.0) 1.0 Crush factor. (= 1-relative volume) 2.0 True strain
VALUE	The value for cut-off stress. (Real $\leq$ 0.0; default=-0.1 $\times$ Young's modulus)
CUTOFF	Cut-off stress. (Real > 0.0; default=3.0) 1.0 Stress for tensile failure 2.0 Minimum stress 3.0 Not used
TABLE	TABLED1 ID defining the variation of stress (y-value) with crush factor or true strain (x- value). (Integer > 0; required)
TABY	TABLED1 ID giving the variation of the scale factor for the stress (y-value) with the strain rate (x-value). (Integer > 0; default=not used)
ALPHA	Energy dissipation factor. ( $0.0 \leq \text{Real} \leq 1.0$ ; default=0.0)
UNLOAD	Unloading option. (Real > 0.0; default=3.0) 1.0 Unloading via exponential curve 2.0 Unloading via piecewise linear curve 3.0 Unloading via quadratic curve

Remarks:

1. The Mooney-Rivlin constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components rather than by Hooke's law.

The strain-energy density function is formulated according to the Mooney-Rivlin model and is

$$\text{defined as } W(I_1, I_2, I_3) = A(I_1 - 3) + B(I_2 - 3) + C\left(\frac{1}{I_3} - 1\right) + D(I_3 - 1)^2$$

$$C = \frac{1}{2}A + B$$

The constants C and D are defined as:

$$D = \frac{A(5\nu - 2) + B(11\nu - 5)}{2(1 - 2\nu)}$$

where  $A$ ,  $B$  and  $\nu$  are input parameters.

$I_1$ ,  $I_2$  and  $I_3$  are strain invariants in terms of stretches.

For a rubber-like material, the shear modulus  $G$  is much less than the bulk modulus  $K$ . As a result, Poisson's ratio is nearly equal to one half.

2. The constitutive behavior of this material is defined as a total stress/total strain relationship. The nonlinear elastic material response is formulated by a strain-energy density function for large strain components using Ogden function.

The strain-energy density function is formulated according to the Ogden model and is defined as

$$W = \sum_{i=1}^3 \sum_{j=1}^8 \frac{\mu_j}{\alpha_j} (\bar{\lambda}_i^{\alpha_j} - 1) + \frac{1}{2}K(J - 1)^2$$

The over bar (-) indicates that the volumetric effects have been eliminated from the principal stretches.

3. Poisson's ratio for FOAM1 model is effectively zero. Therefore, the shear and elastic moduli are calculated from  $K$ , the bulk modulus.
4. For this model, the stress-strain curve is independent of the experimental test performed to obtain the material data (uniaxial, shear, or volumetric). The most common test is the uniaxial compression test where the stress-strain characteristic can either be defined in terms of the amount of crush, which is minus the engineering strain, or in terms of the true strain. Since Poisson's ratio is effectively zero, the

amount of crush is defined as  $\left(1 - \frac{V}{V_0}\right)$ , with  $\left(\frac{V}{V_0}\right)$  as the relative volume, and the true strain is

$$\text{defined as } \int_{t_0}^t \frac{dV}{V} \text{ or } \ln\left(\frac{V}{V_0}\right)$$

The crush factor must be between 0 and 1. The true strain must always be negative and the stress positive (absolute value).

5. The yield surface in three-dimensional space is a sphere in principal stresses, and is defined by

$$\tau_{11}^2 + \tau_{22}^2 + \tau_{33}^2 = R_s^2$$

where the radius of the sphere,  $R_s$ , depends on the strains and strain rates as follows:

$$R_s = f_1(R_e)f_2(R_r)$$

$$\text{with } R_e = \varepsilon_{11}^2 + \varepsilon_{22}^2 + \varepsilon_{33}^2$$

$$\text{and } R_r = \dot{\varepsilon}_{11}^2 + \dot{\varepsilon}_{22}^2 + \dot{\varepsilon}_{33}^2$$

and  $f_1$  is the function supplied in the stress-strain table and  $f_2$  (if defined) is the function supplied in the factor-strain rate table.

6. A minimum (CUTOFF=2.0) or failure (CUTOFF=1.0) tensile stress can be defined. In the first case this corresponds to a tensile cut-off where the stress cannot fall below the minimum value. In the second case, if the stress falls below the failure stress the element fails and cannot carry tensile loading for the remainder of the analysis. Thus the stress can never become negative again.
7. The unloading behavior is piecewise linear (UNLOAD=2.0), quadratic (UNLOAD=3.0) or exponential (UNLOAD=1.0). The unloading curve is constructed such that the ratio of the dissipated energy (area between compressive loading and unloading curve) to total energy (area under the loading curve) is equal to the energy dissipation factor alpha. In the case of piecewise linear unloading, Nastran SOL700 constructs an unloading curve whose segments are parallel to the supplied compression table, except for the first and last segments, which pass respectively through the origin and the point P on the compression curve where the unloading starts. In the case of quadratic unloading, Nastran SOL700 constructs a quadratic curve starting in the origin and ending in point P. If the quadratic unloading curve falls below the strain axis, then the unloading stress is set to zero. In the case of exponential unloading, the unloading curve is constructed in a similarly to quadratic unloading except for the shape of the curve, which is created from an exponential function instead of a quadratic polynomial.
8. Not default values for taby, alpha, unload options are allowed only when cutoff is set to 1.0 or 2.0.



**MATHED****Damage Model Properties for Hyperelastic Materials - SOL 600**

Specifies damage model properties for hyperelastic materials to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATHED	MID	method	Scale1	Relax1	Prop1	Scale2	Relax2	Prop2	
	Dinf	N/A	Scalem1	Relaxm1	N/A	Scalem2	Relaxm2		

Example:

MATHED	100		1.	0.3	0.5	0.8	0.4	1.0	
--------	-----	--	----	-----	-----	-----	-----	-----	--

Describer	Meaning
MID	Identification number of MATHE entry. See Remark 1. (Integer > 0)
Method	Select a method for damage calculation (Character) from: <b>Multip</b> for multiplicative decomposition. (Default) <b>Additiv</b> for additive decomposition. See Remarks 2. and 3.
Scale1,2	Scaling factor $d_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0.0$ ; < 1.0; Default = 0.0)
Relax1,2	Relaxation rate $\eta_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0$ or blank)
Prop1,2	Proportionality factor $\delta_n$ for n=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. See Remark 4. (Real > 0.0; Default = 1.0)
Dinf	$d^\infty$ described in the equations of Remark 3. If Blank, the program will compute it; however even if set, the program usually calculates it. In most cases $d^\infty = 1.0$ -scale1-scale2. (Real > 0.0 or blank)
Scalem1,2	Scaling factor $d_m$ for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real $\geq 0.0$ ; Default = 0.0)
Relaxm1,2	Relaxation rate $\lambda_m$ for m=1 or 2 in Kachanov factor $K(\alpha, \beta)$ described in Remark 3. (Real > 0.0 or blank)

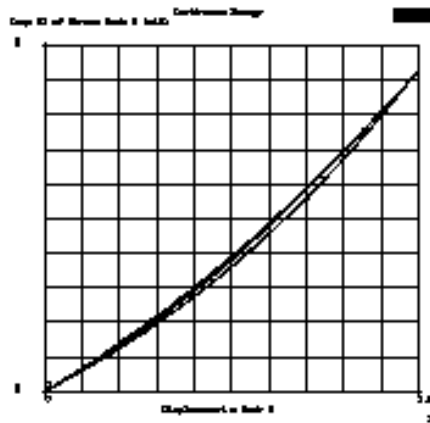
Remarks:

- The MATHE Bulk Data entry with the same MID must exist for MATHED to be effective.  
The damage capability is available for all the elastomeric materials (Mooney-Rivlin, Ogden, Gent, Arruda-Boyce).

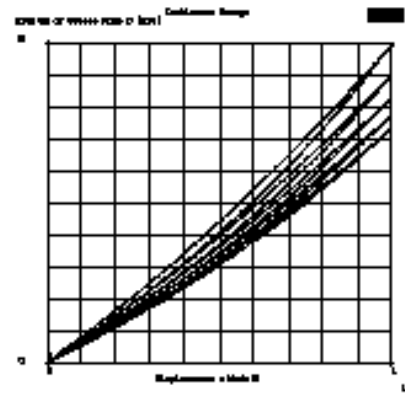
2. Under repeated application of loads, elastomers undergo damage by mechanisms involving chain breakage, multi-chain damage, micro-void formation, and micro-structural degradation due to detachment of filler particles from the network entanglement. The damage model for elastomeric materials is based on the undamaged strain energy function  $W_0$ , multiplied by a Kachanov damage factor,  $K$ , i.e.,

$$W = K(\alpha, \beta)W_0$$

where  $\alpha$  and  $\beta$  are parameters for discontinuous and continuous damage models, respectively. Two types of phenomenological models, discontinuous and continuous, exist to simulate the damage. The discontinuous damage model simulates the “Mullins’ effect,” which involves a loss of stiffness (represented by a parameter  $\alpha$ ) below the previously attained maximum strain. The higher the maximum attained strain the larger the loss of stiffness is. There is a progressive stiffness loss with increasing maximum strain amplitude. Most of the stiffness loss takes place in the first few cycles provided the maximum strain level is not increased. This phenomenon is observed in both filled as well as natural rubber although the higher level of carbon black particles increases the hysteresis and the loss of stiffness. The continuous damage model (Miehe’s formulation) can simulate the damage accumulation for strain cycles for which the values of effective energy is below the maximum attained value of the past history. The evolution of continuous damage parameter is governed by the arc-length of the effective strain energy, represented by a parameter  $\beta$ .



Discontinuous Damage



Continuous Damage

3. Both the continuous damage as well as the discontinuous damage can be modeled by a cumulative Kachanov factor in multiplicative or additive decomposition form.

For multiplicative decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n \exp\left(-\frac{\alpha + \delta_n \beta}{\eta_n}\right)$$

For additive decomposition

$$K(\alpha, \beta) = d^\infty + \sum_{n=1}^2 d_n^\alpha \exp\left(-\frac{\alpha}{\eta_n}\right) + \sum_{m=1}^2 d_m^\beta \exp\left(-\frac{\beta}{\lambda_m}\right)$$

where  $d_n$ ,  $\delta_n$ ,  $\eta_n$ ,  $d_m$ , and  $\lambda_m$  are constants specified by the user, and  $d^\infty$  is calculated by the program such that the Kachanov factor assumes a value of unity at zero damage if left blank.

4. The proportionality factor is not used by additive decomposition which requires the continuation fields to include the continuous damage model.
5. User subroutines must be called out using PARAM,MARCUSUB,CHAR where CHAR is a character variable such as UDAMAG.

**Note:**

1. This Bulk Data entry accommodates Marc's input data under the model definition option DAMAGE.

## MATHP

## Hyperelastic Material Properties

Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).

Format:

1	2	3	4	5	6	7	8	9	10
MATHP	MID	A10	A01	D1	RHO	AV	TREF	GE	
		NA	ND						
	A20	A11	A02	D2					
	A30	A21	A12	A03	D3				
	A40	A31	A22	A13	A04	D4			
	A50	A41	A32	A23	A14	A05	D5		
	TAB1	TAB2	TAB3	TAB4				TABD	

Example:

MATHP	2	80.	20						
		1	1						

Describer	Meaning
MID	Identification number of a MATHP entry. (Integer > 0; No Default)
Aij	Material constants related to distortional deformation. (Real; Default = 0.0)
Di	Material constants related to volumetric deformation. (Real ≥ 0; Default for D1 is $10^3 \cdot (A10 + A01)$ ; Default for D2 through D5 is 0.0.)
RHO	Mass density in original configuration. (Real; Default = 0.0)
AV	Coefficient of volumetric thermal expansion. (Real; Default = 0.0)
TREF	Reference temperature. See <a href="#">MATBV, 2172</a> . (Real; Default = 0.0)
GE	Structural damping element coefficient. (Real; Default = 0.0)
NA	Order of the distortional strain energy polynomial function. (0 < Integer ≤ 5; Default = 1)
ND	Order of the volumetric strain energy polynomial function. (0 < Integer ≤ 5; Default = 1)
TAB1	Table identification number of a TABLES1 entry that contains simple tension/compression data to be used in the estimation of the material constants Aij. xi values in the TABLES1 entry must be stretch ratios $l/l_0$ and yi values must be values of the engineering stress $F/A_0$ . Stresses are negative for compression and positive for tension. (Integer > 0 or blank)

Describer	Meaning
TAB2	Table identification number of a TABLES1 entry that contains equibiaxial tension data to be used in the estimation of the material constants $A_{ij}$ . $x_i$ values in the TABLES1 entry must be stretch ratios $l/l_0$ . $y_i$ values must be values of the engineering stress $F/A_0$ . $l$ is the current length, $F$ is the current force, $l_0$ is the initial length and $A_0$ is the cross-sectional area. In the case of pressure of a spherical membrane, the engineering stress is given by $Pr_0\lambda^2/2t_0$ where $P$ = current value of the pressure and $r_0, t_0$ = initial radius and thickness. (Integer > 0 or blank)
TAB3	Table identification number of a TABLES1 entry that contains simple shear data to be used in the estimation of the material constants $A_{ij}$ . $x_i$ values in the TABLES1 entry must be values of the shear tangent $\gamma$ and $y_i$ values must be values of the engineering shear stress $F/A_0$ . (Integer > 0 or blank)
TAB4	Table identification number of a TABLES1 entry that contains pure shear data to be used in the estimation of the material constants $A_{ij}$ . $x_i$ and $y_i$ values in the TABLES1 entry must be stretch ratios $\lambda_1 = l/l_0$ and values of the nominal stress $F/A_0$ . $l$ is the current length, $F$ is the current force, $l_0$ and $A_0$ are the initial length and cross-sectional area, respectively in the 1-direction. (Integer > 0 or blank)
TABD	Table identification number of a TABLES1 entry that contains pure volumetric compression data to be used in the estimation of the material constants $D_i$ . $x_i$ values in the TABLES1 entry must be values of the volume ratio $J = \lambda^3$ where $\lambda = l/l_0$ is the stretch ratio in all three directions; $y_i$ values must be values of the pressure, assumed positive in compression. (Integer > 0 or blank)

Remarks:

1. The generalized Mooney-Rivlin strain energy function may be expressed as follows:

$$U(J, \bar{I}_1, \bar{I}_2) = \sum_{i+j=1}^{NA} A_{ij}(\bar{I}_1 - 3)^i(\bar{I}_2 - 3)^j + \sum_{i=1}^{ND} D_i(J - 1 - AV(T - T_0))^{2i}$$

$$A_{00} = 0$$

where  $\bar{I}_1$  and  $\bar{I}_2$  are the first and second distortional strain invariants, respectively;  $J = \det F$  is the determinant of the deformation gradient; For small strains, the bulk modulus  $K$  and the shear modulus  $G$  are related to the Mooney-Rivlin Material.

$2D1 = K$  and  $2(A10 + A01) = G$  at small strains, in which  $K$  is the bulk modulus and  $G$  is the shear modulus. The model reduces to a Mooney-Rivlin material if  $NA=1$  and to a Neo-Hookean material if  $NA = 1$  and  $A01 = 0.0$ . See Remark 2. For Neo-Hookean or Mooney-Rivlin materials no continuation entry is required.  $T$  is the current temperature and  $T_0$  is the initial temperature.

2. Conventional Mooney-Rivlin and Neo-Hookean materials are fully incompressible. Full incompressibility is not presently available but may be simulated with a large enough value of D1. Note that a value of D1 lower than  $10^3 \cdot (A10 + A01)$  is, however, not recommended.
3. The material constants  $A_{ij}$  and  $D_i$  are obtained from least squares fitting of experimental data. One or more of four experiments (TAB1 to TAB4) may be used to obtain constants for distortional properties  $A_{ij}$ .  $D_i$  may be obtained from pure volumetric compression data (TABD). If all TAB1 through TAB4 are blank,  $A_{ij}$  must be specified by the user. Parameter estimation, specified through any of the TABLES1 entries, supercedes the manual input of the parameters.
4. IF ND=1 and a nonzero value of D1 is provided or obtained from experimental data in TABD, then the parameter estimation of the material constants  $A_{ij}$  takes compressibility into account in the cases of simple tension/compression, equibiaxial tension, and general biaxial deformation. Otherwise, full incompressibility is assumed in estimating the material constants.
5. See Chapters [Hyperelastic Elements](#), [Hyperelastic Material](#) and [Hyperelastic Material](#) in the *MSC Nastran Reference Guide* for further details.
6. In SOL400, MATHP is supported for 2 sets of elements:
  - a. Incompressible conventional 2D (plane strain and axisymmetric) elements and 3D (continuum) elements without property extensions. They are prescribed through PLPLANE and PLSOLID, respectively. The original element formulations that are also available in SOL106 are used in these cases.
  - b. Incompressible elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for incompressible rubber models, 2D 3-node triangular elements and 3D 4-node tetrahedral elements should be associated with an incompressible formulation by specifying IPS for 2D plane strain and IAX for 2D axisymmetric elements in PSHLN2, as well ISOL for 3D tetrahedral element in PSLDN1.
7. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**Note:**

1. PLSOLID is only used with the elements in (a) and only PSOLID with PSLDN1 can be used for 3D solid nonlinear elements in (b).
2. Prior to MD Nastran 2010, if the property extensions were missing, the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

## MATNLE

## Advanced NLELAST Options - SOL 600

The MATNLEx entries specify advanced forms of nonlinear elastic materials. The simple NLELAST option uses MAT1, MATS1, TALBES1 in the same manner as data is entered for SOL 106 or SOL 129.

## Format 1: Standard Nastran NLELAST Model

Use NLELAST on MATS1 entry.

## Format 2: Invariant-based Material Model

1	2	3	4	5	6	7	8	9	10
MATNLE2	MID	E	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

## Format 3: Principal Strain-Based Material Model

1	2	3	4	5	6	7	8	9	10
MATNLE3	MID	REF	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

## Format 4: Linear Elasticity with Tension and/or Compression Limits

1	2	3	4	5	6	7	8	9	10
MATNLE4	MID	REF	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				

## Format 5: Bi-Modulus Linear Elasticity with Tension/Compression Limits

1	2	3	4	5	6	7	8	9	10
MATNLE5	MID	E	PR	RHO	ALPH	G	TLIM	CLIM	
		TE	TPR	TRHO	TALPH	TG	TTLIM	TCLIM	
	KIND	TYPE	LIM	GE	NAME				
	EC	PRC	TEC	TPRC					

## Format 6: Orthotropic Nonlinear Elasticity based upon strains in local directions

1	2	3	4	5	6	7	8	9	10
MATNLE6	MID	E11	E22	E33	PR12	PR23	PR31	RHO	
		TE11	TE22	TE33	TPR12	TPR23	TPR31	TRHO	
	G12	G23	G31	A11	A22	A33			

	TG12	TG23	TG31	TA11	TA22	TA33			
	KIND	TYPE	LIM	GE	NAME				

Example - Format 3:

MATNLE3	2	2.90E+03	0.49	1.33E-04	2.60E-06		999999	999999	
+									
+		1	1	0	MATNLE2A				

Example - Format 4:

MATNLE4	2	2.90E+03	0.49	1.33E-04	2.60E-06		300	25	
+									
+		1	1	0	MATNLE4A				

Example - Format 5:

MATNLE5	2	2.90E+03	0.49	1.33E-04	2.60E-06		300	25	
+									
+		1	1	0	MATNLE4A				

Example - Format 6:

MATNLE6	2	2.90E+03	2.80E+03	0.491	0.492	0.493	1.00E-04	999999	
+									
+	6.00E+01	5.00E+01	4.00E+01	2.00E-06	1.50E-06	1.00E-06			
+									
+		1	1	0	MATNLE6A				

Describer	Meaning
MID	Identification number of a MATNLEi entry. (Integer > 0; no Default)
E	Young's modulus. (Real; no Default)
PR	Poisson's ratio. (Real; no Default)
RHO	Mass density. (Real; no Default)
ALPH	Coefficient of thermal expansion. (Real; no Default)
G	Shear Modulus. (Real; no Default)
TLIM	Tensile stress limit. (Real; no Default)
CLIM	Compressive stress limit. (Real; no Default)
TE	ID of TABL3Di entry for E vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)



Describer	Meaning
TPR	ID of TABL3Di entry for PR vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for density vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TALPH	ID of TABL3Di entry for ALPH vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG	ID of TABL3Di entry for G vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TTLIM	ID of TABL3Di entry for TLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TCLIM	ID of TABL3Di entry for CLIM vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
KIND	Tension/compression usage flag. (Integer; Default = 1) 1 = Both tension and compression data will be used 2 = Only tension data will be used
TYPE	Poisson ratio/Bulk Modulus flag. (Integer; Default = 1) 1 = Constant Poisson's ratio will be used 2 = Constant bulk modulus will be used
LIM	Stress limit flag. (Integer; Default = 0) 0 = Material has no tension or compression limits 1 = Material has tension limits 2 = Material has compression limits 3 = Material has both tension and compression limits
GE	Structural damping coefficient. (Real; Default = 0.0)
NAME	Material name. (Character; no Default) Optional name of the material, may be blank or as long as 16 characters.
EC	Young's modulus in compression. (Real; no Default)
PRC	Poisson's ratio in compression. (Real; no Default)
TEC	ID of TABL3Di entry for EC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPRC	ID of TABL3Di entry for PRC vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
E11	Orthotropic modulus E11. (Real; no Default)
E22	Orthotropic modulus E22. (Real; no Default)
E33	Orthotropic modulus E33. (Real; no Default)

Describer	Meaning
PR12	Orthotropic Poisson's ratio in 12 direction. (Real; no Default)
PR23	Orthotropic Poisson's ratio in 23 direction. (Real; no Default)
PR31	Orthotropic Poisson's ratio in 31 direction. (Real; no Default)
TE11	ID of TABL3Di entry for E11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TE22	ID of TABL3Di entry for E22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TE33	ID of TABL3Di entry for E33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR12	ID of TABL3Di entry for PR12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR23	ID of TABL3Di entry for PR23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TPR31	ID of TABL3Di entry for PR31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
G12	Orthotropic shear modulus G12. (Real; no Default)
G23	Orthotropic shear modulus G23. (Real; no Default)
G31	Orthotropic shear modulus G31. (Real; no Default)
A11	Orthotropic coefficient of thermal expansion in 1 direction. (Real; no Default)
A22	Orthotropic coefficient of thermal expansion in 2 direction. (Real; no Default)
A33	Orthotropic coefficient of thermal expansion in 3 direction. (Real; no Default)
TG12	ID of TABL3Di entry for G12 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG23	ID of TABL3Di entry for G23 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TG31	ID of TABL3Di entry for G31 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA11	ID of TABL3Di entry for A11 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA22	ID of TABL3Di entry for A22 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)
TA33	ID of TABL3Di entry for A33 vs up to 4 variables such as strain, temperature, strain rate, ... (Integer; no Default; leave blank if table is not required)

Remarks:

1. The structural damping constant GE is specified in MARC under the option DAMPING as a numerical damping  $\gamma$ ,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE;  $\alpha$  and  $\beta$  are equivalent to user parameters ALPHA1 and ALPHA2.

2. This Bulk Data entry accommodates MARC's input data under the model definition option NLELAST for material model types 2-6.
3. 3D tables associated with this entry should normally select independent variable 69 (equivalent mechanical strain) or 73 (any strain component) to define the stress-strain curve. Normally the only other independent 3D table variables used with this entry are 12 (temperature) or 16 (equivalent strain rate).
4. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

## MATORT

## 3D Orthotropic Material Properties

Specifies orthotropic material properties. Parameters for both isotropic and anisotropic yield conditions can also be specified along with elastic material properties. Can be specified for all 3-D and 2-D continuum elements and shells in SOL400/600/700. Can also be specified with solid composite, solid shell composite and pyramid elements in all linear solution sequences between SOL101 and SOL112, and analysis only for SOL200.

Format:

1	2	3	4	5	6	7	8	9	10
MATORT	MID	E1	E2	E3	NU12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	
	IYLD	IHARD	SY		R11 or m	R22 or C1	R33 or C2	N/A	
	R12 or C3	R23 or C6	R31	N/A	N/A	N/A	N/A	N/A	
	OPTION	FILE	X1	Y1	Z1	X2	Y2	Z2	

Example:

MATORT	100	3.e6	2.8e7	1.5e5	0.25				
--------	-----	------	-------	-------	------	--	--	--	--

Describer	Meaning
MID	Identification number of a MATORT entry. (Integer > 0; No Default)
E1	Modulus of elasticity in longitudinal or 1-direction. See Remark 1. (Real > 0)
E2	Modulus of elasticity in lateral direction or 2-direction. (Real > 0; no Default; must be entered)
E3	Modulus of elasticity in thickness direction or 3-direction. (Real > 0; no Default; must be entered)
NU12	Poisson's ratio $\varepsilon_2/\varepsilon_1$ for uniaxial loading in 1-direction. See Remark 2. (Real; no Default; must be entered)
NU23	Poisson's ratio $\varepsilon_3/\varepsilon_2$ for uniaxial loading in 2-direction. (Real; no Default; must be entered)
NU31	Poisson's ratio $\varepsilon_1/\varepsilon_3$ for uniaxial loading in 3-direction. (Real; Default = NU23)
RHO	Mass density. (Real; Default = 0.0)
G12	Shear modulus in plane 1-2. See Remark 3. (Real > 0; no Default; must be entered)
G23	Shear modulus in plane 2-3. (Real > 0.; no Default; must be entered)
G31	Shear modulus in plane 3-1. (Real > 0; no Default; must be entered)
Ai	Coefficient of thermal expansion in i-direction (Real; Default = 0.0).

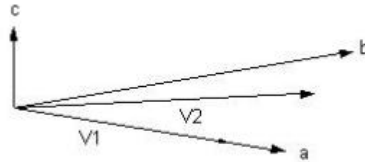
Describer	Meaning
TREF	Reference temperature at which the thermal expansion coefficient is measured. TREF is used only if the thermal expansion coefficient is temperature-dependent. (Real; Default = 0.0)
GE	Structural damping coefficient. (Real; Default = 0.0).
IYLD	Integer pertaining to one of the following yield criterion: (Integer)  -1 = Elastic 1 = von Mises (Default) 2 = Hill (1948) yield 3 = Barlat (1991) yield 4 = Viscoplasticity through user subroutine UVSCPL
IHARD	Integer pertaining to one of the following work hardening rules: (Integer) 1 = Isotropic (Default) 2 = Kinematic 3 = Combined Isotropic/Kinematic
SY	Equivalent (von Mises) tensile yield stress. (Real > 0 or blank; Default=10 <sup>20</sup> )
Rij	Stress ratios of initial yield stresses in various material directions to the reference yield stress. (Real > 0.0 or blank; Default = 1.0 for R11, R22, R33, R12, R23 and R31). See Hill's yield function in <i>MSC Nastran Nonlinear User's Guide</i> (SOL 400) for details.
m	Barlat exponent (Real > 0.0 or blank; Default = 2.0).
Ci	Barlat material coefficient (Real > 0.0 or blank; Default = 1.0 for C1, C2, C3 and C6). See Balart's yield function in <i>MSC Nastran Nonlinear User's Guide</i> (SOL 400) for details.

Describer	Meaning
-----------	---------

OPTION Material axes option used to determine how the local material axis system is defined. See Remark 7. (Character; default=ELEM)

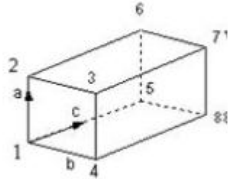
VECT

Globally orthotropic with the material axes defined by two vectors V1 and V2, specified using the fields X1, Y1, Z1 and X2, Y2, Z2. The a-axis is defined by the first vector. The b- and c-axes are then defined as:



ELEM

Globally orthotropic material with the material axes defined by element topology. Grid point 1 defines the origin, grid point 5 lies on the c-axis, and grid point 2 lies in the a-c plane. The a, b, and c axes are defined as follows:



ELMAT

Orthotropic material properties and the material coordinate system is defined by the element. The material properties are read from a formatted file. The filename is specified in the FILE field adjacent to OPTION.

Format of material properties file:

```
Record#
EID, DENSITY, DUMMY, DUMMY, DUMMY,
E1, E2, E3, G12, G23, G31,
NU12, NU13, NU23, NU21, NU31, NU32
```

ELPROP

Globally orthotropic material with the material axes defined by element topology (see also ELEM). The elasticity matrix is available per element.

Describer	Meaning
FILE	Material file name (OPTION=ELMAT only). See Remark 8.  (Character; default=blank)
X1, Y1, Z1	Components of the vector V1 in the basic coordinate system. See Remark 8. (Real; default=0.0)
X2, Y2, Z2	Components of the vector V2 in the basic coordinate system. See Remark 8. (Real; default=0.0)

### Remarks:

1. All the material constants are specified in the orthotropic material coordinates in 1, 2, and 3 direction.
2. In general,  $v_{12}$  is not the same as  $v_{21}$ , but they are related by  $v_{ij}/E_i = v_{ji}/E_j$ . Furthermore, material stability requires that

$$E_i > v_{ij}^2 E_j$$

$$\text{and } 1 - v_{12}v_{21} - v_{23}v_{32} - v_{31}v_{13} - 2v_{21}v_{32}v_{13} > 0.$$

3. It may be difficult to find all nine orthotropic constants. In some practical problems, the material properties may be reduced to normal anisotropy in which the material is isotropic in a plane, e.g., in plane 1-2 and has different properties in the direction normal to the plane 1-2. In the plane of isotropy, the properties are reduced to

$$E_1 = E_2 = E_p$$

$$v_{31} = v_{32} = v_{np}$$

$$v_{13} = v_{23} = v_{pn}$$

$$G_{13} = G_{23} = G_n$$

$$\text{with } v_{np}/E_n = v_{pn}/E_p \quad \text{and } G_p = \frac{E_p}{2(1 + \nu_p)}.$$

There are five independent material constants for normal anisotropy (e.g.,  $E_p$ ,  $E_n$ ,  $\nu_p$ ,  $\nu_{np}$ ,  $G_n$ ).

In case the material has a planar anisotropy, in which the material is orthotropic only in a plane, the elastic constants are reduced to seven, e.g.,  $E_1$ ,  $E_2$ ,  $E_3$ ,  $\nu_{12}$ ,  $G_{12}$ ,  $G_{23}$ ,  $G_{31}$ .

4. Do not enter values for SY, Rij, m and Ci unless plasticity is to be taken into account.
5. For elastoplastic case, MATORT only support perfect plasticity.
6. MATEP option is the recommended approach for elastoplastic materials.

7. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
8. OPTION, FILE, X1, Y1, Z1, X2, Y2 and Z2 fields are only supported by new SOL700.
9. New SOL700 based on Dytran does not support A1, A2, A3, TREF, GE, IYLD, IHARD, SY, Rij, m and Ci fields.

#### Notes:

1. The structural damping constant GE is specified as a numerical damping  $\gamma$ , i.e.,

$$C = \alpha M + \left( \beta + \frac{2\gamma}{\omega} \right) K$$

in which  $2\gamma$  is equivalent to GE.

2. This Bulk Data entry accommodates Marc's input data under the model definition option ORTHOTROPIC.



## MATPE1 Isotropic Poroelastic Material Property definition

Defines the material properties for an isotropic poroelastic material.

Format:

1	2	3	4	5	6	7	8	9	10
MATPE1	MID	MAT1	MAT10	BIOT	POROPT	SRHO			
	VISC	GAMMA	PRANDTL	POR	TOR	AFR	VLE	TLE	

Example:

MATPE1	101	1	10						
	1.84-8	1.4	7.13-1	9.5-1	1.4	2.5-5	9.32-2	9.32-2	

Describer	Meaning
MID	Material identification number, referenced by PSOLID entry. (Integer>0)
MAT1	Identification number of MAT1 (and MATF1 if it is frequency-dependent) entry for the skeleton (solid-phase) of porous material. (Integer>0, Required)
MAT10	Identification number of MAT10 entry for the fluid-phase of porous material. (Integer>0, Required)
BIOT	Biot factor. (Real>0.0, Default=1.0)
POROPT	Simplified porous options: LUMPED, RIGID, MIKI or DELANY
SRHO	Solid density for POROPT=LUMPED (Real≥0.0)
VISC	Fluid dynamic viscosity. See Remark 4. (Real>0.0, Required except for POROPT = "DELANY" or "MIKI")
GAMMA	Fluid ratio of specific heats. See Remark 5. (Real>0.0, Default=1.402)
PRANDTL	Fluid Prandtl number, Pr. See Remark 6. (Real>0.0, Default=0.71)
POR	Porosity of porous material. (Real>0.0, Required except for POROPT = "DELANY" or "MIKI")
TOR	Tortuosity of porous material. (Real>0.0, Default=1.0)
AFR	Air flow resistivity. (Real>0.0, Required)
VLE	Viscous characteristic length. (Real≥0.0, Default=0.0)
TLE	Thermal characteristic length. (Real≥0.0, Default=0.0)

### Remarks:

1. MATPE1 entry is referenced by PSOLID cards when FCTN=PORO. It is used to define the isotropic poroelastic material properties for both solid and fluid phases of a poroelastic medium.
2. MATPE1 entry must be put under BEGIN BULK TRMC.

3. For a frequency-dependent skeleton material, MATF1 specifies the material properties at frequencies on matching MAT1 fields.
4. Fluid dynamic viscosity, VISC, is temperature dependent. At 15°C (Celsius), its value is  $1.82 \times 10^{-6}$  N.s/m<sup>2</sup> in the SI units.
5. GAMMA is defined as  $C_p/C_v$ , where  $C_p$  is the specific heat at constant pressure and  $C_v$  the specific heat at constant volume. Its default value is 1.402 for the air at ambient temperature.
6. PRANDTL number, Pr, is defined as  $\mu C_p/k$  where  $\mu$  is the fluid dynamic viscosity (VISC) and  $k$  the thermal conductivity. For the dry air at ambient temperature, its value is around 0.7-0.8.
7. Porosity,  $\Omega$ , is defined as  $V_f/V_t$  where  $V_f$  is the volume of fluid and  $V_t$  is the total volume of the poroelastic material. Hence, porosity is dimensionless.
8. Tortuosity measures the complexity of the path an air particle must follow to process from one point to another point. It is a dimensionless parameter.
9. Resistivity measures the resistance of poroelastic material against the flow of fluid. In SI unit, it is measured in N.s/m<sup>4</sup>.
10. The density of poroelastic material is defined as  $\rho_{pem} = \Omega\rho_f + (1 - \Omega)\rho_s$  where  $\rho_f$  is the fluid density from MAT10 and  $\rho_s$  is the solid density from MAT1.
11. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
12. Assumptions for simplified BIOT porous material model are as follows:
 

LUMPED	the material skeleton can be assumed as extremely soft (E=0)
RIGID	the material skeleton can be assumed as rigid
MIKI or DELANY	Semi-empirical numerical mode; assuming porosity=1
13. MAT1 field should be left blank for POROPT=RIGID, MIKI or DELANY.
14. For POROPT=LUMPED, solid density can be provided either from MAT1 ID in MAT1 field or directly from SRHO field of MATPE1.

## MATRIG

## Rigid Material

Defines the properties of a rigid material in SOL 700.

Format:

1	2	3	4	5	6	7	8	9	10
MATRIG	MID	RHO	E	NU	MASS	XC	YC	ZC	
	IXX	IXY	IXZ	IYY	IYZ	IZZ	CID		
	VX	VY	VZ	WX	WY	WZ			
	XC-LOCAL	YC-LOCAL	ZC-LOCAL						

Example:

MATRIG	7	7850.	210.E9	0.3	750	0.0	7.0	-3.0	
	17.0	13.2	14.3	20.9	15.7	10.0	12		
			13.3						

Describer	Meaning	Type	Default
MID	Unique material number.	$I > 0$	Required
RHO	Density	$R > 0$	1.0
E	Young's modulus	$R > 0$	1.0
NU	Poisson's ratio	$0.0 < R < 0.5$	0.2
MASS	Mass of the rigid body.	$R > 0.0$	See Remark 2.
XC, YC, ZC	x, y, and z coordinates of the center of gravity.	R	See Remark 6.
IXX, IXY, IXZ, IYY, IYZ, IZZ	Inertia tensor of the rigid body about the center of gravity.	R	See Remark 6.
CID	Number of a coordinate system in which the inertia tensor and the center of gravity are defined.	$I > 0$	See Remarks 7. and 8.
VX, VY, VZ	Initial translational velocity of the center of gravity in the basic coordinate system.	R	0.0
WX, WY, WZ	Initial rotational velocities of the rigid body about the center of gravity in the basic coordinate system.	R	0.0
XC-LOCAL YC-LOCAL ZC-LOCAL	x, y, and z local coordinates of the center of gravity	R	See Remark 8.

Remarks:

1. All coordinates are defined in the basic coordinate system.

2. If MASS is blank or zero, the mass will be calculated from the density and the geometry of the mesh defining the rigid body.
3. The continuation lines are not required.
4. The MATRIG definition is used instead of a material definition and is referenced by properties PSOLIDn, PSHELLn, PBAR, and PBEAMn. Different properties can refer to the same MATRIG entry forming one rigid body.
5. If the fields VX, VY, VZ, WX, WY, and WZ are blank, then the initial conditions of the rigid body are calculated from the initial velocities on the TIC and TIC1 entries referring to grid points attached to the rigid body. The net initial conditions are the average of those for all the grid points attached to the rigid body.  
If the initial conditions are set using the VX, VY, VZ, WX, WY, and WZ fields, the TIC and TIC1 entries referring to grid points attached to the rigid body are ignored.
6. If the inertia tensor or the coordinates of the center of gravity are undefined, then they will be computed from the density or mass and the geometry of the mesh defining the rigid body.
7. The inertia tensor can only be defined in a local rectangular coordinate system. If the entry for a local coordinate system is left blank, then the inertia tensor is defined in the global coordinate system.
8. The center of gravity can be defined in a local rectangular coordinate system (CID). However, XC YC ZC (x, y, and z coordinates of the center of gravity in the basic coordinate system) should be left blank when XC-LOCAL YC-LOCAL ZC-LOCAL (x, y, and z coordinates of the center of gravity in a local coordinate system) defined.

## MATS1 Material Stress Dependence

Specifies stress-dependent material properties for use in applications involving nonlinear materials. This entry is used if a MAT1 entry is specified with the same MID in a nonlinear solution sequence (SOLs 106, 129 and 400). For SOL 400 the MATEP option is the recommended approach for elastic-plastic materials.

Format:

1	2	3	4	5	6	7	8	9	10
MATS1	MID	TID	TYPE	H	YF	HR	LIMIT1	LIMIT2	

Example:

MATS1	17	28	PLASTIC	0.0	1	1	2.+4		
-------	----	----	---------	-----	---	---	------	--	--

Describer	Meaning
MID	Identification number of a MAT1 entry. (Integer > 0)
TID	Identification number of a TABLES1 or TABLEST entry. If H is given, then this field must be blank. See Remark 3. (Integer ≥ 0 or blank)
TYPE	Type of material nonlinearity. See Remarks. (Character: “NLELAST” for nonlinear elastic or “PLASTIC” for elastoplastic.)
H	Work hardening slope (slope of stress versus plastic strain) in units of stress. For elastic-perfectly plastic cases, H = 0.0. For more than a single slope in the plastic range, the stress-strain data must be supplied on a TABLES1 entry referenced by TID, and this field must be blank. See Remark 2. (Real)
YF	Yield function criterion, selected by one of the following values (Integer): <ul style="list-style-type: none"> <li>1 von Mises (Default)</li> <li>2 Tresca</li> <li>3 Mohr-Coulomb</li> <li>4 Drucker-Prager</li> </ul>
HR	Hardening Rule, selected by one of the following values (Integer): <ul style="list-style-type: none"> <li>1 Isotropic (Default)</li> <li>2 Kinematic</li> <li>3 Combined isotropic and kinematic hardening</li> </ul>
LIMIT1	Initial yield point. See Table 9-26. (Real)
LIMIT2	Internal friction angle, measured in degrees, for the Mohr-Coulomb and Drucker-Prager yield criterion. See Table 9-26. (0.0 ≤ Real < 45.0)

Table 9-26 Yield Functions Versus LIMIT1 and LIMIT2

Yield Function (YF)	LIMIT1	LIMIT2
von Mises (1) or Tresca (2)	Initial Yield Stress in Tension, $Y_1$	Not used
Mohr-Coulomb (3) or Drucker-Prager (4)	2* Cohesion, $2c$ (in units of stress)	Angle of Internal Friction $\phi$ (in degrees)

Remarks:

1. If TYPE = “NLELAST”, then MID may refer to a MAT1 entry only. Also, the stress-strain data given in the TABLES1 entry will be used to determine the stress for a given value of strain. The values H, YF, HR, LIMIT1, and LIMIT2 will not be used in this case.

Thermoelastic analysis with temperature-dependent material properties is available for linear and nonlinear elastic isotropic materials (TYPE = “NLELAST”) and linear elastic anisotropic materials. Four options of constitutive relations exist. The relations appear in Table 19 along with the required Bulk Data entries.

Table 19 Constitutive Relations and Required Material Property Entries

Constitutive Relation	Required Bulk Data Entries
$\{\sigma\} = [G_e(T)]\{\varepsilon\}$	MAT1 and MATT1
$\{\sigma\} = \frac{\bar{E}(\sigma, \varepsilon)}{E} [G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E} [G_e]\{\varepsilon\}$	MAT1, MATS1, TABLEST, and TABLES1
$\{\sigma\} = \frac{\bar{E}(T, \sigma, \varepsilon)}{E} [G_e(T)]\{\varepsilon\}$	MAT1, MATT1, MATS1, TABLEST, and TABLES1

In Table 19  $\{\sigma\}$  and  $\{\varepsilon\}$  are the stress and strain vectors,  $[G_e]$  the elasticity matrix,  $\bar{E}$  the effective elasticity modulus, and  $E$  the reference elasticity modulus.

2. If TYPE = “PLASTIC”, the elastic stress-strain matrix is computed from MAT1 entry, and then the isotropic plasticity theory is used to perform the plastic analysis. In this case, either the table identification TID or the work hardening slope H may be specified, but not both. If the TID is omitted, the work hardening slope H must be specified unless the material is perfectly plastic. The plasticity modulus ( $H$ ) is related to the tangential modulus ( $E_T$ ) by

$$H = \frac{E_T}{1 - \frac{E_T}{E}}$$

where  $E$  is the elastic modulus and  $E_T = dY/d\varepsilon$  is the slope of the uniaxial stress-strain curve in the plastic region. See Figure 9-99.

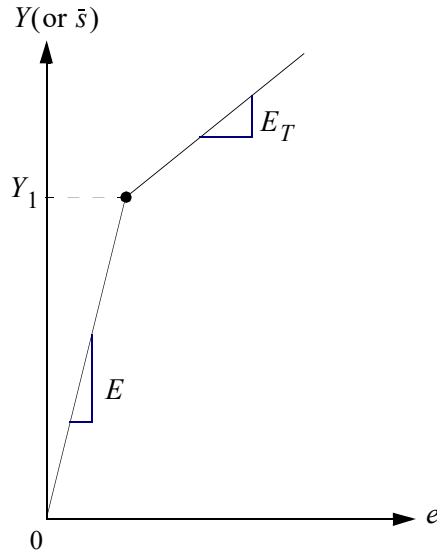


Figure 9-99 Stress-Strain Curve Definition When H Is Specified in Field 5

3. If TID is given, TABLES1 entries ( $X_i, Y_i$ ) of stress-strain data ( $\varepsilon_k, Y_k$ ) must conform to the following rules (see Figure 9-100):
  - If TYPE = "PLASTIC", the stress-total strain curve (i.e., TYPE in TABLES1 must be 1) must be defined in the first quadrant. The first point must be at the origin ( $X_1 = 0, Y_1 = 0$ ) and the second point ( $X_2, Y_2$ ) must be at the initial yield point ( $Y_1$  or  $2c$ ) specified on the MATS1 entry. The slope of the line joining the origin to the yield stress must be equal to the value of  $E$ . Also, TID may not reference a TABLEST entry.

The stress-total strain curve has 2 different definitions

- For conventional element, small strain formulation in terms of engineering strain and stress is used. Therefore, engineering stress-strain curve should be used here.
- For advanced nonlinear element, large strain formulation in terms of logarithmic strain and true stress is adopted. Then true stress-logarithmic strain curve should be used.

Note that SOL 106 and SOL 129 supports conventional element only. SOL 400 supports both of the conventional and advanced elements.

For TYPE = "PLASTIC", it should be noted that for conventional elements, strains should be below .05 to small strain formulation. Within this strain range, difference between logarithmic strain and engineering strain may be negligible. If the element strains exceed .05, then it is recommended to use the MATEP entry which will automatically switch to a large strain formulation in SOL 400

For SOL 600, if use of the MATS1 entry is desired, then it should be used in conjunction with PARAM,MRTABLS1,value.

- If TYPE = "NLELAST", the full stress-strain curve ( $-\infty < x < \infty$ ) may be defined in the first and the third quadrant to accommodate different uniaxial compression data. If the curve is defined only in the first quadrant, then the curve must start at the origin (X1 = 0.0, Y = 0.0) and the compression properties will be assumed identical to tension properties.

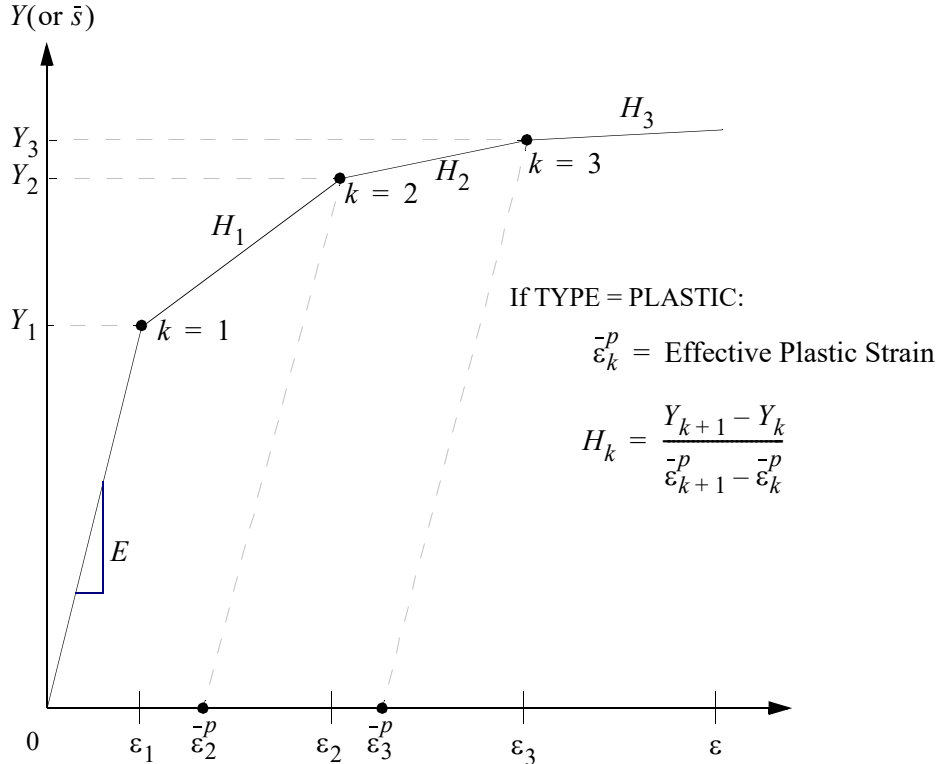


Figure 9-100 Stress-Strain Curve Definition When TID Is Specified in Field 3

4. In SOL 400, MATS1 is supported for 2 sets of elements:



- a. Elements with no property extensions that are prescribed through PROD, PBEAML, PSHELL and PSOLID. The original small-strain element formulations that are also available in SOL106 are used in these cases.
- b. Large-strain elements that are supported through property extensions. For such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Also, for TYPE = PLASTIC, linear 2D triangular elements specified on PSHLN2 and 3D tetrahedral elements specified on PSLDN1 should be associated with an incompressible formulation (IPS for 2D plane strain, IAX for 2D axisymmetric and ISOL for 3D tetrahedral. Note also that, prior to MD Nastran 2010, if the property extensions were missing, then for supported elements (i.e., PROD, PBEAML, PSHELL or PSOLID), the analysis would use the element formulations in (a), or ignore the MATS1 data for unsupported elements. From MD Nastran 2010 onwards, if the property extensions are missing, then they are automatically added with appropriate settings by the program unless the element belongs to group (a) or the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

**MATS3**      Advanced Orthotropic, Nonlinear Elastic Materials for Axisymmetric Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic materials at axisymmetric conditions in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATS3	MID	TEX	TETH	TEZ	TNUXTH	TNUTHZ	TNUZX	TRHO	
			TGZX	TAX	TATH	TAZ			

Example:

MATS3	33	56			67				
			12						

Describer	Meaning
MID	Identification number of a MAT3 entry. (Integer > 0; no Default)
TEX	ID of TABL3Di entry for EX. (Integer; no Default; leave blank if table is not required)
TETH	ID of TABL3Di entry for ETH. (Integer; no Default; leave blank if table is not required)
TEZ	ID of TABL3Di entry for EZ. (Integer; no Default; leave blank if table is not required)
TNUXTH	ID of TABL3Di entry for NUXTH. (Integer; no Default; leave blank if table is not required)
TNUTHZ	ID of TABL3Di entry for NUTHZ. (Integer; no Default; leave blank if table is not required)
TNUZX	ID of TABL3Di entry for NUZX. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TGZX	ID of TABL3Di entry for GZX. (Integer; no Default; leave blank if table is not required)
TAX	ID of TABL3Di entry for AX. (Integer; no Default; leave blank if table is not required)
TATH	ID of TABL3Di entry for ATH. (Integer; no Default; leave blank if table is not required)
TAZ	ID of TABL3Di entry for AZ. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc.

2. In SOL 400, MATS3 is used in conjunction with MAT3 and is only supported for axisymmetric elements with property extensions. This implies that for such elements, PLPLANE should be associated with PSHLN2. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS3 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

## MATS8 Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced orthotropic, nonlinear elastic material for plane stress and shell elements in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATS8	MID	TE1	TE2	TNU12	TG12	TG1Z	TG2Z	TRHO	
	TA1	TA2							

Example:

MATS8	34	12							
	1								

Describer	Meaning
MID	Identification number of a MAT8 entry. (Integer > 0; no Default)
TE1	ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)
TE2	ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)
TNU12	ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required)
TG12	ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)
TG1Z	ID of TABL3Di entry for G1Z. (Integer; no Default; leave blank if table is not required)
TG2Z	ID of TABL3Di entry for G2Z. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TA1	ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)
TA2	ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. As an example, say the analysis required TE1 to be a function of strain. Then the corresponding TABL3D0 entry is TABL3D0, KIND=73 to specify *local strain component* as the independent variable. The dependent variable is *E1*.
2. In SOL 400, MATS8 is used in conjunction with MAT8 and is only supported for shell elements with property extensions. This implies that for such elements, PSHELL should be associated with PSHLN1. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATS8 data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.

## MATSMA Material Properties for Shape Memory Alloys

For SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATSMA	MID	MODEL	$T_0$	L					
	$E_a$	$\nu_a$	$\alpha_a$	$\sigma_a$	$\rho_a$	$\sigma_s^A$	$\sigma_f^{AS}$	$C_a$	
	$E_m$	$\nu_m$	$\alpha_m$	$\sigma_m$	$\rho_m$	$\sigma_s^S$	$\sigma_f^{SA}$	$C_m$	
	$\nu^T$	$M_{frac}$	$\sigma_{eff}^g$	$\sigma_{max}^g$					
	$g_0$	$g_a$	$g_b$	$g_c$	$g_d$	$g_e$	$g_f$	$g_{max}$	

Example:

MATSMA	1	2	77.	0.05					
	50000.	0.33	0.00001	1.0E+20		520.	600.	8.66	
	50000.	0.33	0.00001	1.0e+20		300	200.	6.66	
	0.0	0.0	100.	1.0e+20					
	300.	-4.	2.	0.	0.	0.	3.0	1.0	

Describer	Meaning
MID	Material ID. (Integer > 0)
MODEL	Flag to indicate the model being used. (Integer > 0; see Remark 1.): = 1 (Mechanical: Aruchhio's model) = 2 (Thermo-Mechanical: Asaro-Sayeedvafa model)
$T_0$	Reference temperature used to measure stresses. (Real > 0)
L	For the mechanical model, the parameter L represents maximum deformation, obtainable by detwinning of multiple-variant martensite. (Real > 0; for the thermo-mechanical model, see Remark 2.) (typically 0.06-0.104)
Austenite Properties (all Real):	
$E_a$	Young's modulus of elasticity (typically 60-83 GPa).
$\nu_a$	Poisson's ratio (typically 0.33).
$\alpha_a$	Coefficient of thermal expansion (typically $3.67 \times 10^{-6}/^{\circ}\text{F}$ ).
$\sigma_a$	Equivalent von Mises stress (not used in mechanical model) (typically 195-690 MPa).

Describer	Meaning
$\rho_a$	Mass density.
$\sigma_s^{AS}$	Material parameter representing start of Austenite to Martensite transformation. For thermo-mechanical model, the program calculates the Austenite start temperature in the stress-free configuration ( $A_s^0$ ) related to $\sigma_s^{AS}$ from relations shown in <a href="#">Table 9-27</a> .
$\sigma_f^{AS}$	Material parameter representing finish of Austenite to Martensite transformation. For thermo-mechanical model, the program calculates the Austenite finish temperature in the stress-free configuration ( $A_f^0$ ) related to $\sigma_f^{AS}$ from relations shown in <a href="#">Table 9-27</a> .
$C_a$	Slope of the stress-dependence of austenite start finish and start temperatures (typically 6-8 MPa).
Martensite Properties (all Real):	
$E_m$	Young's modulus of elasticity
$\nu_m$	Poisson's ratio
$\alpha_m$	Coefficient of thermal expansion
$\sigma_m$	Equivalent von Mises stress (not used in mechanical model)
$\rho_m$	Density
$\sigma_s^{SA}$	Material parameter representing start of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite start temperature in the stress-free configuration ( $M_s^0$ ) related to $\sigma_s^{SA}$ from relations shown in <a href="#">Table 9-27</a> .
$\sigma_f^{SA}$	Material parameter representing finish of Martensite to Austenite transformation. For thermo-mechanical model, the program calculates the Martensite finish temperature in the stress-free configuration ( $M_f^0$ ) related to $\sigma_f^{SA}$ from relations shown in <a href="#">Table 9-27</a> .
$C_m$	Slope of the stress-dependence of austenite start finish and start temperatures. (typically 5-6 MPa)
The following quantities are applicable to only thermo-mechanical model (all Real):	
$\nu^T$	Equivalent volumetric transformation strain. (typically 0.0-0.003)
$M_{frac}$	Initial martensite volume fraction. (0.0-1.0)
$\sigma_{eff}^g$	Twinning stress (see Remark 2.). (typically 100-150 MPa)

Describer	Meaning
$\sigma_{\max}^g$	Stress at which the maximum value of $g = g_{\max}$ is reached if a cut-off value is needed (normalized with $g_0$ ).
$g_0$	Stress level used to nondimensionalizing the stress in the function. ( $2.0-10.0 * \sigma_{\text{eff}}^g$ )
$g_a$	g function coefficient (typically $g_a < 0.0$ )
$g_b$	g function exponent (typically $g_b = 2.0$ )
$g_c$	g function coefficient (typically $g_c \geq 0.0$ )
$g_d$	g function exponent (typically $g_d = 2.25 \sim 2.75$ )
$g_e$	g function coefficient (typically $g_e \leq 0.0$ )
$g_f$	g function exponent (typically $g_f = 3.0$ )
$g_{\max}$	Maximum value of function g if a cut-off value is needed (typically $g_{\max} = 1.0$ ).

Remarks:

1. The mechanical (Auricchio's) model can be obtained from the thermo-mechanical model by ignoring the last two rows in the input.
2. Twinning becomes active when the equivalent stress reaches twinning stress. For thermo-mechanical model, the “unstressed transformation temperatures” for Martensite and Austenite,  $M_s^0, M_f^0, A_s^0, A_f^0$  are calculated from the reference temperature, the material parameters representing the start and finish of the Austenite and Martensite transformations, i.e.,  $\sigma_s^{SA}, \sigma_f^{SA}, \sigma_s^{AS}, \sigma_f^{AS}$  as well as the coefficients,  $C_m, C_a$  that provide the stress dependence of the transformation temperatures as shown in [Table 9-27](#).

Table 9-27

The Relationship between Mechanical Model and Thermo-Mechanical Model
$\sigma_s^{AS} = (T_o - M_s^0)C_m$
$\sigma_f^{AS} = (T_o - M_f^0)C_m$
$\sigma_s^{SA} = (T_o - A_s^0)C_a$
$\sigma_f^{SA} = (T_o - A_f^0)C_a$

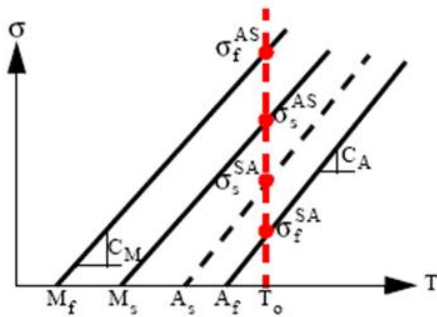


Figure 9-101

3. For the thermo-mechanical model, the equivalent deviatoric strain, (typically 0.05 - 0.085) is automatically calculated by the program as  $\epsilon^T = \sqrt{2/3} * L$  (since it is assumed that the input to this model is an extension of the mechanical model and conversion is done wherever applicable as in Table 9-27. However, in the case where the thermal-mechanical model input parameters are directly used then one must enter a value of  $\sqrt{2/3} * \epsilon_q$  so that a correct value of  $\epsilon_q$  is used in the calculations.
4. The Mechanical (Auricchio's) model is not available for 1-D or plane stress conditions. The thermo-mechanical model must be used.
5. In SOL 400, MATSMA is only supported for nonlinear elements with property extensions, PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MSC Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
6. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
7. In SOL 400; remark 4. implies that the Mechanical model may not be used with PBAR, PBARL, PBEAM, PBEAML, PROD, PSHEAR, PSHELL, PCOMP(G). It can only be used with PLPLANE/PSHLN2 (without BEH=PSTRS) and PSOLID/PSLDN1.



## MATSORT      Advanced Orthotropic, Nonlinear Elastic Material for Shell Elements

Specifies NLELAST option for advanced 3D orthotropic, nonlinear elastic materials in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATSORT	MID	TE1	TE2	TE3	TNU12	TNU23	TNU31	TRHO	
	TG12	TG23	TG31	TA1	TA2	TA3			

Example:

MATSORT	689	77	77	77					
				89	89	89			

Describer	Meaning
MID	Identification number of a MATORT entry. (Integer > 0; no Default)
TE1	ID of TABL3Di entry for E1. (Integer; no Default; leave blank if table is not required)
TE2	ID of TABL3Di entry for E2. (Integer; no Default; leave blank if table is not required)
TE3	ID of TABL3Di entry for E3. (Integer; no Default; leave blank if table is not required)
TNU12	ID of TABL3Di entry for NU12. (Integer; no Default; leave blank if table is not required)
TNU23	ID of TABL3Di entry for NU23. (Integer; no Default; leave blank if table is not required)
TNU31	ID of TABL3Di entry for NU31. (Integer; no Default; leave blank if table is not required)
TRHO	ID of TABL3Di entry for RHO. (Integer; no Default; leave blank if table is not required)
TG12	ID of TABL3Di entry for G12. (Integer; no Default; leave blank if table is not required)
TG23	ID of TABL3Di entry for G23. (Integer; no Default; leave blank if table is not required)
TG31	ID of TABL3Di entry for G31. (Integer; no Default; leave blank if table is not required)
TA1	ID of TABL3Di entry for A1. (Integer; no Default; leave blank if table is not required)
TA2	ID of TABL3Di entry for A2. (Integer; no Default; leave blank if table is not required)
TA3	ID of TABL3Di entry for A3. (Integer; no Default; leave blank if table is not required)

Remarks:

1. TABL3Di is a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc. If one of the independent variables is temperatures then you should not use MATTORT.

2. In SOL 400, MATSORT is used in conjunction with MATORT and is only supported for 2D and 3D elements with property extensions. This implies that for such elements, PCOMP / PCOMP, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PSHLN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the analysis would stop with an error. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with default settings by the program unless the user has set NLMOPTS,SPROP,MAP,-1 in the bulk data.

## MATT1 Isotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT1	MID	T(E)	T(G)	T(NU)	T(RHO)	T(A)		T(GE)	
	T(ST)	T(SC)	T(SS)						

Example:

MATT1	17	32	65	84		15			
	52								

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E)	Identification number of a TABLEMi entry for the Young's modulus. (Integer $\geq 0$ or blank)
T(G)	Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)
T(NU)	Identification number of a TABLEMi entry for the Poisson's ratio. (Integer $\geq 0$ or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer $\geq 0$ or blank)
T(A)	Identification number of a TABLEMi entry for the thermal expansion coefficient. See Remark 4. (Positive or negative integer, 0 or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or blank)
T(ST)	Identification number of a TABLEMi entry for the tension stress limit. (Integer $\geq 0$ or blank)
T(SC)	Identification number of a TABLEMi entry for the compression limit. (Integer $\geq 0$ or blank)
T(SS)	Identification number of a TABLEMi entry for the shear limit. (Integer $\geq 0$ or blank)

## Remarks:

1. Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT1 entry referenced in field 2. The value in a particular field of the MAT1 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E is modified by TABLEMi 32, G is modified by TABLEMi 65, NU is modified by TABLEMi 84, A is modified by TABLEMi 15, and ST is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT1 entry.
2. Any quantity modified by this entry must have a value on the MAT1 entry. Initial values of E, G, or NU may be supplied according to Remark 3 on the MAT1 entry.
3. Table references must be present for each item that is temperature dependent. If any one of Young's modulus, shear modulus or Poisson's ratio is temperature dependent, then T(E), T(G) and T(NU) should be specified together to get accurate results.
4. The T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMP MATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain  $\varepsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative  $ID_{T(A)}$  value will be changed to  $|ID_{T(A)}| + 100000000$ .
5. The continuation entry is not used by SOL 600. For SOL 600, see MATTEP.
6. T(G), T(RHO) and T(GE) are not used by SOL 600. T(E) and T(NU) should be used instead of T(E) and T(G).
7. This entry is not used by SOL 700.

## MATT2 Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT2	MID	T(G11)	T(G12)	T(G13)	T(G22)	T(G23)	T(G33)	T(RHO)	
	T(A1)	T(A2)	T(A3)		T(GE)	T(ST)	T(SC)	T(SS)	
		T(GE11)	T(GE12)	T(GE13)	T(GE22)	T(GE23)	T(GE33)		

Example:

MATT2	17	32					15		
	62								

Describer	Meaning
MID	Material property identification number that matches the identification number on a MAT2 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficient. See Remark 3. (Positive or negative integer, 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the tension compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer ≥ 0 or blank)
T(GEij)	Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer > 0 or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT2 entry referenced in field 2. The value in a particular field of the MAT2 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMk 32, G33 is modified by TABLEMk 15, and A1 is modified by TABLEMk 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT2 entry.

2. Any quantity modified by this entry must have a value on the MAT2 entry.
3. Any of the T(A) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMPMMATT,YES, if the TABLEMk ID for the coefficient of thermal expansion is negative, the TABLEMk ordinate values will be interpreted as the thermal strain  $\epsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative  $ID_{T(A)}$  value will be changed to  $|ID_{T(A)}| + 100000000$

## MATT3 MAT3 Material Temperature Dependence

Specifies temperature-dependent material properties on MAT3 entry fields via TABLEMi entries that are temperature dependent.

Format:

1	2	3	4	5	6	7	8	9	10
MATT3	MID	T(EX)	T(ETH)	T(EZ)	T(NUXTH)	T(NUTHZ)	T(NUZX)	T(RHO)	
			T(GZX)	T(AX)	T(ATH)	T(AZ)		T(GE)	

Example:

MATT3	23	32		15					
			62						

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT3 entry. (Integer > 0)
T(EX) T(ETH) T(EZ)	Identification number of a TABLEMi entry for the Young's modulus in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank)
T(GZX)	Identification number of a TABLEMi entry for the shear modulus. (Integer $\geq 0$ or blank)
T(NUXTH) T(NUTHZ) T(NUZX)	Identification number of a TABLEMi entry for the Poisson's ratio in the $x\theta$ , $\theta z$ , and $zx$ directions. (Integer $\geq 0$ or blank)
T(RHO)	Identification number of a TABLEMi entry for the mass density. (Integer $\geq 0$ or blank)
T(AX) T(ATH) T(AZ)	Identification number of a TABLEMi entry for the thermal expansion coefficients in the $x$ , $\theta$ , and $z$ directions. (Integer $\geq 0$ or blank)
T(GE)	Identification number of a TABLEMi entry for the damping coefficient. (Integer $\geq 0$ or blank)

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT3 entry referenced in field 2. The value recorded in a particular field of the MAT3 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, EX is modified by TABLEMi 32, EZ is modified by TABLEMi 15, and GZX is modified by TABLEMi 62. If Ri is zero or blank, then there is no temperature dependence of the field on the MAT3 entry.
- Any quantity modified by this entry must have a value on the MAT3 entry.

## MATT4 Thermal Material Temperature Dependence

Specifies table references for temperature-dependent MAT4 material properties.

Format:

1	2	3	4	5	6	7	8	9	10
MATT4	MID	T(K)	T(CP)		T(H)	T( $\mu$ )	T(HGEN)		

Example:

MATT4	17	102	103			105			
-------	----	-----	-----	--	--	-----	--	--	--

Describer	Meaning
MID	Identification number of a MAT4 entry that is temperature dependent. (Integer > 0)
T(K)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal conductivity. (Integer $\geq$ 0 or blank)
T(CP)	Identification number of a TABLEMj entry that gives the temperature dependence of the thermal heat capacity. (Integer $\geq$ 0 or blank)
T(H)	Identification number of a TABLEMj entry that gives the temperature dependence of the free convection heat transfer coefficient. (Integer $\geq$ 0 or blank)
T( $\mu$ )	Identification number of a TABLEMj entry that gives the temperature dependence of the dynamic viscosity. (Integer $\geq$ 0 or blank)
T(HGEN)	Identification number of a TABLEMj entry that gives the temperature dependence of the internal heat generation property for QVOL. (Integer $\geq$ 0 or blank)

Remarks:

1. The basic quantities on the MAT4 entry are always multiplied by the corresponding tabular function referenced by the MATT4 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the MAT4 entry.



## MATT5 Thermal Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT5 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT5	MID	T(KXX)	T(KXY)	T(KXZ)	T(KYY)	T(KYZ)	T(KZZ)	T(CP)	
		T(HGEN)							

Example:

MATT5	24	73							
-------	----	----	--	--	--	--	--	--	--

Describer	Meaning
MID	Identification number of a MAT5 entry that is to be temperature dependent. (Integer > 0)
T(Kij)	Identification number of a TABLEMi entry. The TABLEMi entry specifies temperature dependence of the matrix term. (Integer ≥ 0 or blank)
T(CP)	Identification number of a TABLEMi entry that specifies the temperature dependence of the thermal heat capacity. (Integer ≥ 0 or blank)
T(HGEN)	Identification number of a TABLEMi entry that gives the temperature dependence of the internal heat generation property for the QVOL entry. (Integer ≥ 0 or blank)

Remarks:

1. The basic quantities on the MAT5 entry are always multiplied by the tabular function referenced by the MATT5 entry.
2. If the fields are blank or zero, then there is no temperature dependence of the referenced quantity on the basic MAT5 entry.

## MATT8 Shell Orthotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT8	MID	T(E1)	T(E2)	T(NU12)	T(G12)	T(G1Z)	T(G2Z)	T(RHO)	
	T(A1)	T(A2)		T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(S)	
	T(GE)	T(F12)							

Example:

MATT8	17	32							
	15			52					

Describer	Meaning
MAT	Material property identification number that matches the identification number on MAT1 entry. (Integer > 0)
T(E1)	Identification number of a TABLEMi entry for the Young's modulus 1. (Integer ≥ 0 or blank)
T(E2)	Identification number of a TABLEMi entry for the Young's modulus 2. (Integer ≥ 0 or blank)
T(NU12)	Identification number of a TABLEMi entry for Poisson's ratio 12. (Integer ≥ 0 or blank)
T(G12)	Identification number of a TABLEMi entry for shear modulus 12. (Integer ≥ 0 or blank)
T(G1Z)	Identification number of a TABLEMi entry for transverse shear modulus 1Z. (Integer ≥ 0 or blank)
T(G2Z)	Identification number of a TABLEMi entry for transverse shear modulus 2Z. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMi entry for mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMi entry for thermal expansion coefficient 1. See Remark 3. (Positive or negative integer, 0 or blank)
T(A2)	Identification number of a TABLEMi entry for thermal expansion coefficient 2. See Remark 3. (Integer or blank)
T(Xt)	Identification number of a TABLEMi entry for tension stress/strain limit 1. (Integer ≥ 0 or blank)
T(Xc)	Identification number of a TABLEMi entry for compression stress/strain limit 1. (Integer ≥ 0 or blank)

Describer	Meaning
T(Yt)	Identification number of a TABLEMi entry for tension stress/strain limit 2. (Integer $\geq 0$ or blank)
T(Yc)	Identification number of a TABLEMi entry for compression stress/strain limit 2. (Integer $\geq 0$ or blank)
T(S)	Identification number of a TABLEMi entry for shear stress/strain limit. (Integer $\geq 0$ or blank)
T(GE)	Identification number of a TABLEMi entry for structural damping coefficient. (Integer $\geq 0$ or blank)
T(F12)	Identification number of a TABLEMi entry for Tsai-Wu interaction term. (Integer $\geq 0$ or blank)

**Remarks:**

- Fields 3, 4, etc., of this entry correspond, field-by-field, to fields 3, 4, etc., of the MAT8 entry referenced in field 2. The value in a particular field of the MAT8 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, E1 is modified by TABLEMi 32, A1 is modified by TABLEMi 15, and Xt is modified by TABLEMi 52. Blank or zero entries mean that there is no temperature dependence of the fields on the MAT8 entry.
- Any quantity modified by this entry must have a value on the MAT8 entry.
- Any of the T(Ai) table value can be either a positive or negative integer. For a nonlinear static analysis of a composite element with the PARAM,COMP MATT,YES, if the TABLEMi ID for the coefficient of thermal expansion is negative, the TABLEMi ordinate values will be interpreted as the thermal strain  $\epsilon(T)$  rather than the coefficient of thermal expansion ordinate values  $\alpha(T)$ . Internally to Nastran, a negative  $ID_{T(A)}$  value will be changed to  $|ID_{T(A)}| + 100000000$ .

## MATT9 Solid Element Anisotropic Material Temperature Dependence

Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

Format:

1	2	3	4	5	6	7	8	9	10
MATT9	MID	T(G11)	T(G12)	T(G13)	T(G14)	T(G15)	T(G16)	T(G22)	
	T(G23)	T(G24)	T(G25)	T(G26)	T(G33)	T(G34)	T(G35)	T(G36)	
	T(G44)	T(G45)	T(G46)	T(G55)	T(G56)	T(G66)	T(RHO)	T(A1)	
	T(A2)	T(A3)	T(A4)	T(A5)	T(A6)		T(GE)		
	T(GE11)	T(GE12)	T(GE13)	T(GE14)	T(GE15)	T(GE16)	T(GE22)	T(GE23)	
	T(GE24)	T(GE25)	T(GE26)	T(GE33)	T(GE34)	T(GE35)	T(GE36)	T(GE44)	
	T(GE45)	T(GE46)	T(GE55)	T(GE56)	T(GE66)				

Example:

MATT9	17	32			18			17	
				12					
				5			10		

Describer	Meaning
MID	Material property identification number that matches the identification number on MAT9 entry. (Integer > 0)
T(Gij)	Identification number of a TABLEMk entry for the terms in the material property matrix. (Integer ≥ 0 or blank)
T(RHO)	Identification number of a TABLEMk entry for the mass density. (Integer ≥ 0 or blank)
T(Ai)	Identification number of a TABLEMk entry for the thermal expansion coefficients. (Integer ≥ 0 or blank)
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(GEij)	Identification number of a TABLEMk entry for the terms in the material structural damping property matrix. (Integer > 0 or blank).

Remarks:

- Fields 3, 4, etc., of this entry correspond, field by field, to fields 3, 4, etc., of the MAT9 entry referenced in field 2. The value recorded in a particular field of the MAT9 entry is replaced or modified by the table referenced in the corresponding field of this entry. In the example shown, G11 is modified by TABLEMj 32, G14 is modified by TABLEMj 18, etc. If the fields are zero or blank, then there is no temperature dependence of the fields on the MAT9 entry.
- Any quantity modified by this entry must have a value on the MAT9 entry.
- The continuation entries are optional.

**MATTEP**

**Thermo-Elastic-Plastic Material Properties - SOL 400**

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 400 only.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
MATTEP	MID		T(Y0)	T(FID)				T(H)	
	N/A			N/A					
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	$\mu$	$\eta$					

Example:

MATTEP	100		20						
--------	-----	--	----	--	--	--	--	--	--

Describer	Meaning
MID	Identification number of MATEP entry. See Remark 1. (Integer > 0)
T(Y0)	Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)
T(FID)	Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.
T(H)	Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank)
“Chaboche”	A keyword specifying the following data pertains to the Chaboche model.
R0	Identification number of TABLEMi entry for temperature dependent R0 for isotropic hardening (Integer > 0).
Rinf	Identification number of TABLEMi entry for temperature dependent Rinfinity for isotropic hardening (Integer > 0).
B	Identification number of TABLEMi entry for temperature dependent coefficient B for isotropic hardening (Integer > 0).
C	Identification number of TABLEMi entry for temperature dependent coefficient C for kinematic hardening (Integer > 0).
Kap	Identification number of TABLEMi entry for temperature dependent Kappa value for viscosity model (Integer > 0).
N	Identification number of TABLEMi entry for temperature dependent coefficient n for viscosity model (Integer).
Qm	Identification number of TABLEMi entry for temperature dependent coefficient Qm for isotropic hardening (Integer > 0)



Describer	Meaning
$\mu$	Identification number of TABLEMi entry for temperature dependent coefficient $\mu$ for isotropic hardening (Integer > 0).
$\eta$	Identification number of TABLEMi entry for temperature dependent coefficient $\eta$ to introduce progressive memory (Integer > 0).

#### Remarks:

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, SOL 400 will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

## MATTEP

## Thermo-Elastic-Plastic Material Properties - SOL 600

Specifies temperature-dependent elasto-plastic material properties to be used for static, quasi static or transient dynamic analysis in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTEP	MID		T(Y0)	T(FID)				T(H)	
	N/A		T(yc10)	N/A					
	“Chaboche”	R0	Rinf	B	C	Gam	Kap	N	
		Qm	$\mu$	$\eta$					
	“PwrLaw”	A	M	B	N	$\sigma_0 \epsilon_0$			

Example:

MATTEP	100		20						
--------	-----	--	----	--	--	--	--	--	--

Describer	Meaning
MID	Identification number of MATEP entry. See Remark 1. (Integer > 0)
T(Y0)	Identification number of TABLEMi entry for thermo-elasto-plastic material. See Remarks 2. (Integer > 0 or blank)
T(FID)	Identification number of TABLEST entry for temperature-dependent stress-strain curves (Integer > 0 or blank). See Remark 4.
T(H)	Identification number of TABLEMi entry for temperature-dependent plasticity moduli in thermo-elasto-plastic material. See Remarks 3. (Integer > 0 or blank)
T(yc10)	Identification number of TABLEMi entry for equivalent 10th cycle tensile yield stress specified in the Yc10 field of MATEP entry. (Integer > 0 or blank).
“Chaboche”	A keyword specifying the following data pertains to the Chaboche model.
R0	Table for R0 for isotropic hardening. (Integer > 0)
Rinf	Table for Rinfinity for isotropic hardening (Integer > 0)
B	Table for b coefficient for isotropic hardening. (Integer > 0)
C	Table for C coefficient for kinematic hardening. (Integer > 0)
Kap	Table for Kappa value for viscosity model. (Integer > 0)
N	Table for n coefficient for viscosity model. (Integer > 0)
Qm	Table for Qm coefficient for isotropic hardening. (Integer > 0)
$\mu$	Table for $\mu$ coefficient for isotropic hardening. (Integer > 0)
$\eta$	Table for $\eta$ coefficient to introduce progressive memory. (Integer > 0)

Describer	Meaning
“PwrLaw”	A keyword specifying the following data pertains to the Power Law or Rate Power Law model (see Marc Vol C , ISOTROPIC option for more details).
A	Table for coefficient A. (Integer > 0)
M	Table for coefficient m. (Integer > 0)
B	Table for coefficient B. (Integer > 0)
N	Table for exponent n. (Integer > 0)
$\sigma_0 \epsilon_0$	Table for $\sigma_0 \epsilon_0$ . (Integer > 0)

#### Remarks:

1. The MATEP Bulk Data entry with the same MID must exist for MATTEP to be effective. All the fields defined in MATTEP correspond to the same fields of MATEP. The value in a particular field of the MATEP entry is replaced or modified by the table referenced in the corresponding field of this entry.
2. The table represents yield stresses as a function of temperature. Therefore, the curve should comprise the initial stress from Y0 or FID field on MATEP (most likely at room temperature). T(Y0) field accommodates FID field in case FID field defines the initial yield stress instead of Y0 field. In this case, the yield stresses at any plastic strain will be scaled by the same ratio as the initial yield stress at the same temperature.
3. The table represents a normalized plasticity moduli (work hardening slope) as a function of temperature.
4. Temperature dependent stress-strain curves may be entered in a general manner using the T(FID) option. The integer value entered in this field represents the ID of a TABLEST entry which provides IDs of TABLES1 stress-plastic strain curves vs. temperature. All such curves must be entered as stress vs. plastic strain. No curves should be referenced on the MATS1 entry. For this option T(Y0) and T(H) should be left blank and if entered, MSC Nastran will re-set them to blank if T(FID) is a positive integer.
5. This entry must be used in conjunction with MAT1, MATEP and MATT1 all with the same MID. The MATT1 entry must have at least one non-blank entry in fields 3-7 of the primary MATT1 entry.

#### Note:

This Bulk Data entry accommodates Marc’s input data under the model definition options TEMPERATURE EFFECTS.



## MATTF Material Failure Model Temperature Variation

Describes the temperature, strain rate, or other type of variation of material failure properties used in conjunction with MATF in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTF	MID		T(SB)						
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	1st
	T(Sxy)	T(Syz)	T(Szx)	T(Find)	T(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	2nd
	T(Sxy)	T(Syz)	T(Szx)	F(Find)	F(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	
	KIND	Criteria	T(Xt)	T(Xc)	T(Yt)	T(Yc)	T(Zt)	T(Zc)	3rd
	T(Sxy)	T(Syz)	T(Szx)	T(Find)	T(Fxy)	T(Fyz)	T(Fzx)	T(Ext)	
	T(Exc)	T(Eyt)	T(Eyc)	T(Ezt)	T(Ezc)	T(Gxy)	T(Gyz)	T(Gzx)	

Example:

MATTF	100								
	12	1	51	52	53	54	55	56	
	61	62	63						
	12	2							2nd
	71	72	73	74	75	76	77	78	
	12	4	81	82	83	84	85	86	3rd
	91	92	93	94					

(Note: The 4th and 6th lines cannot be entirely blank and the last line of the 3rd criterion has been omitted.)

Describer	Meaning
MID	Identification number of a matching MATF entry. (Integer > 0; no Default; Required)
T(SB)	Identification of a TABLEMi entry providing the variation of the allowable shear stress of the bonding material between layers (composites only) (Integer; Default = 0 meaning no variation).
KIND	Enter the type of variation of the failure properties using the values listed below. (Integer; no Default) 12 Temperature

Describer	Meaning
Criteria	Enter same value as used by MATF (reference only). (Integer) If more than one KIND is required, enter the three lines for each KIND and Criteria as many times are required. (Integer; must be same as used by companion MATF, reference only)
T(SB)	Identification of a TABLEMi entry providing the variation of the allowable shear stress of the bonding material between layers (composites only) (Integer; Default = 0 meaning no variation)
T(Xt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in x-direction (Integer; Default = 0 meaning no variation)
T(Xc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in x-direction (Integer; Default = 0 meaning no variation)
T(Yt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in y-direction (Integer; Default = 0 meaning no variation)
T(Yc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in y-direction (Integer or blank)
T(Zt)	Identification of a TABLEMi entry providing the variation of the maximum tensile stress in z-direction (Integer; Default = 0 meaning no variation)
T(Zc)	Identification of a TABLEMi entry providing the variation of the maximum compressive stress (absolute value) in z-direction (Integer; Default = 0 meaning no variation)
T(Sxy)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in xy-plane (Integer; Default = 0 meaning no variation)
T(Syz)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in yz-plane (Integer; Default = 0 meaning no variation)
T(Szx)	Identification of a TABLEMi entry providing the variation of the maximum shear stress in zx-plane (Integer; Default = 0 meaning no variation)
T(Find)	Identification of a TABLEMi entry providing the variation of the Failure index (Real > 0., (Integer; Default = 0 meaning no variation)
T(Fxy)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for xy-plane (Integer; Default = 0 meaning no variation)
T(Fyz)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for yz-plane (Integer; Default = 0 meaning no variation)
T(Fzx)	Identification of a TABLEMi entry providing the variation of the interactive strength constant for zx-plane (Integer; Default = 0 meaning no variation)
T(Ext)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in x-direction (Integer; Default = 0 meaning no variation)

Describer	Meaning
T(Exc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in x-direction (Integer; Default = 0 meaning no variation)
T(Yet)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in y-direction (Integer; Default = 0 meaning no variation)
T(Eyc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in y-direction (Integer; Default = 0 meaning no variation)
T(Ezt)	Identification of a TABLEMi entry providing the variation of the maximum tensile strain in z-direction (Integer; Default = 0 meaning no variation)
T(Ezc)	Identification of a TABLEMi entry providing the variation of the maximum compressive strain (absolute value) in z-direction (Integer; Default = 0 meaning no variation)
T(Gxy)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in xy-plane (Integer; Default = 0 meaning no variation)
T(Gyz)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in yz-plane (Integer; Default = 0 meaning no variation)
T(Gzx)	Identification of a TABLEMi entry providing the variation of the maximum shear strain in zx-plane (Integer; Default = 0 meaning no variation)

Remarks:

1. See the MATF entry for a complete description of the meaning of each of the variables for the various failure criterion.
2. If there is no variation of a particular entry, leave the field blank or enter zero.
3. Continuation entries except the last line are required. The last line is optional.

**MATTG**

Temperature Variation of Interlaminar Materials

Specifies gasket material property temperature variation to be used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTG	MID	IDYM	IDVM	IDDM	IDLD	IDU1	IDU2	IDU3	
	IDU4	IDU5	IDU6	IDU7	IDU8	IDU9	IDU10	IDYPR	
	IDEPL	IDGPL	IDGAP	N/A	N/A	N/A	N/A	N/A	

Example:

MATTG	100	10	20	1001	1002	1003			
								1010	
	1020	1030							

Describer	Meaning
MID	Material ID number that matches the material ID of a corresponding MATG material. (Integer > 0; Required)
IDYM	ID of TABLEMi entry that gives the temperature variation of Young’s modulus for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDVM	ID of TABLEMi entry that gives the temperature variation of Poisson’s ratio for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDDM	ID of TABLEMi entry that gives the temperature variation of the mass density for the membrane behavior of the material. (Integer ≥ 0 or blank)
IDLD	ID of TABLEMi entry that gives the temperature variation of the loading curve of the material. (Integer ≥ 0 or blank)
IDUi	ID of TABLEMi entry that gives the temperature variation of the unloading curve of the material. There can be up to 10 unloading curves and each can have a different temperature variation. If there is no unloading, there does not need to be any unloading curves. (Integer ≥ 0 or blank)
IDYPR	ID of TABLEMi entry that gives the temperature variation of the yield pressure for the out-of-plane behavior of the material. (Integer ≥ 0 or blank)
IDEPL	ID of TABLEMi entry that gives the temperature variation of the tensile modulus for the out-of-plane behavior of the material. (Integer ≥ 0 or blank)
IDGPL	ID of TABLEMi entry that gives the temperature variation of the transverse shear modulus for the out-of-plane behavior of the material. (Integer ≥ 0 or blank)
IDGAP	ID of TABLEMi entry that gives the temperature variation of the initial gap for the out-of-plane behavior of the material. (Integer ≥ 0 or blank)

**MATTHE**

**Hyperelastic Material - Temperature Dependence**

Specifies temperature-dependent properties of hyperelastic (rubber-like) materials (elastomers) for nonlinear (large strain and large rotation) analysis in SOLs 106, 129, 600 and SOL 400 only.

Format 1 (Default): Generalized Mooney-Rivlin model (Model=Mooney). The first two lines are required, the others may be omitted depending on how many terms are desired.)

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(C10)	T(C01)	T(D1)						
	T(C20)	T(C11)	T(C02)	T(D2)					
	T(C30)	T(C21)	T(C12)	T(C03)	T(D3)				
	T(C40)	T(C31)	T(C22)	T(C13)	T(C04)	T(D4)			
	T(C50)	T(C41)	T(C32)	T(C23)	T(C14)	T(C05)	T(D5)		

Format 2: Ogden Model

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(D1)	T(D2)	T(D3)	T(D4)	T(D5)				

Format 3: Arruda-Boyce model or Gent Model (Model = Aboyce or Gent)

1	2	3	4	5	6	7	8	9	10
MATTHE	MID	Model		T(K)	T(RHO)	T(Texp)		T(GE)	
	T(NKT)	T(N/E)	T(Im)						
	T(D1)	T(D2)	T(D3)	T(D4)	T(D5)				

Example - Format 1:

MATTHE	1	Monney							
	10	1	100						
	20	11	2	200					
	30	21	12	3	300				
	40	31	22	13	4	400			
	50	41	32	23	14	5	500		

Example - Format 2:

MATTHE	1	Ogden							
	100	200	300	400	500				

Example - Format 3:

MATTHE	1	Aboyce							
	1	2							
	100	200	300	400	500				

Describer	Meaning
MID	Identification number of a MATHE entry. See Remark 1. (Integer > 0; no Default)
Model	Select hyperelastic material model form. (Character; Default = Mooney)
	Mooney For generalized Mooney-Rivlin hyperelastic model
	Ogden For Ogden hyperelastic model
	Foam For Foam model
	Aboyce For Arruda-Boyce strain energy model
	Gent For Gent strain energy model
T(Value)	Identification number of a TABLEMi entry for the temperature variation of the matching value from the MATHE entry - see the MATHE entry for the definition of each coefficient value. (Integer; Default = 0 which means no table variation for that particular value)

Remarks:

1. The MATTHE entry must have the same ID as the corresponding MATHE entry. Each table ID on the MATTHE entry corresponds to a parameter on the MATHE entry.
2. If the thermal expansion coefficient is temperature-dependent, the thermal strain is computed as

$$\varepsilon_{th} = \bar{\alpha}(T - T_{ref}) - \bar{\alpha}_0(T_0 - T_{ref})$$

where  $T_{ref}$  is the reference temperature at which measurement of the thermal expansion coefficient is based,  $T_0$  is an initial temperature, and  $\bar{\alpha}_0$  is the thermal expansion coefficient at the initial temperature  $T_0$ . If the thermal expansion is not temperature-dependent, the thermal strain expression is reduced to the usual expression:

$$\varepsilon_{th} = \bar{\alpha}(T - T_0)$$

3. If experimental data is provided, it is expected that the user has the data for multiple tests of the same type at different temperatures. The use of TABLEST's for this entry is not presently available. If experimental data is used to define the constants, use a GUI to define the constants at each temperature, enter the baseline values on the MATHE entry and the variation of each coefficient using TABLEMi.

## MATTORT

## Thermoelastic Orthotropic Material

Specifies temperature-dependent properties of elastic orthotropic materials for linear and nonlinear analyses used in SOL 600 and SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
MATTORT	MID	T(E1)	T(E2)	T(E3)	T(NU12)	T(NU23)	T(NU31)	N/A	
	T(G12)	T(G23)	T(G31)	T(A1)	T(A2)	T(A3)	T(SY)	T(WHS)	

Example:

MATTORT	100	5	6	7					
---------	-----	---	---	---	--	--	--	--	--

Describer	Meaning
MID	Identification number of a MATORT entry. See Remark 1. (Integer > 0; no Default)
T(Ei)	Identification number of a TABLEMi entry for the Young's modulus as a function of temperature in each respective direction. Remark 2. (Integer > 0 or blank)
T(Nuij)	Identification number of a TABLEMi entry for the Poisson's ratio as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Gij)	Identification number of a TABLEMi entry for the shear modulus as a function of temperature in each respective direction. (Integer > 0 or blank)
T(Ai)	Identification number of a TABLEMi entry for the coefficient of thermal expansion as a function of temperature. (Integer > 0 or blank)
T(SY)	Identification number of a TABLEMi entry for the yield stress as a function of temperature. (Integer > 0 or blank)
T(WHS)	Identification number of a TABLEMi entry for the work hardening slope as a function of temperature. (Integer > 0 or blank)

Remarks:

1. The MATTORT entry must have the same ID as the corresponding MATORT entry. Each table ID on the MATTORT entry corresponds to a parameter on the MATORT entry.
2. The table represents material constants as a function of temperature. Therefore, the curve should comprise the original value specified in the MATORT entry (most likely at room temperature).
3. For SOL 600, the continuation line is required for versions prior to MD Nastran 2010 and MSC Nastran 2009.



## MATTUSR Specifies Table Variation of User Defined Generic Materials

Specifies table variation of user defined generic materials in SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATTUSR	MID			T(RHO)	T(A1)	T(A2)	T(A3)		
	T(GE)	T(ST)	T(SC)	T(SS)					

Example:

MATTUSR	20			51	52				
---------	----	--	--	----	----	--	--	--	--

Describer	Meaning
MID	Identification number of MATUSR. (Integer > 0; no Default)
T(RHO)	Table ID of a TABLEM1 or TABL3Di for the mass density for structural analysis. (Integer, no Default)
T(Ai)	Table ID of a TABLEM1 or TABL3Di for the coefficient of thermal expansion. (Integer) If T(A1) is not zero and T(A2) and T(A3) are zero, then T(A1)=T(A2)=T(A3).
T(GE)	Identification number of a TABLEMk entry for the damping coefficient. (Integer ≥ 0 or blank)
T(ST)	Identification number of a TABLEMk entry for the tension stress limit. (Integer ≥ 0 or blank)
T(SC)	Identification number of a TABLEMk entry for the compression limit. (Integer ≥ 0 or blank)
T(SS)	Identification number of a TABLEMk entry for the tension shear limit. (Integer ≥ 0 or blank)

**MATVE**

Thermo-Visco-Elastic Material Properties

Specifies temperature-dependent visco-elastic material properties in terms of Thermo-Rheologically Simple behavior to be used for quasi-static or transient dynamic analysis in SOL 600 and SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	function	RT	ENER	FRACT	TDIF	TREF	NP	
	A1	A2							
	C0	C1	C2	C3	C4	etc.			
	W0	W1	W2	W3	W4	etc.			
	T0	T1	T2	T3	T4	etc.			
	IFICT								

Example:

MATVE	100	WLF	100.						
	0.8	1.2							
MATVE	101	POWER	100.					11	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W0-7
	.92	.915	.914						W8-10
MATVE	102	NARA		4.3E-8	.75	200.	400.	11	
	0.0	0.0							
	1.0	.99	.98	.97	.96	.95	.94	.93	W0-7
	.92	.915	.914						W8-10
	235.	234.	233.	232.	231.	230.	229.	228.	T0-7
	227.	226.	225.						T8-10
MATVE	103	USER	345.					8	

Describer	Meaning
MID	Identification number. See Remark 1. (Integer > 0)
Function	Name of the shift function. See Remarks 2. and 3. (Character)
	W Williams-Landell-Ferry form, Requires A1 and A2. (Default)
	LF

Describer	Meaning
	P power series form
	O
	W
	E
	R
	N Narayanaswamy model
	A
	R
	A
	U specify the shift function with a user subroutine
	SE
	R
RT	Enter the reference or glass transition temperature. (Real; Default = 0.)
ENER	Used for NARA model only, enter the activation energy divided by the gas constant Q/R. (Real; no Default)
FRACT	Used for NARA only, enter the fraction parameter. (Real; no Default)
TDIF	Used for NARA only, enter the temperature shift between your temperature and absolute temperature for calculating fictitive temperatures. (Real; no Default)
TREF	Used for NARA only, enter the reference temperature for structural relaxation. (Real; no Default)
NP	For POWER, enter the number of coefficients in the power series representation. For NARA, NP is the member of terms in the Proxy series.
A1, A2	For WLF model enter the constants A1 and A2. For other models, enter 0.0 for A1 and A2. Do not enter a blank line as it will be stripped out. (Real; no Default)
Ci	Coefficients of the shift function, enter NP values for POWER only. Do not enter Ci for NARA. (Real; no Defaults)
Wi	For NARA model only, enter the weighting factors in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults)
Ti	For the NARA model only, enter the relaxation time values in increasing order of subscript. Enter NP values. For WLF and POWER, skip these values. (Real; no Defaults)
IFICT	For NARA model only, enter initial fictive temperature (Integer > 0)
	1= Initial temperature
	2= Structural relaxation reference temperature
	3= 0.0 (default)

## Remarks:

1. The MATVE Bulk Data entry with the same MID must exist for MATTVE to be effective.

- The viscoelastic behavior is especially noticeable in the organic high polymers. There are many different kinds of such materials including various plastics, natural and synthetic rubbers. Their mechanical properties depend strongly on temperature, and these properties change drastically in the vicinity of a critical temperature called the glass-transition temperature  $T_g$ . The polymer well below  $T_g$  is an organic glass with a relatively high modulus. The viscoelastic behavior predominates in the transition range around  $T_g$ . The polymer above the transition region (but below the melting point) becomes a rubbery solid with a low modulus.

Polymers are broadly classified as amorphous polymers and polycrystalline polymers. Under stress-relaxation at a constant strain in the glass-transition region temperature, the amorphous polymer exhibits a phase change over time from the glassy state to the rubbery state. The response is manifested in the shear modulus as a function of time, in which initially high shear modulus changes into low shear modulus. The relaxation curve of the modulus in a log-log scale plot appears as a flat plateau of glassy modulus  $G_g$  shifting down to the equilibrium modulus  $G_e$  at the rubbery plateau. Such a relaxation behavior of the amorphous polymer is observed even when the temperature is well below  $T_g$  for a prolonged period of time in a very slow process. A similar behavior is found in the rubbery elastic region, but the process is faster.

Fortunately, the mechanical properties of amorphous polymers obey a time-temperature superposition principle, which allows the use of data obtained at different temperatures to extend the time scale at any given temperature. For such a behavior, the amorphous polymer is characterized as thermo-rheologically simple (TRS). Since the relaxation process extends several decades on the logarithmic time scale at lower temperatures, it is not feasible to determine the whole curve by a constant strain test at one temperature. Instead, the relaxation characteristics are measured at elevated temperatures in reduced time scale. Then the polymers exhibit a translational shift of all the characteristic responses with a change of temperature along the logarithmic time axis. This shift occurs parallel to the time axis without a change in properties: glassy and rubbery moduli. The modulus curve shifts towards shorter time with an increased temperature.

- The reduced ( $\xi$ ), or pseudo, time is related to the actual time ( $t$ ) through a shift function which is a function of temperature, i.e.,

$$\xi(t) = \int_0^t \frac{ds}{A(T(s))}$$

where  $A(T)$  is a shift function in terms of temperature  $T$  at time  $t$ . The shift function is a material property and must be determined experimentally. A shift function approximated by Williams-Landell-Ferry, known as WLF equation, has the form:

$$\log A = h(T) = -\frac{A_1(T - T_0)}{A_2 + (T - T_0)}$$

where  $T_0$  is the reference temperature at which relaxation data are given and  $A_1, A_2$  are calibration constants obtained at this temperature. Notice that  $A = 1$  if the reduced time is the same as the actual time. If  $T \leq T_0 - A_2$ , the deformation will be elastic.

Another form of the shift function is available as a power series in  $(T - T_0)$ , i.e.,

$$\log A = h(T) = \sum_{i=0}^{10} C_i (T - T_0)^i$$

4. The WLF shift function requires A1 and A2. The power series can have a maximum 11 coefficients C0 through C10.

**Note:**

1. This entry matches the three options for SHIFT FUNCTION in Marc: William-Landell-Ferry, Power Series and Narayanaswamy models.
2. The user subroutine is TRSFAC in SOL 600; this is not supported in SOL 400.

## MATUDS

## Material User Defined Service or Subroutine

Allows the user to provide material routines for use with enhanced material models in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MATUD S	MID	MTYPE	GROUP	UNAME					
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	“CHAR”	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE MATERIAL 'SCA.MDSolver.Util.Ums '
```

*In Bulk Data:*

MATUDS	17	MATEP	MATERIAL	UELASTOM					
MATUDS	17	MATUSR	MATERIAL	HYPELA2					
	REAL	.00134	1.467+4	.03					
	INT	8	3						

Describer	Meaning
MID	Identification number of a MAT1, MAT2, MAT3, MAT8, MAT9, MATHE, MATHP, MATORT, MATUSR, MCOHE, or MATD* entries.
MTYPE	The name of the material entry. MAT1, MATEP, MATE, MATHE, MATORT, MATUSR, MATVE, or MATVP. (Character; no Default).
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. See Remark 6.and 7.(Character).
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATA <sub>i</sub>	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

- “CHAR” Keyword indicating that the following data is Character. (Character)
- CDATAi Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

#### Remarks:

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials. The GROUP must match the GROUP field of the CONNECT SERVICE FMS statement.
2. On the FMS CONNECT statement, only the CONNECT SERVICE can be used with this entry.
3. The MID must match an existing MID.
4. A CDATAi entry cannot be the Character “REAL”, “INT”, or “CHAR”
5. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
6. UNAME must be truncated to 8 characters in the bulk data field.
7. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

MTYPE	UNAME	SOL 400
MATHE	uelastomer	X
MATUSR	hypela2	X
MAT1	crplaw	X
MATF	ufail	X
MATF	uprogfail	X
MATORT	orient	X
MATUSR	umat	X
MCOHE	ucohes	X

8. In SOL 400, MATUDS is only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
9. The UMAT and UCOHES user subroutines support user defined state variables, see UDSESV entry.
10. For hyper-elastic(or plastic) materials. the deformation gradient is available for bricks and shells in the user-defined material subroutines, the variable IHYPER on the material card should be set to 1. The deformation gradient components  $F_{11}$ ,  $F_{21}$ ,  $F_{31}$ ,  $F_{12}$ ,  $F_{22}$ ,  $F_{32}$ ,  $F_{13}$ ,  $F_{23}$  and  $F_{33}$  can be found in the history variables array in positions IDATA2+1 to IDATA2+9, the positions coming right after the requested number of history variables. For shell elements, the components of the deformation

gradient are with respect to the co-rotational system for the element currently used. In this case the third row of the deformation gradient, the components  $F_{31}$ ,  $F_{32}$  and  $F_{33}$  will not be properly updated when entering the user-defined material routine. These components depend on the thickness strain increment which in turn must be determined so that the normal stress in the shell vanishes. For a given thickness strain increment  $d3$ , these three components,  $f31$ ,  $f32$  and  $f33$ , can be determined by calling the subroutine. For hyper-elastic materials there are push forward operations that can be called from within the user defined subroutines too. If the local coordinate system option is invoked ( $IDATA3=1$ ), then the deformation gradient is transformed to this local system prior to entering the user-defined material routine according to:

$$\overline{F}_{ij} = Q_{ki}^s F_{kj}$$

where  $Q_{ki}^s$  refers to a transformation between the current global and material frames. For  $IDATA$  equal to 3 one can choose to put  $IHYPER$  equal to -1 which results in that the deformation gradient is transformed according to:

$$\overline{F}_{ij} = F_{ik} Q_{kj}^r$$

where  $Q_{kj}^r$  is the transformation between the reference global and material and frames. For this latter option the spatial frame remains the global one so the stresses should be expressed in this frame of reference upon exiting the user defined routines. The suitable choice of  $IHYPER$  depends on the formulation of the material model. For shells, there is also the option of setting  $IHYPER=3$  which will make the deformation gradient computed from the nodal coordinates and in the global coordinate system. With this option the user must compute the stress in the local system of interest, whence a transformation matrix between the global and this local system is passed to the user material routines. The columns in this matrix correspond to local basis vectors expressed in global coordinates, and this is the system that stress needs to be computed in. The user must be aware that since the deformation gradient is calculated directly from the element deformation it may not be consistent with the theory of the element that is used for the material. Also, the thickness used in the calculations is constant and there is currently no thickness change treatment for this option. In the following, a Neo-Hookean material is used as an example of the usage of the deformation gradient in user-defined materials. With  $\lambda$  and  $\mu$  being the Lamé parameters in the linearized theory, the strain energy density for this material is given by:

$$\psi = \frac{1}{2} \lambda [\ln(\det F)]^2 - \mu [\ln(\det F)] + \frac{1}{2} \mu [\text{tr}(F^T F) - 3]$$

meaning that the Cauchy stress can be expressed as:

$$\sigma = \frac{1}{\det F} [\lambda \ln(\det F) I + \mu (F^T F - 1)]$$



**MATUDS****Material User Defined Service or Subroutine - SOL 600**

Allows the user to provide material routines for use with enhanced material models in SOL 600.

**Format:**

1	2	3	4	5	6	7	8	9	10
MATUDS	MID	MTYPE		UNAME	IOPT				
	“INT”	IDATA1	IDATA2	IDATA3	IDATA4	IDATA5	IDATA6	IDATA7	
		IDATA8	IDATA9	...	...	IDATA <sub>n</sub>			
	“REAL”	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	
		RDATA8	RDATA9	...	...	RDATA <sub>n</sub>			
	“CHAR”	CDATA1	CDATA2	...	...	CDATA <sub>n</sub>			

**Examples:**

MATUDS	17	MATEP		uelastomer	2				
MATUDS	33	MATHYP		hypela2	13				
	REAL	1.0E-4	2.0E-6	4.8E-6	38.6		1.2E-4		
	INT	1	3	51	52	53	54		
		55	56						

Describer	Meaning
MID	Material property identification number that matches the identification number on a MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, or MATHE, primary material entry or a MATVE, MATVP, MATEP, MATF, associated material entry for structural analysis. MAT4 and MAT5 material entries for heat analysis. COHESIV for Cohesive zone modeling. (Integer > 0)
MTYPE	The name of the material entry. MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHE, MATG, MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, MAT4, MAT5, COHESIV. (Character; no Default)
UNAME	A primary or secondary name to identify the user subroutine. See Remark 6. (Character; no Default)
IOPT	A user subroutine option flag. See Remark 9. (Character; Default blank)
“INT”	Keyword indicating that the following data is integer. (Character)
IDATA <sub>i</sub>	Additional user supplied Integer data not already existing on the specified MAT entry. (Integer; no Default)
“REAL”	Keyword indicating that the following data is real. (Character)
RDATA <sub>i</sub>	Additional user supplied Real data not already existing on the specified MAT entry. (Real; no Default)

“CHAR” Keyword indicating that the following data is Character. (Character)  
 CDATAi Additional user supplied Character data not already existing on the specified MAT entry. (Character; no Default)

Remarks:

1. This entry triggers the call to a user material subroutine for advanced nonlinear materials.
2. If the matching material entry is not present, the RDATA and IDATA fields may be used to construct the applicable material entry in which case the real and integer field values are in the order shown for the material entry (MATxxx and MATTxxx). The applicable material entries will be internally constructed from the MID, RDATA and IDATA fields.
3. Certain user subroutines, such as MATHE, require integer or real data input as specified in the User Defined Services document.
4. This entry triggers a call to a user subroutine.
5. The following user subroutines are currently available for user convenience. See the User Defined Services document for details.

MTYPE	EVAL	SOL 600
MATHE	uelastomer	X
MATVP	crplaw	X
	vswell	X
	crpvis	X
	ucrplw	X
	uvscpl	X

Material-related structural and heat transfer user subroutines documented in Marc Volume D “User Subroutines and Special Routines” may be entered using MATUDS in addition to those tabulated above. They may alternatively be entered using Bulk Data entry [USRSUB6](#).

6. If IDATAi values are defined they are stored in common block /userii-mid/ ival(1000) Up to 1000 integer values may be specified. The userii common block would then be placed in the user subroutine along with other coding.
7. If RDATAi values are defined they are stored in common block /userr8-mid/ rvals(1000) Up to 1000 real\*8 values may be specified. The userr8 common block would then be placed in the user subroutine along with other coding.
8. The MATUDS entry may be used instead of the USRSUB6 entry. Both entries should not be used in the same run. When USERSUB is used, flags such as IUSER on the MATVP entry or parameters that are needed will be filled in automatically although they may also be entered if desired. EVAL will be stored as a character\*16 name in common block /userch/
9. This name identifies the user subroutine name to be called.



10. IOPT is a user subroutine option flag required by certain MAT<sub>xxx</sub> entries. See the applicable MAT<sub>xxx</sub> entry for values IOPT may use.

**MATUSR**

## Defines Generic Material Properties for Hypoelastic or User Material Model

Specifies user-defined, generic material properties for hypoelastic material models in SOL 600 and user defined material models in SOL 400 only. This entry is used in conjunction with the MATUDS option to activate user subroutine md\_hypela2 (SOL 400) or hypela2 (SOL 600) for planar, shell or solid elements. For nonintegrated beam elements it is used in conjunction with user subroutine ubeam (SOL 600).

Format:

1	2	3	4	5	6	7	8	9	10
MATUSR	MID	IPREF	IKINEM	RHO	A1	A2	A3	TREF	
	GE	ST	SC	SS					

Example:

MATUSR	20	1	2	1.0E-4	2.0E-6				
--------	----	---	---	--------	--------	--	--	--	--

Describer	Meaning
MID	Identification number. (Integer > 0; no Default)
IPREF	Flag to indicate that user defined material is given with respect to user defined coordinate system. (Integer; Default = 0; see Remark 14.) 0 User stress-strain law is isotropic (Default) 1 User stress strain is with respect to user defined coordinate system
IKINEM	Flag to indicate which additional kinematic quantity is passed into user subroutine. Incremental and total strains are always passed into user subroutine. (Integer; Default = 0, see Remark 3.) 0 Pass in incremental strain $\Delta\epsilon$ only. 1 Pass in $\Delta\epsilon$ , deformation gradient (F) and rotation (R) 2 Pass in $\Delta\epsilon$ , deformation gradient (F) and stretch ratios ( $\dot{\lambda}$ ) 3 Pass in $\Delta\epsilon$ , F, R and $\dot{\lambda}$ 13 Pass in $\Delta\epsilon$ , F, R and $\dot{\lambda}$ . Kinematic quantities are calculated at the mid increment if large strain is requested resulting in logarithmic strains. 23 Pass in $\Delta\epsilon$ , F, R and $\dot{\lambda}$ . Kinematic quantities are calculated at the end of the increment if large strain is requested.
RHO	Mass density for structural analysis (Real $\geq 0$ ; no Default)
Ai	Coefficient of thermal expansion. (Real; no Default)
TREF	Reference temperature for the calculation of thermal loads, or a temperature-dependent thermal coefficient. See Remarks 6, 8, and 15. (Real; Default = 0.0)

Describer	Meaning
GE	Structural element damping coefficient. See Remarks 7, 8, 9. and 15.
ST,SC,SS	Stress limits for tension, compression and shear are optionally supplied ( these are used only to compute margins of safety in certain elements) and have no effect on the computational procedures. See Remark 15. (Real; Default = 0.0)

#### Remarks:

1. This option is used together with MATUDS to define a generic material model. MATUDS is not required for SOL 600.
2. Only enter those fields necessary for the analysis being performed.
3. For SOL 400, IKINEM should be greater than 0 and user subroutine HYPELA2 is used. For SOL 600, if IKINEM=0, then subroutine HYPELA is used, else HYPELA2.
4. The generic material model is able to represent a nonlinear material behavior. For this constitutive theory, it is assumed that

$$\dot{\sigma}_{ij} = L_{ijkl} \dot{\epsilon}_{kl} + g_{ij}$$

where  $L$  is a function of the mechanical strain and  $g$  is a function of the temperature.

The stress and strains are true stresses and logarithmic strains, respectively, when used in conjunction with large strain requests.

When used in conjunction with PARAM,LGDISP,1 (without large strain) the above equation is expressed as

$$\dot{S}_{ij} = L_{ijkl} \dot{E}_{kl} + g_{ij}$$

where  $E$  and  $S$  are the Green-Lagrange strain and second Piola-Kirchhoff stress, respectively.

5. For the MD\_HYPELA2 user subroutine, in order to provide an accurate solution, a tangent stiffness should be evaluated at each iteration. In addition, the total stress should be defined as its exact value at the end of the increment. This allows the residual load correction to work effectively. In MD\_HYPELA2, additional information is available regarding the kinematics of deformation. In particular, the deformation gradient (F), rotation tensor (R), and the eigenvalues ( $\dot{\lambda}$ ) and eigenvectors (N) to form the stretch tensor (U) are also provided. This information is available only for the continuum elements namely: plane strain, generalized plane strain (SOL 600 only), plane stress, axisymmetric, axisymmetric with twist, and three-dimensional cases.
6. TREF is used as the reference temperature for the calculation of the thermal loads. TEMPERATURE (INITIAL) may be used for this purpose, in which case TREF should be blank
7. Note that the structural damping will change with time, to obtain the damping coefficient, multiply the critical damping ratio  $C/C_0$  by 2.
8. TREF and GE are ignored if this entry is referenced by a PCOMP or a PCOMPG entry.
9. If PARAM,W4 is not specified, GE is ignored in a transient analysis. See [Parameter Descriptions](#).

10. This material model should not be used with the PSHELL option or with PBARN1, PBEAMN1 or PCOMP(G) and smeared section integration.
11. For SOL 400 the user subroutines md\_hypela2 or md\_ubeam are used. For SOL 600 user subroutine hypela or hypela2 and ubeam are used, unless a MATUDS option is also included.
12. For SOL 400 this option automatically invokes the PSHLN1, PSHLN2, PSLDN1 element formulation
13. In a thermal-mechanically coupled simulation, MAT4 or MAT5 should be used to define the material behavior.
14. For SOL 600, IPREF = 0 means user subroutines ANEXP and ORIENT are not called. IPREF = 1 means the two routines are called.
15. TREF, GE, ST, SC and SS are ignored in SOL 600.
16. If A1 is given and A2, A 3 are not then isotropic thermal expansion and  $A1 = A2 = A3$ .
17. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

**MATVE**

**Visco-Elastic Material Properties**

Specifies visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 600 and SOL 400.

Format for Model = Iso:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alpha1	G1	Td1	K1	Tv1	
	G2	Td2	G3	Td3	G4	Td4	G5	Td5	
	K2	Tv2	K3	Tv3	K4	Tv4	K5	Tv5	

Format for Model = Iso1:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal					
	G1	Td1	K1	Tv1					
	G2	Td2	K2	Tv2					
	G3	Td3	K3	Tv3					
	G4	Td4	K4	Tv4					
	G5	Td5	K5	Tv5					
	Gi	Tdi	Ki	Tvi					

Format for Model = Mooney, Ogden and Foam:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal	Wd1	Td1	Wv1	Tv1	
	Wd2	Td2	Wd3	Td3	Wd4	Td4	Wd5	Td5	
	Wv2	Tv2	Wv3	Tv3	Wv4	Tv4	Wv5	Tv5	

Format for Model = Mooney1, Ogden1 and Foam1:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model	Alphas	Alphal					
	Wd1	Td1	Wv1	Tv1					
	Wd2	Td2	Wv2	Tv2					
	Wd3	Td3	Wv3	Tv3					
	Wd4	Td4	Wv4	Tv4					
	Wd5	Td5	Wv5	Tv5					
	Wdi	Tdi	Wvi	Tvi					

Format for ORTHO:

1	2	3	4	5	6	7	8	9	10
MATVE	MID	ORTHO							
	Td1	Exx1	Eyy1	Ezz1	Vxy1	Vyz1	Vzx1		
	Gxy1	Gyz1	Gzx1						
	Td2	Exx2	Eyy2	Ezz2	Vxy2	Vyz2	Vzx2		
	Gxy2	Gyz2	Gzx2						
	Td3	Exx3	Eyy3	Ezz3	Vxy3	Vyz3	Vzx3		
	Gxy3	Gyz3	Gzx3						
	Td4	Exx4	Eyy4	Ezz4	Vxy4	Vyz4	Vzx4		
	Gxy4	Gyz4	Gzx4						
	Td5	Exx5	Eyy5	Ezz5	Vxy5	Vyz5	Vzx5		
	Gxy5	Gyz5	Gzx5						
	Tdi	Exxi	Eyyi	Ezzi	Vxyi	Vyzi	Vzxi		
	Gxyi	Gyzi	Gzxi						

Example

1	2	3	4	5	6	7	8	9	10
MATVE	1	ISO			3030	0.98			

1	2	3	4	5	6	7	8	9	10
MATVE	1	ISO1							
	3030	0.98							



Describer	Meaning
MID	Identification number of primary material (MAT1 or MAT3 or MAT8 or MATROT or MATHE) entry. (Integer > 0). See Remark 2.
Model	Selects a visco-elastic model defining time-dependent deformation behavior (Character): ISO1 for isotropic materials referenced by MAT1 (Default) (an alternate name is Linear). ORTHO if referenced by MAT3, MAT8 and MATORT. MOONEY1 for Mooney-Rivlin model if referenced by MATHE. OGDEN1 for Ogden model if referenced by MATHE. FOAM1 for foam model if referenced by MATHE (SOL 400 only). See Remarks 2., 3., 4., 6. and 8.
Alphas	Solid coefficient of thermal expansion (Real; Default = 0). See Remark 6.
Alphal	Liquid coefficient of thermal expansion (Real; Default = 0). See Remark 6.
Gi	Shear modulus for $i^{\text{th}}$ term in Prony series (Real > 0.; Default = 0).
Ki	Bulk modulus for $i^{\text{th}}$ term in Prony series (Real > 0.; Default = 0).
Wdi	Multiplier (scale factor) for $i^{\text{th}}$ term deviatoric behavior in Prony series (Real $\geq$ 0.; Default = 0). for Hyperelastic Material.
Wvi	Multiplier (scale factor) for $i^{\text{th}}$ term volumetric behavior in Prony series (Real $\geq$ 0.; Default = 0.). See Remark 7.
Tdi	Defines time constants for deviatoric behavior in Prony series (Real $\geq$ 0.; Default = 0). See Remark 7.
Tvi	Defines time constants for volumetric behavior in Prony series (Real $\geq$ 0.; Default = 0). See Remark 7.

Remarks:

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MAT9, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. The time-dependent behavior in the viscoelastic material is modeled by a Prony series expression for both small and large strain problems. The stress relaxation behavior can be modeled by relaxation functions of the shear modulus and bulk modulus and total strain energy in terms of a series of exponential decay terms, which is known as the Prony series. This is equivalent to the generalized Maxwell model, which consists of many Maxwell models connected in parallel along with an elastic spring representing a long-term behavior. The constitutive behavior of viscoelasticity depends not only on the current state of stress and strain, but also on the entire history of the development of these states. Such a behavior is most readily expressed by hereditary or Duhamel integral. The Prony series is a discrete form of this hereditary integral.
3. ISO1 supports both volumetric and deviatoric viscoelastic behavior by allowing deviatoric terms (Gi and Tdi) and volumetric terms (Ki and Tvi) in the Prony series. The shear and bulk moduli for isotropic material (ISO1) can be expressed in a Prony series as

$$G(t) = G^\infty + \sum_{i=1} G_i e^{-t/T_{di}}$$

$$K(t) = K^\infty + \sum_{i=1} K_i e^{-t/T_{di}}$$

where instantaneous values are given by:

$$G^0 = G^\infty + \sum_{i=1} G_i$$

$$K^0 = K^\infty + \sum_{i=1} K_i$$

In terms of instantaneous values, the shear and the bulk modulus relaxation can also be written as.

$$G(t) = G^0 - \sum_{i=1} G_i (1 - e^{-t/T_{di}})$$

$$K(t) = K^0 - \sum_{i=1} K_i (1 - e^{-t/T_{di}})$$

If a Prony series is selected, at least one set of modulus and time constant must be provided i.e., (G1, Td1) and /or (K1, Tv1).

4. ORTHO supports only deviatoric viscoelastic behavior by allowing deviatoric terms (Gij and Tdi) in the Prony series. For orthotropic material behavior, the relaxation coefficients are applied to the shear modulus as

$$G_{ij}(t) = G_{ij}^0 - \sum_{n=1} G_{ij}^n (1 - e^{-t/T_{di}})$$

where  $G_{ij}^n$  is the shear modulus for  $n^{\text{th}}$  term is Prony series (Real > 0; Default = 0) for Orthotropic Material. (Here  $G_{ij}$  imply  $G_{xy}$ ,  $G_{yz}$  and  $G_{zx}$  as mentioned in ORTHO table).

5. In case of a viscous hyperelastic material, total strain energy be expressed as a Prony series expansion in terms of energy functional

$$\Psi = \Psi^\infty + \sum_{n=1} \Psi_n e^{-t/T_{di}}$$

Above equation can also be expressed as a function of the elastic strain energy density for instantaneous deformations  $\Psi^\infty$ .

$$\psi = \psi^{\infty} + \sum_{n=1} W_i \psi^0 e^{-t/T_i}$$

where  $W_i$  are weighting factors  $\psi_i/\psi^0$  and  $T_i$  are time constants. Two Prony series can be formed

based on deviatoric ( $\psi^d$ ) and volumetric strain energy ( $\psi^v$ ). If a viscous hyperelastic Prony series is selected, at least one pair of weighting factor and time constant must be provided: ( $W_{d1}$ ,  $T_{d1}$ ) and /or ( $W_{v1}$ ,  $T_{v1}$ ). OGDEN support both deviatoric ( $W_{di}$ ,  $T_{di}$ ) and volumetric strain energy ( $W_{vi}$ ,  $T_{vi}$ ) Prony series. However, Mooney supports only deviatoric strain energy ( $W_{di}$ ,  $T_{di}$ ) Prony series. Foam is defined in total strain energy density function and specified through  $W_i$  and  $T_i$ .

6. If ALPHAS or ALPHAL is specified, the thermal expansion coefficient specified in MAT1, MATT1, MATHE, MATTHE, MATORT or MATTORT will be ignored. Use of ALPHAS or ALPHAL requires the MATTVE Narayanaswami model.
7. If the weighting factor is left blank, the relaxation function corresponding to that coefficient is omitted.
8. In SOL 400, MATVE is used only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MSC Nastran 2010, if the property extensions were missing, then the MATVE data was not considered in the element's formulation. If the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS,SPROPMAP,-1 in the bulk data.
9. FOAM (FOAM1) no difference between deviatoric and volumetric multipliers and relaxation time. Program will use deviatoric input to apply on entire strain energy function.

#### Notes:

1. The Prony series uses values of  $G_i$  or  $K_i$  for Isotropic and Orthotropic materials. It uses weighting ( $W_i$ ) functions for Mooney, Ogden and Foam materials.
2. The instantaneous material properties are specified on primary material entries (i.e., for MAT1 or MAT3 or MAT8 or MATORT or MATHE)
3. In the MATVE entries, material properties with the Prony series parameters are specified.
4. MATVE with "ISO", "MOONEY", "OGDEN", "FOAM" models are similar to the corresponding "ISO1", "MOONEY1", "OGDEN1", "FOAM1" models with the only exception being that the former models are limited to 5 terms in the Prony series. With the latter models, there is no limit to the number of terms in the Prony series.

**MATVE**

Visco-Elastic Material Properties- SOL700

Specifies isotropic visco-elastic material properties to be used for quasi-static or dynamic analysis in SOL 700. Used in SOL 700 only.

Format (for types Iso, Mooney, Ogden and Foam):

1	2	3	4	5	6	7	8	9	10
MATVE	MID	Model			G0	GI	BETA	VISC	

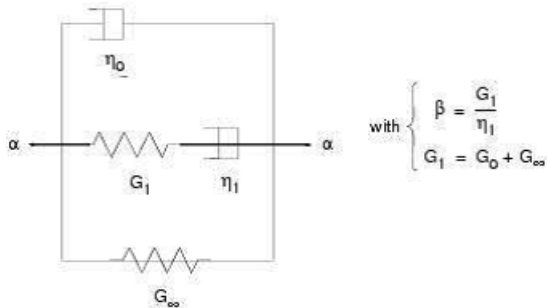
Example:

MATVE	1	ISOLIN			8.E7	1.E7	0.1		
-------	---	--------	--	--	------	------	-----	--	--

Describer	Meaning
MID	Identification number of MAT1 or MATHE entry (Integer > 0; required).
Model	Selects a visco-elastic model defining time-dependent deformation behavior (Character): <b>ISOLIN</b> for linear isotropic materials referenced by MAT1
G0	Short-time shear modulus. (Real; default=0.0)
GI	Long-time shear modulus. (Real; default=0.0)
BETA	Decay constant. (Real; default=0.0)
VISC	Shear viscosity constant. (Real; default=0.0)

Remarks:

1. The spring-damper analog of this model is:



2. The deviatoric stress is given by

$$\sigma'_{ij}(t) = 2G_I \varepsilon'_{ij}(t) + 2 \int_0^t G(t-\tau) \frac{\partial \varepsilon'_{ij}(\tau)}{\partial \tau} d\tau + 2\eta_0 \frac{\partial \varepsilon'_{ij}(t)}{\partial t}$$

$$G(t - \tau) = G(G_0 - G_I)e^{-\beta(t - \tau)}$$

The above equation for the deviatoric stress is the integral form of the differential equation

$$\dot{\sigma}'_{ij} + \beta\sigma'_{ij} = 2\eta_0\ddot{\varepsilon}'_{ij} + (2\eta_0\beta + 2G_0)\dot{\varepsilon}'_{ij} + 2G_I\beta\varepsilon'_{ij}$$

A special case is for which is often written  $\eta_0 = G_I = 0$

$$\dot{\varepsilon}'_{ij} = \dot{\varepsilon}'_{ijelastic} + \dot{\varepsilon}'_{ijviscous} = \frac{\dot{\sigma}'_{ij}}{2G_0} + \frac{\beta}{2G_0}\sigma'_{ij}$$

## MATVP

## Viscoplastic or Creep Material Properties - SOL600

Specifies viscoplastic or creep material properties to be used for quasi-static analysis.

Primary Format: PARAM, MARCMATT, 1 required

1	2	3	4	5	6	7	8	9	10
MATVP	MID	A	IT3D	M	N	P	Q	IUSER	

Alternate Format: PARAM, MARCMATT, -1 required

1	2	3	4	5	6	7	8	9	10
MATVP	MID	FORM	COEFF	STRESS	STRAIN	TEMP	TIME	IUSER	

Example 1: TABL3D used to specify computation of A.

MATVP	101	1.0	20	1.5	1.1	1.0	1.0		
-------	-----	-----	----	-----	-----	-----	-----	--	--

Example 2: Coefficient A provided with exponents for stress, strain, temperature, and time.

MATVP	102	.00375		1.5	1.1	1.0	1.0		
-------	-----	--------	--	-----	-----	-----	-----	--	--

Example 3: Primary format user subroutine **crplaw.f** and **vswell.f** are used.

MATVP	100							12	
-------	-----	--	--	--	--	--	--	----	--

Example 4: Alternate format with four table inputs

MATVP	10	TABLE	3.5-15	101	102	103	104		
-------	----	-------	--------	-----	-----	-----	-----	--	--

Example 5: Alternate User subroutine **ucrplw.f** is used

MATVP	75	USER						4	
-------	----	------	--	--	--	--	--	---	--

Descriptor	Meaning
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. See Remark 1. (Integer > 0)
<b>Power-Law Creep Model</b>	
A	Enter the coefficient A in for the equations of Remark 2. (this value could alternatively be built into the table referenced using IT3D in which case A should be set to 1.0. If A=0.0, user subroutine(s) designated by the flag in field 9 may be used to define the creep law and fields 4-8 will be ignored if entered. (Real; no Default)
IT3D	ID of a TABL3Di entry that defines the variation with respect to stress, strain, temperature and/or time per the equations discussed in Remark 2. The exponents are provided in the next 4 fields. (Integer > 0; Default = 0)
M	Exponent m for stress in equation in Remark 2. (Real; Default = 0.0)

Describer	Meaning
N	Exponent n for strain in equation in Remark 2. (Real; Default = 0.0)
P	Exponent p for temperature in equation in Remark 2. (Real; Default = 0.0)
Q	Exponent q for time in equation in Remark 2. (Real; Default = 0.0)
IUSER	Packed list designating which user subroutines (if any) apply to this analysis. The user must confirm that the selected user subroutines are appropriate for the current analysis and are consistent with other entries in the model. (Integer; Default = 0)
	1            crplaw.f
	2            vswell.f
	3            crpvis.f
	4            ucrplw.f
	5            uvscpl.f
	(examples, if crplaw.f is used, enter 1 if both crplaw.f and vswell.f are required, enter 12).

**Power-Law Creep Model Alternate Format**

FORM	Selects a creep data input form defining creep strain rate from the options listed below. (Character; no Default):
	“POWER”    for exponent input in power law form
	“TABLE”    for piece-wise linear curve input in TABLEM1 entry.
	“USER”     user subroutine crplaw.f will be used instead of the values/table ID’s on the entry (if this form is used, fields 4-8 should be ignored if entered).
	See Remark 2.
COEFF	Specifies the coefficient value, A in equation. (Real > 0)
STRESS	Identification number of TABLEM1 (Integer > 0) for the function f or exponent m (Real > 0.) for an effective stress function, depending on the Form field.
STRAIN	Identification number of TABLEM1 (Integer > 0) for the function g or exponent n (Real; Default = 0.) for an equivalent creep strain function, depending on the Form field.
TEMP	Identification number of TABLEM1 (Integer > 0) for the function h or exponent p (Real; Default = 0) for a temperature function, depending on the Form field.
TIME	Identification number of TABLEM1 (Integer > 0) for the function K or exponent q (Real > 0; Default = 1) for a time function, depending on the Form field.

## Remarks:

1. The MATVP entry is activated if a MAT1, MAT2, MATORT, or MAT9 entry with the same MID is specified in an implicit nonlinear analysis. This creep capability is available for isotropic, orthotropic, and anisotropic elasticity, which can be coupled with plasticity using MATEP entry. Coupling with plasticity is allowed only for selected plasticity models, which include von Mises, Hill's anisotropy (creep stays isotropic), and Mohr-Coulomb models. However, viscoelasticity (MATVE) cannot be combined with viscoplasticity (MATVP).
2. The creep behavior of the material is expressed in terms of creep strain rate as a product of a number of terms (functions of effective stress, equivalent creep strain, temperature, and time) in either exponential form or piecewise linear curves, i.e.,

$$\dot{\bar{\varepsilon}}^c = A \bar{\sigma}^{-m} (\bar{\varepsilon}^c)^n T^p (qt^{q-1}) \text{ or}$$

$$\dot{\bar{\varepsilon}}^c = Af(\bar{\sigma})g(\bar{\varepsilon}^c)h(T)\frac{dK(t)}{dt}$$

Power-Law creep (exponential form) can be used with either explicit or implicit creep methods. The choice of explicit or implicit creep is made via the NLMOPTS option.

- a. Explicit creep can be combined with other plasticity options defined through the MATEP card. The supported plasticity models include Von Mises, Hill's Anisotropy (creep remains isotropic) and Mohr-Coulomb models. The creep and plastic strains are treated separately with the algorithm handling the creep in an explicit manner and handling the plasticity in an implicit manner.
- b. Implicit power law creep can be combined with the Von Mises yield criterion defined through the MATEP card. Both the creep and plasticity are handled simultaneously through an implicit viscoplasticity algorithm and a combined inelastic strain is computed.
- c. Note that for the explicit creep strain-rate evaluation, N should only be specified if some creep strain is already present in the model. Otherwise, N should be zero (blank), or implicit creep should be selected in CREEP of NLMOPTS.
- d. The dependence of time (Q) is specified as function of total equivalent creep strain i.e.,  $\dot{\bar{\varepsilon}}^c = At^q \bar{\varepsilon}^c = Aqt^{q-1}$ .

The functions  $f$ ,  $g$ ,  $h$  and  $K$  are specified as piece-wise linear functions in a tabular form using TABLEM1 entry, if the Table Form is selected. Notice that the last term in time shows function  $K$  for the equivalent creep strain in terms of time, instead of creep strain rate.

The creep strain from the creep material is a permanent strain unlike the creep strain for materials using the CREEP Bulk Data entry. As such, this creep material may be classified as viscoplastic material. This creep capability is provided for the primary and the secondary creep behavior, because the tertiary creep involves material instability such as necking.

3. See associated Bulk Data entry, MPCREEP.
4. The alternate format is determined by field 3 with a character string POWER, TABLE, or USER.
5. There must be a non-blank entry in field 3 for either format.



6. MATVP is used in conjunction with NLPARM with no restrictions on the value of the total time ( $NINC * DT$ ). NLAUTO may be added to specify under stress, strain or other criteria and/or for advanced convergence controls not available on the first line of NLPARM. Additional convergence and solver controls are available using NLSTRAT.

## MATVP

## Viscoplastic or Creep Material Properties - SOL400

Specifies viscoplastic or creep material properties to be used for quasi-static analysis.

Format: Power-Law Creep Model

1	2	3	4	5	6	7	8	9	10
MATVP	MID	A	IT3D	M	N	P	Q	IUSER	

Example 1: TABL3D used to specify computation of A for Power-Law Creep Model

MATVP	10	1.0	20	1.5	1.1	1.0	1.0		
-------	----	-----	----	-----	-----	-----	-----	--	--

Example 2: Coefficient A provided with exponents for stress, strain, temperature, and time for Power-Law Creep Model

MATVP	10	.00375		1.5	1.1	1.0	1.0		
-------	----	--------	--	-----	-----	-----	-----	--	--

Example 3: User subroutine crplaw.f is used for Power-Law Creep Model

MATVP	10							1	
-------	----	--	--	--	--	--	--	---	--

Format: Anand Solder Creep Model

1	2	3	4	5	6	7	8	9	10
MATVP	MID	ANAND	PREXF	ACTEN	MULST	STNRT	SATCO	STNSA	
	HRCN	STNHR	DEFRS						

Example 1: Anand Solder Creep Model

MATVP	2	ANAND	2.23+4	15583	7	0.143	72.73	0.00437	
	1787.02	3.73	15.09						

Describer	Meaning
MID	Identification number of MAT1, MAT2, MATORT or MAT9 entry. See Remark 1. (Integer > 0)
<b>Power-Law Creep Model</b>	
A	Enter the coefficient A in for the equations of Remark 2. (this value could alternatively be built into the table referenced using IT3D in which case A should be set to 1.0. If A=0.0, user subroutine(s) designated by the flag in field 9 may be used to define the creep law and fields 4-8 will be ignored if entered. (Real; no Default)
IT3D	ID of a TABL3Di entry that defines the variation with respect to stress, strain, temperature and/or time per the equations discussed in Remark 2. The exponents are provided in the next 4 fields. (Integer > 0; Default = 0)

Describer	Meaning
M	Exponent m for stress in equation in Remark 2. (Real; Default = 0.0)
N	Exponent n for strain in equation in Remark 2. (Real; Default = 0.0)
P	Exponent p for temperature in equation in Remark 2. (Real; Default = 0.0)
Q	Exponent q for time in equation in Remark 2. (Real; Default = 0.0)
IUSER	IUSER=1 if user subroutine crplaw.f is desired. (Integer; Default = 0)

### Anand Solar Creep Model

ANAND	A keyword specifying the Anand solder creep model. The Anand solder creep model consists of a simple set of constitutive equations for large, isotropic, viscoplastic problems. If this key word is used, next 9 fields for parameters of Material properties must be filled with values. Please note that Anand solder creep model is only for isotropic materials. See Remark 6, 7, and 8.
PREXF	Pre-exponential factor, $A(s^{-1})$ .
ACTEN	Activation Energy, $Q (J/mol)$ .
MULST	Multiplier of stress, $\xi$ .
STNRT	Strain rate sensitivity of stress, m.
SATCO	Deformation resistance saturation coefficient, $\hat{s}$ (MPa).
STNSA	Strain rate sensitivity of saturation, n.
HRCN	Hardening constant, $h_0$ (MPa).
STNHR	Strain rate sensitivity of hardening, a.
DEFRS	Initial value of deformation resistance, $s_0$ (MPa).

### Remarks:

1. This MATVP entry is activated if a MAT1, MAT2, MATORT, or MAT9 entry with the same MID is specified in an implicit nonlinear analysis. This creep capability is available for isotropic, orthotropic, and anisotropic elasticity, which can be coupled with plasticity using MATEP entry. Coupling with plasticity is allowed only for selected plasticity models, which include von Mises, Hill's anisotropy (creep stays isotropic), and Mohr-Coulomb models. However, viscoelasticity (MATVE) cannot be combined with viscoplasticity (MATVP).

Note that ANAND solder creep model may be associated with isotropic material, i.e., MAT1, only.

2. The creep behavior of the material is expressed in terms of creep strain rate as a product of a number of terms (functions of effective stress, equivalent creep strain, temperature, and time) in exponential form, i.e.,

$$\dot{\epsilon}^c = A \bar{\sigma}^{-m} (\bar{\epsilon}^c)^n T^p (qt^q - 1)$$

Power-Law creep (exponential form) can be used with either explicit or implicit creep methods. The choice of explicit or implicit creep is made via the NLMOPTS option.

- a. Explicit creep can be combined with other plasticity options defined through the MATEP card. The supported plasticity models include Von Mises, Hill's Anisotropy (creep remains isotropic) and Mohr-Coulomb models. The creep and plastic strains are treated separately with the algorithm handling the creep in an explicit manner and handling the plasticity in an implicit manner.
- b. Implicit power law creep can be combined with the Von Mises yield criterion defined through the MATEP card. Both the creep and plasticity are handled simultaneously through an implicit viscoplasticity algorithm and a combined inelastic strain is computed.
- c. The dependence of time (Q) is specified as function of total equivalent creep strain i.e.,  

$$\dot{\varepsilon}^c = A t^q.$$

Note that the creep strain rate is internally computed as:

$$\dot{\varepsilon}^c = A t^q$$

The creep strain from the creep material is a permanent strain unlike the creep strain for materials using the CREEP Bulk Data entry. As such, this creep material may be classified as viscoplastic material. This creep capability is provided for the primary and the secondary creep behavior, because the tertiary creep involves material instability such as necking.

3. See associated Bulk Data entry, NLMOPTS
4. For NLSTEP adaptive stepping, use of TABSCTL to specify user criteria on creep strains and stresses is also recommended. MATVP is used only supported for elements with property extensions. This implies that for such elements, PBAR / PBARL, PBEAM / PBEAML, PCOMP / PCOMPG, PROD, PSHEAR, PSHELL, PLPLANE and PSOLID should be associated with PBARN1, PBEMN1, PSHLN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 respectively. Note that, prior to MD Nastran 2010, if the property extensions were missing, then the MATVP data was not considered in the element's formulation. From MD Nastran 2010 onwards, if the property extensions are missing, they are automatically added with appropriate settings by the program unless the user has set NLMOPTS, SPROPMAP, -1 in the bulk data.
5. When used with PBAR, PBEAM, PBEAML, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL or PLPLANE (with BEH=PSTRS) the explicit formulation must be used, set VALC3=0 on NLMOPTS.
6. Flow equation for Anand Solder model is given by:

$$\dot{\varepsilon}_p = A \exp\left(-\frac{Q}{RT}\right) \left[ \sinh\left(\xi \frac{\sigma}{s}\right) \right]^{1/m}$$

where A is the pre-exponential factor; Q is the activation energy; m is the strain rate sensitivity;  $\xi$  is the multiplier of stress, R is the gas constant (8.314 J/mol/K); T is the absolute temperature; s is deformation resistance with an initial value  $s_0$ . The evolution of deformation resistance is given by

$$\frac{ds}{dt} = \left\{ h_0 |B|^a \frac{B}{|B|} \right\} \frac{d\varepsilon_p}{dt}$$

where;

$$B = 1 - \frac{s}{s''}$$

and

$$s'' = \hat{s} \left[ \frac{1}{A} \frac{d\varepsilon_p}{dt} \exp\left(\frac{Q}{RT}\right) \right]^n$$

where  $h_0$  is hardening constant;  $a$  is the strain rate sensitivity of hardening;  $\hat{s}$  is the deformation resistance saturation coefficient,  $n$  is the strain rate sensitivity of saturation.

7. To activating creep analysis with ANAND creep model, the following parameters are mandatory:
  - a. In Bulk Data Card NLMOPTS: keyword “CREEP” and “LRGSTRN” must be selected with value:
    - “LRGSTRN” with value 1 for updated Lagrange formulation, Hypo-elasticity and additive plasticity with mean normal value.
    - “CREEP” with valc3=1, Implicit Creep on; and valc4=0/1/2, for elastic tangent as default (Blank or 0), 1 for secant tangent, and 2 for Radial Return, respectively.
  - b. In Bulk Data Card NLSTEP: CREEP in “GENERAL” is required for ANAND creep model.
  - c. In Bulk Data Card MATVP: material parameters of ANAND creep model are defined in MATVP card with keyword “ANAND” in the 3<sup>rd</sup> field and following 9 fields.
8. NLSTRESS in Case Control will output the equivalent creep strain on grid point. To output the component of the creep strain, NLOUT in NLSTRESS Case Control command must be set and request Creep output in Bulk Data Card.

In case control section:

```
NLSTRESS (NLOUT=num) = ALL
```

In bulk data section:

```
NLOUT, num, TOTTEMP, CCRPSTRN
```

## MAUXCMD Auxiliary Command to Spawn one Nastran Process from Another - SOL 600

Defines auxiliary command to spawn on Nastran process from another Nastran process in SOL 600.

Format:

1	2	3	4	5	6	7	8	9	10
MAUXCMD	Command	Cont	Cont	Cont	Cont	Cont	Cont	Cont	

Example:

MAUXCMD	/mscinst/nast2008/bin/mdnast2008	
---------	----------------------------------	--

Describer	Meaning
Command	Enter the command you normally use to execute Nastran from the command line (Character; no default; limit of 64 characters)

Remark:

1. At present, this entry may only be used for CINTC when PARAM,MBLADCMD,10 is set.

**MBOLT** Defines a Bolt For Use in SOL 600 in Countries Outside the USA

Defines a bolt for use in SOL 600 in countries outside the USA. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLT	ID	GRIDC	V1	V2	V3				
	GRIDS	G1	G2	G3	G4	G5	G6	G7	
		G8	G9	Etc.					
	ELEMS	E1	E2	E3	E4	E5	E6	E7	
		E8	E9	Etc.					

Example:

MBOLT	100	1025	0.0	1.0	0.0				
	GRIDS	101	102	103	104	105			
	ELEMS	1	2	3	4	5	6	7	
		10							

Describer	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
V1	First component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
V2	Second component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
V3	Third component of vector normal to the bolt cross section in basic coordinate system. (Real; Default = 0.0)
GRIDS	Enter the character string GRIDS to define the start of the entry that defines all of the grids at the bolt intersection cross section (do not enter the ID for GRIDC). (Integer; no Default)
G1, G2, etc.	Grid IDs of the grid points at the bolt intersection. (Integer; no Default)
ELEMS	Enter the character string ELEMS to define the start of the entry that defines all of the elements at the bolt intersection cross section lying on the side of the cross section corresponding to the negative normal direction. (Integer; no Default)
E1, E2, etc.	Element IDs of the grid points at the bolt intersection. (Integer; no Default)

**Remarks:**

1. Enter as many GRIDS and ELEMS lines as necessary to define all the grid and element IDs in the cross section.
2. All GRIDS must precede all ELEMS.
3. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersecting surfaces are described by this entry.
4. Specify a different MBOLT entry for each individual bolt.
5. This entry can only be used with Marc 2003 or later outside the USA.
6. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation).
7. This entry maps to Marc's CROSS SECTION entry.
8. GRIDC must already exist. It is not generated by the MBOLT entry. It typically would not be used by any other element, MPC, etc.



## MBOLTUS

### Defines a Bolt for Use in SOL 600 in the USA

Defines a bolt for use only in SOL 600 and only in the USA.

Format:

1	2	3	4	5	6	7	8	9	10
MBOLTUS	ID	GRIDC							
	TOP	GT1	GT2	GT3	GT4	GT5	GT6	GT7	
		GT8	GT9	Etc.					
	BOTTOM	GB1	BG2	BG3	BG4	GB5	GB6	GB7	
		GB8	GB9	Etc.					

Example:

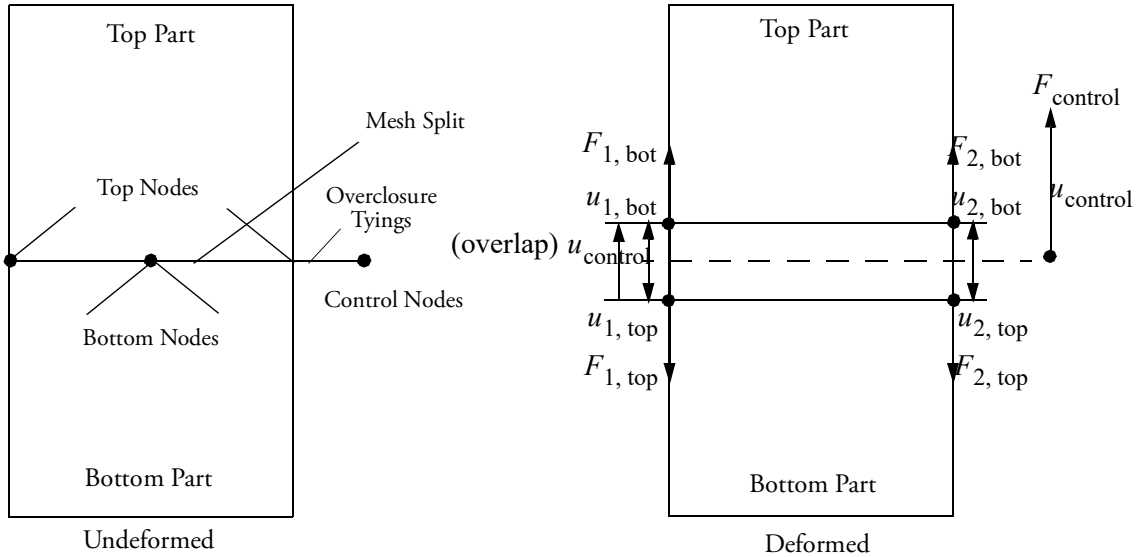
MBOLTUS	100	1025		1.0					
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Describer	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC) (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection (Integer; no Default)

Remarks:

1. Enter as many GRIDS as necessary (up to the limit of 999 top grids and 999 bottom grids) to define all the grids at the “top” and “bottom” of the bolt intersection with the structure.
2. The bolt itself is not actually modeled, just the intersecting surfaces. The nodes and elements where the bolt goes through the intersection surfaces are described by the entry.
3. Specify a different MBOLTUS entry for each individual bolt.
4. This entry can only be used with Marc 2003 or later.

5. For more information, please consult the Marc Theoretical Manual (Volume A of the Marc documentation)
6. This entry maps to Marc's TYING type 69.
7. GRIDC must already exist. It is not generated by the MBOLTUS entry. It typically would not be used by any other element, MPC, etc.
8. The following figure indicates the required modeling and data input.



## MCHSTAT      Option to Change State Variables for SOL 600

This option provides various ways of changing state variables throughout the model. It is required if Bulk Data entry, MTCREEP is used. It may also be used to enter temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file in which case MINSTAT is used to define the initial stress-free temperatures and MCHSTAT is used to define the temperatures that cause thermal strains.

Format:

1	2	3	4	5	6	7	8	9	10
MCHSTAT	ID	IDV	IOPT		INCR	NSET	IFORM	IPRT	
		NPST							
	"FILE"	Name							
	"ELEM"	ELE1	ELE2	INT1	INT2	LAY1	LAY2	VAL	
	"STATE"	NS	IS1	IS2	IS4	IS5	IS6		
		IS7	IS8	IS9	etc.				

Example:

MCHSTAT	0	1	3		1		0		
		9							
	FILE	change_state_example01							

Describer	Meaning
ID	ID of a matching Case Control MCHSTAT command. If ID=0 this entry is in the Marc Model definition, otherwise it is in Marc's History Definition for the applicable subcase. (Integer; no Default)
IDV	State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.
IOPT	Option of how to enter the data. (Integer; Default = 3) (2,2) 1      Use the "ELEM" continuation line for as many elements as necessary 2      Enter the data using user subroutine NEWSV 3      Read the data from a t16 or t19 file (see IFORM)
INCR	Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)
NSET	Number of sets to be read to define the temperature history if Marc's iteration method is controlled using MOTHERM or MTCREEP. (Integer; no Default) (2,6)
IFORM	Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) 0      Use binary (t16) file

Describer	Meaning
	1 Use formatted (t19) file
IPRT	Enter a value of 1 to suppress printing of state variable values defined in user subroutine NEWSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)
NPST	Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9) See MARCOUT for a list of the post codes.
FILE	Enter the character string FILE if IOPT=3. (Character, no Default; Required if IOPT=3)
NAME	Enter the file root name of the previous heat transfer job without any extension. For example if the previous heat transfer job was heat32.dat or heat32.bdf, enter heat32 The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file. (Character, no Default.
ELEM	Enter the character string ELEM to start a list of elements and associated values if IOPT=1 (Character)
ELE1	First element with value VAL. (Integer; no Default; Required) (3,1)
ELE2	Last element with value VAL. (Integer; Default = ELE1) (3,2)
INT1	First integration point with value VAL. (Integer; no Default; Required) (3,3)
INT2	Last integration point with value VAL. (Integer; Default = INT1) (3,4)
LAY1	First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)
LAY2	Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)
VAL	New state value for these elements. (Real; no Default; Required) (4,1)
STATE	Enter the character string STATE to start a list of state variables. (Character)
NS	Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)
ISi	State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.

Remarks:

1. All MCHSTAT ID's must be unique.
2. This entry maps to Marc's CHANGE STATE entry.
3. This entry must be entered if the MTCREEP (Marc's AUTO THERM CREEP) entry is used.
4. (i,j) refer to Marc's CHANGE STATE (data block, field).
5. MCHSTAT (and/or MINSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a Case Control LOAD command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a Case Control LOAD command to reference it.

## MCOHE

## Interface Cohesive Zone Modeling Element Material Properties

Specifies material cohesive properties for a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400 only.

Format:

1	2	3	4	5	6	7	8	9	10
MCOHE	MID	MODEL		TID					
	COHE	CRTOD	MAXOD	SNSR	EXP	VED	RRRD	SFC	
	SNER								

Example:

MCOHE	701	2		357					
	136.5	0.0	0.0	1.0	0.02	0.0		1.0	

Descriptor	Meaning
MID	Identification number of a MCOHE entry. (Integer > 0) (See Remark 1.)
MODEL	(Integer > 0; Default = 1) (See Remark 3.)
	1          Bilinear model
	2          Exponential model
	3          Linear-exponential model
	-1         User Defined Subroutine (See Remark 2.)
TID	Table identifier for a combination of TABLES1/TABLEST for cohesive energy vs temperature. (Integer ≥ 0; Default = 0)
COHE	Cohesive energy. (Real ≥ 0.0)
CRTOD	Critical opening distance. (Real ≥ 0.0)
MAXOD	Maximum opening displacement (bilinear model only). (Real ≥ 0.0)
SNSR	Shear Normal Stress Ratio. (Real > 0.0; Default = 1.0)
EXP	Exponential decay factor (linear-exponential model only). (Real > 0.0; Default = 1.0)
VED	Factor for viscous energy dissipation. (Real ≥ 0.0; Default = 0.0)
RRRD	Reference rate of relative displacement. Used only if VED ≠ 0.0 . A value of 0.0 implies that the reference rate will be automatically calculated. (Real ≥ 0.0; Default = 0.0)
SFC	Stiffening factor in compression. (Real ≥ 0.0; Default = 1.0)
SNER	Shear Normal Energy Ratio. (Real > 0.0; Default = 1.0)

**Remarks:**

1. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.
2. User subroutine UCOHES is used. User must also use the MATUDS bulk data entry.
3. For details on model see Nonlinear User's Guide (SOL 400).

**MDBCATP****Defines Parameters of Across Modules for Automatic Contact Generation**

Defines parameters for detecting contact pairs across modules for automatic contact generation (ACG). It is similar to BCAUTOP but is used for contact pair detection across modules, all the parameters for contact bodies in BCAUTOP are excluded.

Format:

1	2	3	4	5	6	7	8	9	10
MDBCATP			Param1	Value1	Param2	Value2	Param3	Value3	
			Param <i>i</i>	Value <i>i</i>					

Example:

MDBCATP			DISTANCE	0.3	DELSLD	YES			
---------	--	--	----------	-----	--------	-----	--	--	--

Describer	Meaning
Param( <i>i</i> )	Name of a parameter. Allowable names are given below (Character).
Value( <i>i</i> )	Value of the parameter. See below (Real, Integer or character). See Remark 3.

Remarks:

1. Only one entry of MDBCATP is allowed.
2. This entry does not have effect if BCONTACT=AUTO is not present in case control section.
3. The parameters in this entry may be divided into two types - primary parameters and secondary parameters. The primary parameters are the most important parameters for contact pair detection. The secondary parameters are helpful for user to provide the more detail and further requirement for contact model establishment and contact analysis with automatic contact generation. The primary and secondary parameters are listed in the tables as follows.

Table 9-28 Primary Parameters in MDBCATP

Describer	Meaning
CTYPE	Characters, one of TOUCH(default), PGLUE, SGLUE and GGLUE. If CTYPE is specified in both BCAUTOP entry and Case Control Command BCONTACT=AUTO, the specification in this entry will be used.

Describer	Meaning
DISTANCE	<p>Distance tolerance of contact pair. If the distance between any two points which are belonging to two different contact bodies is less than this value, these two contact bodies are recognized to be a contact pair. Default value of DISTANCE is 100 times of contact ERROR tolerance. ERROR may be defined in either MDBCATP or left blank. With default of ERROR, the code calculates ERROR as the smallest one of the following values:</p> <ol style="list-style-type: none"> <li>1           1/20 of the smallest nonzero element dimension (plates or solids) in the contact body.</li> <li>2           1/4 of the thinnest shell thickness in the contact body</li> </ol> <p>See BCPARA for ERROR definition in detail.</p>

Table 9-29          Secondary Parameters in MDBCATP

Describer	Meaning
BEAMCNT	<p>Characters, YES or NO (default). Determining if beam is included in contact definition.</p> <p>YES      BEAMs are included in the contact description.</p> <p>NO      No BEAM is included in the contact description.</p>
EDGECNT	<p>Characters, YES or NO (default). Determining if shell edge is included in contact description.</p> <p>YES      The free and hard shell edges are included in the contact definition.</p> <p>NO      No shell edges are included in the contact definition.</p>
FTYPE	<p>Characters, BLCOUL/BLSHEAR; or Integer, 6 / 7. BLCOUL or 6 is Bilinear Coulomb friction. BLSHEAR or 7 is Bilinear Shear friction. Default is No Friction.</p>
FRIC	<p>Friction coefficient. If the value is an integer, it represents the ID of a TABLEM1, TABLEM2 or TABL3D, i.e., a temperature-dependent or multi-dimensional table. (Real <math>\geq 0.0</math> or Integer <math>&gt; 0</math>; Default = 0.0)</p>
IGNTHK	<p>Character, YES or NO (default), Ignore thickness of shell for contact.</p> <p>YES      Ignore the thickness of the shell</p> <p>NO      Include the thickness of the shell</p>
INISTF	<p>Character, YES or NO (default). Set the option of initial stress free.</p>



Describer	Meaning
	YES Initial stress free contact.
	NO General contact without implementation of initial stress free.
SelfCont	Characters, YES or NO (default). Option of self-contact.

**Note:** The parameters defined in MDBCATP are applied only for BCONATCT=AUTO. Their default values are defined only when BCONTACT=AUTO is applied. All the parameters in BCPARA, BCONPRP and BCONPRG entries may be used in MDBCATP directly. For the detailed list of these parameters, please refer to BCPARA, BCONPRP and BCONPRG

**MDBCNCCT****Defines the Touching and Touched Contact Bodies in Different Modules**

Defines the touching and touched contact and used in SOLs 101, 103, 105, 107, 108, 109, 110, 111, 112 and 400 for general contact, glued, step glue or permanent glue between contact bodies residing in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDBCNCCT	ID	BCGPID	BCPPID	MODS	IDSCND	MODP	IDPRIM		
	“SECNDRY”	MODS1	IDSEC1	MODS2	IDSEC2	MODS3	IDSEC3		
		MODS4	IDSEC4	-etc.-					
	“PRIMARY”	MODP1	IDPRIM1	MODP2	IDPRIM2	MODP3	IDPRIM3		
		MODP4	IDPRIM4	-etc.-					

Example:

MDBCNCCT	57	306		101	2	201	1002		
----------	----	-----	--	-----	---	-----	------	--	--

MDBCNCCT	9		108						
	SECNDRY	101	30	201	26				
	PRIMARY	101	294	201	135	301	528		

Descriptor	Meaning
ID	Unique identification number referenced by a MDBCTB1 entry (Integer $\geq 0$ ). See Remark 5.
BCGPID	Parameter identification number of a BCONPRG entry (Integer $\geq 0$ or blank). See Remark 6.
BCPPID	Parameter identification number of a BCONPRP entry (Integer $\geq 0$ or blank). See Remark 6.
MODS	Identification number of a Module that defines IDSCND (Integer $\geq 0$ or blank)
IDSCND	Identification number of BCBODY1 entry defining the touching body (Integer $\geq 0$ or blank). See Remarks 7. and 8.
MODP	Identification number of a Module that defines IDPRIM (Integer $\geq 0$ or blank)
IDPRIM	Identification number of BCBODY1 entry defining the touched body (Integer $\geq 0$ or blank). See Remarks 9. and 10.
“SECNDRY”	Indicates the start of the list of the touching bodies. See Remark 8.
MODSi	Identification number of a Module that defines IDSECI (Integer $\geq 0$ or blank)
IDSECI	Identification number of BCBODY1 entry defining touching bodies (Integer $\geq 0$ or blank,). See Remarks 7. and 8.

Describer	Meaning
“PRIMARY”	Indicates the start of the list of bodies touched by touching bodies. See Remark 10.
MODPi	Identification number of a Module that defines IDPRIMi (Integer $\geq 0$ or blank)
IDPRIMi	Identification number of BCBODY1 entry defining touched bodies (Integer $\geq 0$ or blank).

#### Remarks:

- Both BCTABL1 and MDBCTB1 may only be specified in the main Bulk Data section.
- BCTABL1 BCID must be unique with respect to all other BCTABL1 and MDBCTB1.
- BCONTACT Case Control command may only select BCTABL1 or MDBCTB1 but not both. If BCTABL1 is selected, then all bodies matching IDi in any and all Modules will be activated.
- Only MDBCTB1 can select MDBCNCCT.
- MDBCNCCT can be selected by the Case Control command BCONTACT=ID to define surface contact if MDBCTB1 entry does not exist. See Remarks 2 and 3 of MDBCTB1 entry.
- If BCGPID or BCPPID field is blank, then default values are set for the parameters of touching bodies.
- A short input to define a single touching body exists if the user provides IDSCND. On the other hand, if the user leaves IDSCND blank, then “SECNDRY” descriptor is required and IDSECI must be specified.
- “SECNDRY” and IDSECI fields will be ignored if IDSCND exists. If IDSCND field is blank, then “SECNDRY” and IDSECI must be specified. In this case, each IDSECI will be processed separately.
- A short input to define a single touched body exists if the user provides IDPRIM. On the other hand, if the user leaves IDPRIM blank, then “PRIMARY” descriptor is required and IDPRIM1 must be specified.
- “PRIMARY” and IDPRIMi fields will be ignored if IDPRIM exists. If IDPRIM field is blank, then “PRIMARY” and IDPRIMi must be specified.
- The concept of Secondary and Primary relation is important to the node-to-segment contact but not relevant for segment-to-segment contact. In segment-to-segment contact, they are mainly used to define the contact pair(s).
- If all the BCONPRG that are referenced by a MDBCNCCT (which is referenced by MDBCTB1) have a value of IGLUE  $> 0$ , and this MDBCNCCT is referenced in the 1st Loadcase (SOL 100\*) then the connections are considered to be permanent and do not change (unless a BCPARA is used to deactivate the permanent glue).

**MDBC PAR****Defines Candidates Modules for Automatic Contact Detection Across Modules**

Defines modules as candidates for automatic contact detection across modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDBC PAR		SET3ID							
	MODID11	MODID21			MODID12	MODID22			
	MODID13	MODID23			MODID1i	MODID2i			

Example:

MDBC PAR		2							
	2	3			4	7			

Descriptor	Meaning
SET3ID	Integer $\geq -1$ , default=-1. SET3 ID refers to a SET3 with DES="MODULE".
MODID1i	The first module ID of the candidate of contact detection between two modules
MODID2i	The second module ID of the candidate of contact detection between two modules.

Remarks:

1. Only one entry of MDBC PAR is allowed in module 0, and it must be in module 0.
2. MDBC PAR is used when BCONTACT= AUTO is set in case control command, and modules are present.
3. When SET3ID is -1, all the modules will be used to detect contact pairs across them, when it is 0, and when there is no continuation, no contact pair will be detected across modules. When SET3ID > 0, the modules defined in the SET3 entry with SID=SET3ID and DES="MODULE" will be used to detect contact pairs across them. The distance tolerance of the pair detection is defined by DISTANCE parameter in MDBC ATP.
4. Continuations are optional.
5. The pairs of MODID1i and MODID2i are the pairs of two modules to be used to detect contact pairs between them. The distance tolerance of the pair detection is defined by DISTANCE parameter in MDBC ATP

## MDBCTB1 Defines a Contact Table for Bodies in Different Modules

MDBCTB1 specifies a list of contact pairs (for bodies residing in different Modules) through the MDBCNT option for SOL 101 and SOL 400. This contact table is activated in the BCONTACT Case Control command.

Format:

1	2	3	4	5	6	7	8	9	10
MDBCTB1	BCID	MOD1	ID1	MOD2	ID2	MOD3	ID3	MOD4	
	ID4	MOD5	ID5	-etc.-					

Examples:

MDBCTB1	2	101	198	201	62	301	75	401	
	501	8	601	159	701	31	801	44	

MDBCTB1	0	1001	23	1002	56				
---------	---	------	----	------	----	--	--	--	--

Describer	Meaning
BCID	Unique identification number referenced by a BCONTACT Case Control command. See Remark 3. (Integer $\geq 0$ ; Required)
MODi	Identification number of module that defines BCONECT IDi (Integer $\geq 0$ or blank,)
IDI	Identification number of BCONECT entry (Integer $> 0$ ). See Remark 4. and 5.

Remarks:

1. MDBCTB1 defines surface contact. If MDBCTB1 does not exist, the Case Control command BCONTACT=BCID may refer to the MDBCNT Bulk Data entry directly.
2. If neither MDBCTB1 nor MDBCNT is given, the default for contact analysis is assumed, every body detects the possibility of contact relative to all other bodies and itself if it is a deformable body. If MDBCTB1 or MDBCNT is given, the default for everybody is overwritten. The touching body does not contact itself unless requested. When the touched body is deformable, double-sided contact is applied by default. MDBCTB1 or MDBCNT is useful for deactivating or activating bodies to reduce computational effort and to change contact conditions between subcases.
3. For SOLs 101 and 400, the MDBCTB1 or MDBCNT with ID=0 will be used in loadcase 0 automatically that does not need a corresponding Case Control command BCONTACT=0. The loadcase 0 is purely elastic and it can also be used to (1) move rigid contact bodies so that they just touch flexible contact bodies, and/or (2) remove any prestressed condition by adjusting the coordinates of the active nodes, which are the Grid Points on the surface of all deformable

BCBODY1's. To place an entry in any physical loadcase (SUBCASE or STEP), the BCID must be selected by the Case Control command BCONTACT=BCID or MDBCNCT=BCID. When BCONTACT=ALLBODY, there is no BCID of MDBCTB1 or MDBCNCT specified; therefore, the default values of all entries of BCONPRG and BCONPRP are assumed. Case Control command BCONTACT=ALLBODY cannot be used for permanently glued contact.

4. The "THRU" option is not supported.
5. Intervening blank fields are not allowed.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

## MDBKSYS Defines Brake System for Modules

Defines data for brake system for brake squeal calculations involving one wheel or multiple wheels with one axis or multi-axes in different modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDBKSYS	ID	OMETH	IVEC	BSONLY	ISLIDER3				
	Disk1	MODID11	BODY1ID1	MODID21	BODYID21	MOTN1ID1	MOTN2ID1	BRKPID1	
	Disk2	MODID12	BODY1ID2	MODID22	BODYID22	MOTN1ID2	MOTN2ID2	BRKPID2	
	DiskI	MODID1i	BODY1IDi	MODID2i	BODYID2i	MOTN1IDi	MOTN2IDi	BRKPIDi	

Example:

MDBKSYS	900	0.5	0	YES					
	INNER1	1	2	2	3	1	2	4	
	OUTPER	2	1	3	4	2	4	2	

Describer	Meaning
ID	Identification ID of number of a corresponding BSQUEAL Case Control command
OMETH	Specifies the corresponding load factor (or time step) where the brake squeal analysis is to be performed. (Real; Default = 0.0)
BSONLY	Brake-Squeal-Only flag to control if or not to continue nonlinear iterations after brake squeal analysis is performed. YES Means to perform brake squeal analysis only and exit nonlinear iteration immediately. NO Means to continue nonlinear analysis. (Character; Default = YES).
IVEC	Flag specifying whether friction vector at center of primary contact body is in the same direction as that of the secondary nodes; see <a href="#">Figure 9-3</a> . (Integer; Default = 0). 0: In same direction 1: In tangential direction
ISLIDEB3	Flag indicating whether contact status is treated as sliding or general static contact for brake disk/pad pairs.

Describer	Meaning
	0: (default) sliding contact for brake disk/pad pairs.
	1: general static contact as usual.
DISKi	Name of the i <sup>th</sup> brake system.
MODID1i	Module id of the BCBODY1 of the first body which is defined in this brake system.
BODY1D1i	ID of BCBODY1 of the first body which is defined in this brake system.
MODID2i	Module id of the BCBODY1 of the second body which is defined in this brake system.
BODY1D2i	ID of BCBODY1 of the second body which is defined in this brake system.
MOTN1IDi	ID of MOTION which is defined for the first body.
MOTN2IDi	ID of MOTION which is defined for the second body.
BRKPID1i	ID of BRKSYS which is defined for the first body
BRKPID2i	ID of BRKSYS which is defined for the second body

Remarks:

1. This entry must be in module 0 and modules must be present.
2. The ID must be unique.
3. BRKSYS entries with the same ID in different modules can be selected also by corresponding BSQUEAL case control command.
4. When multiple BRKSYS entries and/or MDBKSYS are selected, the first lines of those entries can be not identical, it is treated as: Use the line defined in the MDBKSYS when it exists and selected, otherwise use default value of the line. When the first lines of all the entries are identical, use the line as is.



## MDBNDRY      Module to Module Boundary Point Definitions

Defines a list of grid points in a module for the automatic boundary search between a specified Module or between all other Modules. In other words, the automatic boundary search will only search for connections in this list.

Format:

1	2	3	4	5	6	7	8	9	10
MDBNDRY	MIDA	MIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	etc.						

Examples 1:

MDBNDRY	400	4	10	20	30	40			
---------	-----	---	----	----	----	----	--	--	--

Examples 2:

MDBNDRY	400	ALL	10	20	30	THRU	35		
---------	-----	-----	----	----	----	------	----	--	--

Describer	Meaning
MIDA	Module identification number. (Integer $\geq 0$ , Default=0)
MIDB	Reference Module identification number. (Integer $\geq 0$ or Character "ALL"; Default=ALL)
GIDAi	Identification number of a boundary grid point in the Module MIDA. (Integer $> 0$ or "THRU"; for "THRU" option, $GID1 < GID2$ .)

Remarks:

1. MDBNDRY may only be specified in the Main Bulk Data Section and cannot be specified in other BEGIN MODULE sections.
2. This entry is valid only if Modules exist.

**MDBOLT** Defines the Multi-Point Constraints for a Bolt Between Two Modules

Defines a rigid bolt between two Modules by a set of MPC constraints.

Format:

1	2	3	4	5	6	7	8	9	10
MDBOLT	ID	GRIDC	MODC	MODT	MODB				
	TOP	GT1	GT2	GT3	GT4	GT5	GT6		
		GT8	GT9	etc.					
	BOTTOM	GB1	GB2	GB3	GB4	GB5	GB6		
		GB8	GB9	etc.					

Example:

MDBOLT	100	1025	11	11	30				
	TOP	101	102	103	104	105			
	BOTTOM	1	2	3	4	5			

Descriptor	Meaning
ID	Element ID of the bolt. (Integer; Required; no Default)
GRIDC	Control GRID ID where forces or displacements are applied. (Integer; no Default; Required)
TOP	Enter the character string TOP to define the start of the entry that defines all of the grids at the “top” of the bolt intersection with the structure. (Integer; no Default)
MODC	Module containing the GRID entry for GRIDC. (Integer; Default=0, If MODC>0 then must be same as MODT or MODB)
MODT	Module containing the GRID entries for GTi. (Integer; Default=0)
MODB	Module containing the GRID entries for GBi. (Integer; Default=0)
GT1, GT2, etc.	Grid IDs of the grid points at the top of the bolt intersection. (Integer; no Default)
BOTTOM	Enter the character string BOTTOM to define the start of the entry that defines all of the grids at the “bottom” of the bolt intersection with the structure (do not enter the ID for GRIDC). (Integer; no Default)
GB1, GB2, etc.	Grid IDs of the grid points at the bottom of the bolt intersection. (Integer; no Default)

Remarks:

1. The MDBOLT entry must be defined in the main Bulk Data section only (Module 0).



2. See all remarks and figures under the BOLT entry description. All remarks and figures under BOLT apply to MDBOLT.

## MDBULK Module Type Definitions

Defines module search options and typing such as repeated or mirrored.

Format:

1	2	3	4	5	6	7	8	9	10
MDBULK	MODID	TYPE	RMODID	METHOD	TOL				

Example:

MDBULK	14			AUTO	1.0e-3				
--------	----	--	--	------	--------	--	--	--	--

Describer	Meaning
MODID	Module identification number. (Integer > 0 or Character = "ALL")
TYPE	Module type (Character; default = "PRIMARY")
PRIMARY	Module with at least one GRID entry in its BEGIN MODULE section.
REPEAT	Copied Module via reposition or mirroring of a primary Module. See Remarks 9 and 10.
MIRROR	Copied Module via mirroring of a primary Module. See Remarks 9 and 10.
MOVE	Copied or primary Module repositioning via MDMOVE entry. See Remarks 8 and 10.
RMODID	Identification number of the reference Module, used only if making a copy; i.e., RMODID>0 and RMODID?MODID and TYPE= "REPEAT", "MIRROR" or "MOVE". (Integer>0; Default=0). See Remarks 9 and 10.
METHOD	Method to be used when searching for boundary grid points. (Character = "AUTO" or "MANUAL"; Default= "MANUAL")
TOL	Location tolerance to be used when search for boundary grid points.  (Real; Default = 1.0e-5)

Remarks:

1. METHOD="MANUAL" requires MDCONNECT entries. MDBNDRY and MDEXCLD, that reference MODID, will produce fatal messages.
2. MDCONNECT, MDBNDRY and MDEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
3. For combined automatic and manual boundary search, the METHOD="AUTO" should be specified and connections should be specified on a MDCONNECT entry.
4. TOL is the default value that can be modified between two Modules by providing the required tolerance on the MDCONNECT entry.

5. MDBULK may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections
6. This entry is only valid if Modules exist.
7. With MDBULK,ALL the TYPE and RMODID fields must be blank and MDBULK,ALL means that the MDBULK options METHOD and TOL will be applied to all Modules which are not already defined on an MDBULK entry.
8. TYPE="MOVE" refers to the enhanced interface. If TYPE="MOVE" then an MDMOVE Bulk Data entry must be defined. The MDMOVE entry references MDTRAN, MDROT*i*, and MDMIR*i* entries.
  - a. If TYPE="MOVE" is specified for a given Module, then MDLOC and MDMPLN cannot be specified and vice-versa.
  - b. If MDMIR*i* is referenced by MDMOVE then it must be the first MVID and cannot be specified as the second or subsequent MVIDs.
  - c. If MODID defines a secondary Module (RMODID>0) then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
  - d. A primary Module (RMODID=0) may be re-positioned as indicated in Remark 10.
9. TYPE="REPEAT" and "MIRROR" refer to the classic interface. These types have many common features, but they also have some important differences. These are described below:
  - a. The following comments apply to both TYPE="REPEAT" and "MIRROR":
    - A Module (MODID) whose TYPE is "REPEAT" or "MIRROR" and RMODID is greater than 0 is referred to as a secondary Module. The RMODID specified in this case is regarded as the primary Module.
    - The primary Module may be re-positioned as indicated in Remark 10.
    - If MODID defines a secondary Module then it will automatically inherit all of the Primary Module's Bulk Data. The inheritance of a particular entry may be overridden by specifying new Bulk Data entries or ignored via the EXCLUDE Bulk Data entry in the secondary Module's Bulk Data section.
    - If both an MDLOC and MDMPLN entry are specified for a secondary Module, then a mirror image copy of the primary Module will be created by first using the plane defined by the MDMPLN entry and then repositioned at the location implied by the MDLOC entry.
  - b. The following comments apply only to TYPE="REPEAT":
    - The secondary Module in this case may reference an MDLOC entry, an MDMPLN entry or both or none.
    - If it references an MDLOC entry, then an identical copy of its primary Module will be positioned at the location implied by the MDLOC entry.
    - If it references an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.

- If it references neither an MDLOC entry nor an MDMPLN entry, then the secondary Module will merely be a duplicate of the primary Module positioned at the same location as the primary Module. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message
  - c. The following comments apply only to TYPE= "MIRROR":
    - The secondary Module in this case must reference an MDMPLN. (Otherwise, the program will terminate the execution with an appropriate user fatal message.)
    - If the secondary Module references only an MDMPLN entry, then a mirror image copy of the primary Module will be positioned using the plane defined by the MDMPLN entry.
10. A Module whose TYPE is "PRIMARY" (or TYPE="MOVE" and RMODID=0) may itself be repositioned by the use of MDLOC or MDMPLN entry or both (or MDTRAN, MDROTi or MDIRi or any combination).
- a. If it references an MDLOC (or MDTRAN, MDROTi) entry, then the primary Module will be positioned at the location implied by the MDLOC (or MDTRAN, MDROTi) entry.
  - b. If it references an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will be repositioned using the plane defined by the MDMPLN (or MDMIRi) entry.
  - c. If it references both an MDLOC (or MDTRAN, MDROTi) entry and an MDMPLN (or MDMIRi) entry, then a mirror image of the primary Module will first be created using the plane defined by the MDMPLN (or MDMIRi) entry and then repositioned at the location specified by the MDLOC (or MDTRAN, MDROTi) entry.

## MDCONCT      Module Boundary Point Connections

Explicitly defines grid and scalar connection procedures for a module to module operation.

Format:

1	2	3	4	5	6	7	8	9	10
MDCONCT	BID	TYPE	TOL	X	Y	Z		CID	
	MID1	GID1	MID2	GID2	MID3	GID3	MID4	GID4	
	MID5	GID5	etc.						

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MDCONCT	BID	TYPE	TOL	GRID			MODID		
	MID1	GID1	MID2	GID2	MID3	GID3	MID4	GID4	
	MID5	GID5	etc.						

Example 1:

MDCONCT	123		0.4	0.5	74.12	100.3			
	10	423	20	123					

Example 2:

MDCONCT	123	RRBE2	0.4	777					
	10		20						

Describer	Meaning
BID	Unique Boundary identification number. (Integer > 0)
TYPE	Connection Option. (Character: "MERGE", "RIGID", "RRBE2", "MRBE2" or Blank, Default="MRBE2").
TOL	Location tolerance to be used when searching or checking for boundary grid points. (Real; Default=1.0e-5)
X,Y,Z	Connection (Search) location. (Real, Default=0.0)
GRID	Grid Identification for Connection (Search). (Integer > 0) See Remark 13.
MODID	Identification number of Module containing GRID. (Integer ≥ 0, Default=0)
CID	Coordinate System in Module 0 to be applied to XYZ location. (Integer ≥ 0, Default=0)

Describer	Meaning
MIDi	Module Identification number for grid/scalar reference. (Integer $\geq 0$ )
GIDi	Identification number of a grid or scalar point in the reference module MIDi, which will be verified to be in TOL of the Location. (Integer $\geq 0$ )

#### Remarks:

1. MDCONCT may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.
2. TOL can be used to override the default value specified on the MDBULK entries.
3. The continuation entry is optional when used in automatic searching.
4. The GIDi selections must be all of the same class, grids or scalar points.
5. All six degrees-of-freedom of grids will be defined as boundary degrees-of-freedom.
6. This entry is only valid if Modules exist.
7. When GIDi is blank or zero, then all grids of the module will be tested against the location and tolerance.
8. MIDi values can be specified more than once.
9. For TYPE="MERGE", the boundary grids will be moved to the location specified by MODID and connected by an RBE2 element.
10. For TYPE="MRBE2", the boundary grids will be moved to the location specified by MODID and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
11. For TYPE="RIGID", the boundary grids will not be moved and will be connected via RBE2 elements.
12. For TYPE="RRBE2", the boundary grids will not be moved and only those grids that are specified as the independent grids on RBE2 elements will be used to connect the Modules. If any of the GIDi grids are not specified as independent grids on RBE2 elements or there are no RBE2 elements then the run will terminate with User Fatal Message 6717.
13. When GRID is used, then this identification/location will be used for the independent point of the RBE2 element.
14. When scalar points are specified for GIDi, then GRID, X, Y, Z, MODID and CID should not be specified.
15. If any GIDi are not connected due to exceeding TOL then User Fatal Message 6783 will be issued.



**MDDMIG****Direct Matrix Input at Points Defined in Two or More Modules**

Defines direct input matrices related to grid, extra, and/or scalar points defined in two or more Modules. The matrix is defined by a single header entry and one or more column entries. A column entry is required for each column with nonzero elements.

**Header Entry Format:**

1	2	3	4	5	6	7	8	9	10
MDDMIG	NAME	"0"	IFO	TIN	TOUT	POLAR		NCOL	

**Column Entry Format:**

MDDMIG	NAME	MODJ	GJ	CJ					
		MOD1	G1	C1	A1	B1			
		MOD2	G2	C2	A2	B2	-etc.-		

**Example:**

MDDMIG	STIF	0	1	3	4				
MDDMIG	STIF	11	27	1					
		20	2	3	3+5	3+3			
		20	2	4	2.5+10	0.			
		45	50		1.0	0.			

Describer	Meaning
NAME	Name of the matrix. See Remark 1 under DMIG. (One to eight alphanumeric characters, the first of which is alphabetic.)
IFO	Form of matrix input. IFO = 6 must be specified for matrices selected by the K2GG, M2GG, and B2GG Case Control commands. (Integer) <ul style="list-style-type: none"> <li>1 Square</li> <li>2 or 9 Rectangular</li> <li>6 Symmetric</li> </ul>
TIN	Type of matrix being input: (Integer) <ul style="list-style-type: none"> <li>1 Real, single precision (One field is used per element.)</li> <li>2 Real, double precision (One field is used per element.)</li> <li>3 Complex, single precision (Two fields are used per element.)</li> <li>4 Complex, double precision (Two fields are used per element.)</li> </ul>
TOUT	Type of matrix that will be created: (Integer)

Describer	Meaning
	0 Set by precision system cell (Default)
	1 Real, single precision
	2 Real, double precision
	3 Complex, single precision
	4 Complex, double precision
POLAR	Input format of Ai, Bi. (Integer=blank or 0 indicates real, imaginary format; Integer > 0 indicates amplitude, phase format.)
NCOL	Number of columns in a rectangular matrix. Used only for IFO = 9. See Remarks 5 and 6 under DMIG. (Integer > 0)
MODJ	Module IDs that contain the GRID entry for GJ. (Integer ≥ 0).
GJ	Grid, scalar or extra point identification number in Module MODJ for column index. (Integer > 0)
CJ	Component number for grid point GJ. (0 < Integer < 6; blank or zero if GJ is a scalar or extra point.)
MODi	Module IDs that contain the GRID entry for Gi. (Integer ≥ 0).
Gi	Grid, scalar, or extra point identification number in Module MODi for row index. (Integer > 0)
Ci	Component number for Gi for a grid point. (0 < CJ < 6; blank or zero if Gi is a scalar or extra point.)
Ai, Bi	Real and imaginary (or amplitude and phase) parts of a matrix element. If the matrix is real (TIN = 1 or 2), then Bi must be blank. (Real)

Remarks:

1. The MDDMIG entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the DMIG entry description. All remarks and figures under DMIG apply to MDDMIG.
3. When IFO=2 or 9, MODJ indicates the column number and not a module ID. GJ and CJ are ignored.

**MDELAM****Delamination materials in SOL 600**

Defines materials for which delamination may occur in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MDELAM		MID1	MID2	ITYPE	STRN	EXPN	STRT	EXPT	
	ITSTRN	ITEXPN	ITSTRT	ITEXPT					

Example:

MDELAM		4	5	2	35000.	1.	20000.	1.	
--------	--	---	---	---	--------	----	--------	----	--

Describer	Meaning
MID1	First material ID. (Integer; no Default; required field)
MID2	Second material ID. If MID2=MID1 delamination occurs within the material. If MID2 is not the same as MID1, delamination may occur at the interfaces between the materials. (Integer; Default = MID1)
ITYPE	Delamination type (Integer; Default = 1) 1 Determine if delamination occurs but do not actually allow delamination 2 Split the mesh when delamination occurs
STRN	Allowable normal stress (Real; no Default; required value)
EXPN	Exponent for normal stress (Real; Default = 1.0)
STRT	Allowable tangential stress (Real; no Default; required value)
EXPT	Exponent for tangential stress (Real; Default = 1.0)
ITSTRN	ID of a TABL3D entry providing the variation of STRN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITEXPN	ID of a TABL3D entry providing the variation of EXPN vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITSTRT	ID of a TABL3D entry providing the variation of STRT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)
ITEXPT	ID of a TABL3D entry providing the variation of EXPT vs up to 4 variables. A value of zero means no table is required. (Integer; Default = 0)

Remarks:

- The delamination criteria is based upon  $(S_n/STRN)**EXPN+(S_t/STRT)**EXPT > 1.0$   
Where  $S_n$  and  $S_t$  are the normal and tangential stresses
- If no tables are required, the continuation entry may be omitted.

## MDEXCLD Module to Module Excluded Boundary Point Definitions

Defines a list of grid points in a Module that will be excluded during the attachment to another Module.

Format:

1	2	3	4	5	6	7	8	9	10
MDEXCLD	MIDA	MIDB	GIDA1	GIDA2	GIDA3	GIDA4	GIDA5	GIDA6	
	GIDA7	GIDA8	etc.						

Example 1:

MDEXCLD	400	4	10	20	30	40			
---------	-----	---	----	----	----	----	--	--	--

Example 2:

MDEXCLD	400	ALL	10	20	30	THRU	35		
---------	-----	-----	----	----	----	------	----	--	--

Describer	Meaning
MIDA	Module identification number. (Integer $\geq 0$ , Default=0)
MIDB	Reference Module identification number. (Integer $\geq 0$ or Character "ALL"; Default=ALL)
GIDAi	Identification number of a boundary grid point in the module MIDA to be excluded from connection to module MIDB. (Integer $> 0$ or "THRU"; for "THRU" option, $GID1 < GID2$ .)

Remarks:

1. This entry is only valid if Modules exist.
2. MDEXCLD may only be specified in the Main Bulk Data Section and cannot be placed in other BEGIN MODULE sections.

## MDFAST

## A Shell Patch Fastener Connection Between Two Modules

Defines a fastener with material orientation connecting two surface patches defined in two Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDFAST	EID	PID	TYPE	IDA	IDB	GS	GA	GB	
	XS	YS	ZS			MODA	MODB		

Example using PROP:

MDFAST	3	20	PROP	21	24	206			
							10	20	

Example using ELEM:

MDFAST	7	70	ELEM	27	74	707			
							11	12	

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PFAST entry. (Integer > 0; Default = EID)
TYPE	Specifies the surface patch definition: (Character)  If TYPE = 'PROP', the surface patch connectivity between patch A and patch B is defined with two PSHELL (or PCOMP) properties with property ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28.  If TYPE = 'ELEM', the surface patch connectivity between patch A and patch B is defined with two shell element ids given by IDA and IDB. Under CFAST see Remark 1. and Figure 8-28.
IDA,IDB	Property id (for PROP option) or Element id (for ELEM option) defining patches A and B in Modules MODA and MODB, respectively. IDA≠IDB (Integer > 0)
GS	Grid point defining the location of the fastener. Its GRID entry must be defined in the main Bulk Data section (Module 0). See also Remark 2. under CFAST. (Integer > 0 or blank)
GA,GB	Grid ids of piercing points on patches A and B. Their GRID entries must be defined in the main Bulk Data section (Module 0). See also Remark 2. under CFAST. (Integer > 0 or blank)

Describer	Meaning
XS,YS,ZS	Location of the fastener in basic. Required if neither GS nor GA is defined. See also Remark <a href="#">2</a> .under CFAST. (Real or blank)
MODA,M ODB	Module IDs that contain the surface patches defined by property or element entries IDA and IDB (Integer $\geq 0$ ).

**Remarks:**

1. The MDFAST entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CFAST entry description. All remarks and figures under CFAST apply to MDFAST.

**MDFSET****Set Definition for Fluid Elements or Grid Points in Modules**

Defines a set of fluid grid points or element identification numbers in a Module to be referenced by the ACMODL Bulk Data entry.

Format:

1	2	3	4	5	6	7	8	9
MDFSET	MODID	ID1	ID2	ID3	ID4	ID5	ID6	ID7
	ID8	-etc.-						

Example 1:

MDFSET	22	31	62	93	124	16	17	18
	19							

Example 2:

MDFSET	22	29	32	THRU	50	61	THRU	70
	17	57						

Describer	Meaning
MODID	Module identification number (Integer $\geq 0$ )
IDI	List of fluid grid point or element identification numbers. (Integer $> 0$ or "THRU"; for the "THRU" option: ID1 $<$ ID2).

Remarks:

1. The MDFSET entry can only be used in the presence of Modules and can only be specified in the main Bulk Data section or Module 0.
2. The MDFSET entry is used when FSET is set to -1 on the ACMODL entry.
3. Multiple MDFSET entries may reference more than one Module.

## MDLABEL

## Module Output Label

Defines a label or name to be printed in the Module's results output page headings.

Format:

1	2	3	4	5	6	7	8	9	10	
MDLABEL	MID	LABEL								

Example:

MDLABEL	400	LEFT REAR FENDER, MODEL XYZ2000								
---------	-----	---------------------------------	--	--	--	--	--	--	--	--

Describer	Meaning
MID	Module identification number. (Integer ≥ 0, Default=0)
LABEL	Label associated with Module MID for results output page headings. (Character)

Remarks:

1. MDLABEL can only be specified in the main Bulk Data Section and is ignored in-between the BEGIN MODULE=n and ENDMODULE commands.
2. Only one MDLABEL per Module may be specified.
3. The label will appear in all Module's results output page headings. However, in some headings the label may be truncated.
4. This entry is valid only if Modules exist.



**MDLOC****Module Reposition by Translation and/or Rotation**

Defines reposition of a Module by listing three non-collinear points in the Module and three corresponding points in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDLOC	MODID	PA1	PA2	PA3	PB1	PB2	PB3		

Example:

MDLOC	110	10	100	111	1010	112	30		
-------	-----	----	-----	-----	------	-----	----	--	--

Describer	Meaning
MODID	Module identification number of primary, repeated, or mirrored module (Integer>0).
PAi	GRID or POINT identification numbers of three non-collinear points in (Integer>0): <ul style="list-style-type: none"> <li>■ "RMODID's Bulk Data section. RMODID is referenced on MDBULK if this MODID is a secondary (copied) Module</li> <li>■ "Module MODID's Bulk Data section if this is a primary Module.</li> </ul>
PBi	GRID or POINT identification numbers of three non-collinear points in Module 0 (main Bulk Data section). Must have the same relative locations as PAi. (Integer>0).

Remarks:

1. MDLOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. The Module will be rotated and translated for alignment of the PAi and PBi locations.
3. The PAi and PBi can be either GRIDs or POINTs.
4. PA1, PA2, and PA3 must be contained in the BEGIN Module=MODID Bulk Data section.
5. PB1, PB2, and PB3 must be specified in the main Bulk Data Section or Module 0. If they belong to a Module that is also relocated, then the original (unmoved) positions of PB1, PB2, and PB3 are used.
6. PB1, PB2, and PB3 must have the same relative locations as PA1, PA2, and PA3.
7. Three GRIDs or POINTs are required even if the Module connects to only one or two boundary grids.
8. Coordinate systems, global displacement directions, and element coordinate systems for the Module will rotated and translated.

9. The global coordinate directions of the grid points in the Module will be transformed internally to the global coordinate directions of the grid points in the Module 0. For displacement data recovery, the output will be in the original global coordinate system.
10. The translation and rotation of the Module to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:
  - The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
  - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
  - The z-axis is defined by the cross product of the x-axis into the y-axis.
  - The rotation and translation transformation aligns the local system defined by the Module grids with the local system defined by the main Bulk Data Section grids
11. This entry will only work if Modules (BEGIN MODULE) exist.

## MDLPRM

## Model Parameters

Specifies parameters which affect the solution of the structural model.

Format:

1	2	3	4	5	6	7	8	9	10
MDLPRM	PARAM1	VAL1	PARAM2	VAL2	PARAM3	VAL3	-etc.-		

Example:

MDLPRM	QR6ROT	2	QRSHEAR	1					
--------	--------	---	---------	---	--	--	--	--	--

Describer	Meaning
PARAMi	Name of the parameter. Allowable names are given in <a href="#">Table 20</a> . (Character)
VALi	Value of the parameter. (Real or Integer; see <a href="#">Table 20</a> )

Remark:

- Multiple entries of MDLPRM are allowed in the Bulk Data Section. However, multiple entries of a particular parameter PARAMi are illegal.
- This entry is not supported in SOL 600.
- If Modules are present then this entry may only be specified in the main Bulk Data section.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values
BOLTCNEL	Flag to control automatic mapping of BOLT1 elements to Advanced Nonlinear Elements using internally generated nonlinear property extensions.
	0                      Map BOLT1 elements as well as other elements that share same element properties with BOLT1 elements (Default).
	1                      Map all elements connected to BOLT1 elements as well as other elements that share same element properties.
BRTOBM	Flag to determine whether to convert a CBAR element to a CBEAM element for the nonlinear analysis.
	0                      Do not convert CBAR to CBEAM. (Default)
	1                      Convert CBAR to CBEAM.
	-1                     Same as 1, but print the converted Bulk Data entries on f06 file.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values
BUSHRT	Flag to control large rotation effects for CBUSH or CFAST elements in nonlinear analysis (SOL400). For CBUSH, this flag is overridden by the LRGR field on the element PBUSHT entry.
	0 Large rotation effects are included. Element coordinate system is rotated with the rotation of grid A. (Default)
	1 No large rotation effects. Small rotation is used.
	2 Large rotation effects are included. A mid-increment method is used to rotate the element system. This is recommended for large rotation analysis. For the CBUSH element, CID must equal zero.
COMPN1	Control if stresses and strains of advanced composite elements use MSC Nastran conventional layer format
	NO (default) Don't use
	YES Use conventional layer format
DBCTOLE	Tolerance for DBC date conversion, real(default=1.e-4) >= 0.0
DCFLTEXP	Determines the exponent of the tolerance value used to filter out small entries from the coefficient matrix before a matrix decomposition. In each column, matrix entries which are DCFLTEXP orders of magnitude smaller than the diagonal entry of the column are filtered out. If DCFLTEXP=0, then the coefficient matrix is not altered. (Integer; Default = 0)
DELELAS	Randomly delete c*100% of CELASi elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: c = 0.02). If entry is not present, all CELASi elements are retained. Default of .02 is used only if entry is written as: MDLPRM, DELELAS.
DELFAST	Randomly delete c*100% of CFAST elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: c = 0.02). If entry is not present, all CFAST elements are retained. Default of .02 is used only if entry is written as: MDLPRM, DELFAST  The value on this entry will override the PARAM, CFRANDEL, value entry. PARAM, CFDIAGP, YES will give the ID of the CFAST elements deleted.
DELMASS	Randomly delete c*100% of CMASSi, CONM1, and CONM2 elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: c = 0.02). If entry is not present, all CMASSi elements are retained. Default of .02 is used only if entry is written as: MDLPRM, DELMASS.
DELSEAM	Randomly delete c*100% of CSEAM elements in a job. ( $0.0 \leq c \leq 1.0$ ; Default: c = 0.02). If entry is not present, all CSEAM elements are retained. Default of .02 is used only if entry is written as: MDLPRM, DELSEAM.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values				
DELWELD	<p>Randomly delete <math>c \cdot 100\%</math> of CWELD elements in a job. (<math>0.0 \leq c \leq 1.0</math>; Default: <math>c = 0.02</math>). If entry is not present, all CWELD elements are retained. Default of .02 is used only if entry is written as: MDLPRM, DELWELD.</p> <p>The value on this entry will override the PARAM, CWRANDEL, value entry. PARAM, CWDIAGP, YES will give the ID of the CWELD elements deleted.</p>				
GEV1417	<p>See Remark 9 of PBUSH</p> <table border="1" data-bbox="429 491 1326 571"> <tr> <td data-bbox="429 491 591 534">0</td> <td data-bbox="591 491 1326 534">Use GE default rules on PBUSH. (Default)</td> </tr> <tr> <td data-bbox="429 534 591 571">1</td> <td data-bbox="591 534 1326 571">Use Nastran Version 2014-2017 incompatible rule for GE defaults.</td> </tr> </table>	0	Use GE default rules on PBUSH. (Default)	1	Use Nastran Version 2014-2017 incompatible rule for GE defaults.
0	Use GE default rules on PBUSH. (Default)				
1	Use Nastran Version 2014-2017 incompatible rule for GE defaults.				

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values												
GMOVR3	<p>In SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected <math>G_{i,j}</math> nodes, a small perturbation is applied to the <math>G_{i,j}</math> nodes.</p> <p>The RBE3 element is a least square fit element, where a matrix, <math>A=S^T W S</math> is formed. <math>S</math> is a skew-symmetric matrix of coordinate differences and <math>W</math> is a weighting matrix. For least squares, the <math>A</math> matrix must be inverted and for connections such as a cascading system (shown in figure), in SOL400 nonlinear analysis (RIGID=LAGRANGE required) where rotations are considered, the Collector RBE3 “Master REFGRID” grid should not be coincident with the coincident RBE3 weighted <math>G_{i,j}</math> nodes as a “small” length is necessary to provide rotation information. To correct this condition, a small perturbation is provided to ALL connected <math>G_{i,j}</math> nodes of <math>(x^k + \Delta v, y^k + \Delta v, z^k + \Delta v)</math>.</p> <p>The <math>G_{i,j}</math> nodes are the preferred nodes to move, because, often they are just the spatial locations where FEM models from different groups join a common structural component and the REFGRID is often an actual structural mesh node.</p> <p>This entry only affects RBE3 elements meeting the following criterion:</p> <ol style="list-style-type: none"> <li>Let <math>x^q, y^q, z^q</math> be the coordinates of the REFGRID and <math>x^k, y^k, z^k</math> the coordinates of one of the connected <math>G_{i,j}</math> nodes.</li> <li>If <math> x^k - x^q ,  y^k - y^q ,</math> and <math> z^k - z^q </math> are all less than or equal to <math>GMOVR3T</math> (default=0.001) for ALL connected <math>G_{i,j}</math> nodes, the grids are considered coincident.</li> </ol> <p>(Integer; Default = -1)</p> <table border="1" data-bbox="425 1064 1322 1604"> <tbody> <tr> <td data-bbox="425 1064 582 1104">-1</td> <td data-bbox="582 1064 1322 1104">Do not check coincidence</td> </tr> <tr> <td data-bbox="425 1104 582 1182">0</td> <td data-bbox="582 1104 1322 1182">Check coincidence but do not perturb. The program issues a fatal message and aborts run while coincidence happens.</td> </tr> <tr> <td data-bbox="425 1182 582 1222">1</td> <td data-bbox="582 1182 1322 1222">Perturbate all coincident coordinates of the connected <math>G_{i,j}</math> nodes.</td> </tr> <tr> <td data-bbox="425 1222 582 1262">2</td> <td data-bbox="582 1222 1322 1262">Perturbate the coincident REFGRID grid.</td> </tr> <tr> <td data-bbox="425 1262 582 1437">3</td> <td data-bbox="582 1262 1322 1437">GMOVR3 Perturbation is overwritten for the <math>G_{i,j}</math> nodes of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected <math>G_{i,j}</math> nodes. For elements not overwritten case, set <math>GMOVR3 = 1</math>. The elements to be overwritten are defined on a <math>GMOVR3</math> Bulk Data entry.</td> </tr> <tr> <td data-bbox="425 1437 582 1604">4</td> <td data-bbox="582 1437 1322 1604">GMOVR3 Perturbation is overwritten for the REFGRID grid of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected <math>G_{i,j}</math> nodes. For elements not overwritten case, set <math>GMOVR3 = 2</math>. The elements to be overwritten are defined on a <math>GMOVR3</math> Bulk Data entry.</td> </tr> </tbody> </table>	-1	Do not check coincidence	0	Check coincidence but do not perturb. The program issues a fatal message and aborts run while coincidence happens.	1	Perturbate all coincident coordinates of the connected $G_{i,j}$ nodes.	2	Perturbate the coincident REFGRID grid.	3	GMOVR3 Perturbation is overwritten for the $G_{i,j}$ nodes of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected $G_{i,j}$ nodes. For elements not overwritten case, set $GMOVR3 = 1$ . The elements to be overwritten are defined on a $GMOVR3$ Bulk Data entry.	4	GMOVR3 Perturbation is overwritten for the REFGRID grid of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected $G_{i,j}$ nodes. For elements not overwritten case, set $GMOVR3 = 2$ . The elements to be overwritten are defined on a $GMOVR3$ Bulk Data entry.
-1	Do not check coincidence												
0	Check coincidence but do not perturb. The program issues a fatal message and aborts run while coincidence happens.												
1	Perturbate all coincident coordinates of the connected $G_{i,j}$ nodes.												
2	Perturbate the coincident REFGRID grid.												
3	GMOVR3 Perturbation is overwritten for the $G_{i,j}$ nodes of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected $G_{i,j}$ nodes. For elements not overwritten case, set $GMOVR3 = 1$ . The elements to be overwritten are defined on a $GMOVR3$ Bulk Data entry.												
4	GMOVR3 Perturbation is overwritten for the REFGRID grid of specific RBE3 elements in SOL400, RIGID=LAGRANGE, for any RBE3 element with REFGRID grid coincident with all its connected $G_{i,j}$ nodes. For elements not overwritten case, set $GMOVR3 = 2$ . The elements to be overwritten are defined on a $GMOVR3$ Bulk Data entry.												

Table 20 PARAMi Names and Descriptions

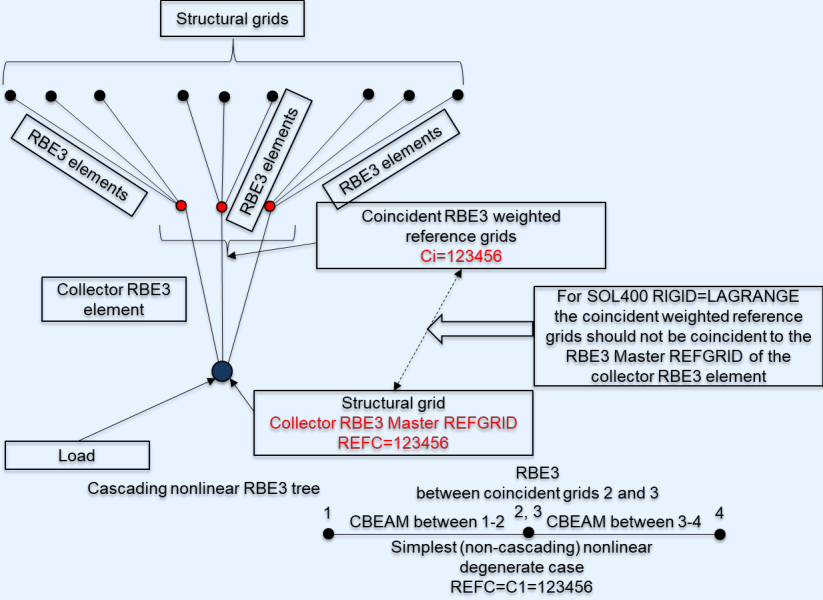
Name	Description, Type, and Default Values			
GMOVRD	<p>The Grids defined by above GMOVR3 entry will be given a <math>\Delta v = \text{GMOVRD}</math> value. The user should avoid very small perturbation values as the point is to provide a finite rotation length. For large structural models, <math>0.01 \leq \text{GMOVRD} \leq 0.1</math> seem to have relatively small effect on displacement values and reasonable convergence times. (Real; Default=0.01)</p> 			
GMOVR3T	Coincident grid zero tolerance. (Real $\geq 0.0$ ; Default=0.001)			
GNLSTN	Strain formulation flag for QUADR/TRIAR elements used in geometric nonlinear analysis. (Parameter LGDISP=1)			
	<table border="1"> <tr> <td data-bbox="432 1185 589 1220">0</td> <td data-bbox="589 1185 1318 1220">Small strain. (Default)</td> </tr> <tr> <td data-bbox="432 1220 589 1255">1</td> <td data-bbox="589 1220 1318 1255">Green strain.</td> </tr> </table>	0	Small strain. (Default)	1
0	Small strain. (Default)			
1	Green strain.			
HDF5	Parameter to create NH5RDB database			
	-1 Do not create NH5RDB(default)			
	0 Create NH5RDB without compression			
	1 Create NH5RDB with compression			
	2 Create uncompressed NH5RDB database without input data.			
3 Create compressed NH5RDB database without input data.				

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
H5GM34	Write GEOM3 and GEOM4 data in NH5RDB	
	1	Write GEOM3 and GEOM4 data in NH5RDB.
	0	Do not write GEOM3 and GEOM4 data in NH5RDB.
	-1	Use OP2GM34 setting for GEOM3 and GEOM4 output (Default).
H5INFO	Write job run information in NH5RDB or not.	
	1	Yes (Default)
	0	No
H5MDL	Write model input data in separate file	
	2	Write model input data only
	1	Write model input data into a separate file
	0	Do not write model input data into a separate file (Default)
	-1	Use OP2GM34 setting for GEOM3 and GEOM4 output
H5MTX	Write matrix data in separate file	
	1	Write matrix data into a separate file
	0	Do not write matrix data into a separate file (Default)
H5NORDOF	Parameter to suppress output rotational components to NH5RDB database	
	0	Output both translational and rotational components to NH5RDB database (Default).
	1	Do not output rotational components to NH5DB database.
		Only used when HDF5 is 0 or 1.
H5SGENL	Set compression factor for NLOUT output datasets	
	0 to 10	Scale factor with scaleoffset lossy compression (Default=3)
	-1	Use lossless compression
H5SSTRN	Set compression factor for NLOUT strain type output datasets	
	0 to 10	Scale factor with scaleoffset lossy compression (Default=6)
	-1	Use lossless compression
H5XHH	Write BHH, MHH and KHH matrices in NH5RDB	
	0	Do not write BHH, MHH and KHH in NH5RDB (Default)
	1	Write BHH, MHH and KHH in NH5RDB



Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
IGNSHBDN	Check adjacent shell elements have opposite direction of normal vector, if found	
	0(default)	Issue a user fatal message
	1	Issue a user warning message
INTOUT	Flag to control FORCE/STRESS/STRAIN OUTPUT location for QUADR/TRIAR elements	
	0	Corner output. (Default)
	1	Integration point output.
LMT2MPC	Parameter to enable the use of elimination method to process Lagrange Multipliers.	
	0	Do not force the use of elimination method. (Default)
	1	Use elimination method to process Lagrange Multipliers using unsymmetric method.
	2	Use elimination method to process Lagrange Multipliers using symmetric method (may be faster).
LRGDHFLT	The threshold ratio to set small terms in displacement gradient matrix for Lagrange rigid elements to zero to overcome numerical difficulties.	
	Suppose HMAX is the maximum absolute element of the matrix, H is another element, if $ABS(H) \leq LRGDHFLT * HMAX$ , set H to zero. (Real, LRGDHFLT $\geq$ 0.0, Default: LRGDHFLT = 1.E-10)	
MLTSPLIN	Parameter to specify whether an aerodynamic grid can be splined more than once.	
	0	References on separate splines to the same aero grid are not allowed. (Default)
	1	Aero grids can be referenced on multiple spline entries.
MPCF129	Request the calculation of MPC forces in SOL129. The forces include only the contributions from linear elements.	
	0	Don not compute MPC Forces (default)
	1	Compute MPC Forces
MXMNMEM	Working area memory size of MAXMIN command in machine words. (Integer, MXMNMEM > 0, Default: MXMNMEM = 9710).	

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values
NLDIFF	Flag to determine whether the differential stiffness matrix and follower force stiffness are to be computed for nonlinear elements with geometric nonlinear analysis (parameter LGDISP=1) in SOL 400. Options 3, 4, and 5 are available only for elements with PSHNL1, PSHNL2, PSLN1, or PSHEARN Bulk Data entry.
	0 Compute. (Default except for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis)
	1 Do not compute
	2 The differential stiffness matrix and follower force stiffness will not be computed if the tangential stiffness matrix is negative definite (Default for CTRLDEF=SEVERELY of NLSTEP and not 3D contact analysis).
	3 Include only the deviatoric part of the differential (or initial stress or geometric) stiffness
	4 Include only the tensile part of the differential (or initial stress or geometric) stiffness
	5 Include the stress at the beginning of the increment for the differential (or initial stress or geometric) stiffness
NLRSTRM	Flag to allocate extra necessary memory in a cold start run with Advanced Nonlinear Elements to allow subsequent NLRESTART to be used in a linear perturbation analysis when the cold start run itself is not a linear perturbation analysis.
	0 Do not allocate extra memory in cold start run (Default)
	1 Allocate extra memory in cold start run to allow NLRESTART to be used with Advanced Nonlinear Elements in a linear perturbation analysis.
NONUIV	Parameter to select the numeric compute kernel and pivoting methods in MSCLDL and MSCLU sparse direct solvers.
	0 Use the native Bunch-Kauffman threshold pivoting in MSCLDL, and the native threshold partial pivoting in MSCLU (Default).
	1 Use no numeric pivoting in MSCLDL and MSCLU. BLAS3 TRSMs are called to compute the pivot column update to improve performance. Ill-conditioned models may die of “singular matrix” during sparse factorization.
	2 Not documented.
	3 LAPACK SYTRFs with Bunch-Kaufman pivoting and GETRFs with partial pivoting are called to perform factorizations, and BLAS3 TRSMs are called to compute pivot column update to improve performance.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values
NSGRDS4	Number of structural grids to be used in dividing a SPLINE4 using the RIS spline method. The spline will be divided into NGRIDS/NSGRDS4 regions, where NGRIDS is the number of grids listed on the associated SET1 entry. (Integer > 0, default=0. If NGRIDS < NSGRDS4, or NSGRDS4 is not specified, no divisions will occur.)
OFFDEF	Element offset definition. A flag to determine how shell elements and bar and beam elements behave when the user supplies ZOFF values on the shell connection entries (CQUAD4, CQUADR, CTRIA3, CTRIAR, CQUAD8, and CTRIA6) and WiA and WiB on CBAR, CBEAM, and CBEAM3 connection entries. (Character)
ELMOFF	Standard Nastran offset method. The ZOFF rotate with the shell element. The WiA and WiB offsets for beams are fixed. MD Nastran R3 and earlier. (Character, Default)
LROFF	<b>Large rotation offsets.</b> The shell normal directions are used to define the offset direction at each shell grid for CQUAD4, CTRIA3, CQUADR, CTRIA4, CQUAD8 and CQUAD8 elements. For beams, WiA and Wib defines the offset direction. This method allows for thermal load effects on ZOFF for shells and WiA and WiB for beams. Thermal load effect for offset is computed based on the grid point or element temperature, and thermal coefficient of the element (see NOTHRM). The mass moment of inertia is computed for the offset due to the grid point location change introduced by offset. Differential stiffness is computed for the offset using the same method as that of the Lagrange formulation of the RBAR. Not supported in SOL106, SOL129 and SOL 200. (Character)
NODIF	LROFF is used but the differential stiffness effect is turned off. (Character)
NOTHRM	LROFF is used but the thermal load effects are turned off. The thermal load has two effects: 1) the location of thermal load changes due to offset and 2) the length of offset changes due to thermal load. Effect (1) is computed for all solution sequences and Effect (2) is computed for SOL 400 only. Both effects are turned off by NOTHRM. (Character)
NODT	LROFF is used but the differential stiffness and thermal load effect are turned off. (Character)
ELMZ	LROFF is used but the element z-direction is used for the offset direction. IF PARAM, SNORM, 0.0 or the computed value for SNORM is greater then the PARAM,SNORM,value, then the LROFF option will revert to this method for CQUAD4, CTRIA3, CQUADR, CTRIAR, CQUAD8 and CTRIA6. This option is not applicable for beams. (Character)
NOMASS	LROFF is used but the no mass effects are included. (Character)

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values									
	NDMTZ	LROFF is used but the element z-direction is used for the offset direction and the differential stiffness, the thermal load effects, and the mass effects are turned off. For CQUAD4 and CTRIA3 elements this method should get similar results to the standard ELMOFF method. (Character)								
	<p>Note: This entry only effects ZOFF calculations for ZOFF specified on the shell connection entries. For Z0 specified on the PCOMP or PCOMPG entries, the standard ELMOFF method will be used.</p>									
	If the computed value for SNORM is greater then the PARAM,SNORM,value and the user wishes not to change the parameter value, the Bulk Data entry SNORM can be used to override the shell normal.									
	Solution sequences affected: For linear - all solution sequences. For nonlinear - SOL 400 only. <i>The LROFF method is not implemented into SOLs 106 and 129.</i>									
PEXTS4	Used in conjunction with NSGRDS4. After partitioning the spline, each of the smaller splines will be extended by PEXTS4 in each direction (top,bottom, left and right). The value is expressed in percent so that PEXTS4=10.0 would extend the four boundaries by 10%. Real (0.0<PEXTS4<100.0, default=10.0)									
PIVTHRS4	<p>Parameter to set the threshold 'uu' for numeric pivot selection in the MSCLU and Intel MKL Pardiso solvers.</p> <table border="1" data-bbox="421 921 1322 1239"> <tr> <td data-bbox="421 921 582 1164">0 (Default)</td> <td data-bbox="582 921 1322 1164">           uu is set to           <ul style="list-style-type: none"> <li>■ MSCLU: <math>10^{-9}</math></li> <li>■ MKL Pardiso: <math>10^{-13}</math> for non-symmetric matrices and <math>10^{-8}</math> for symmetric indefinite matrices. For more general information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a>, iparm(10).</li> </ul> </td> </tr> <tr> <td data-bbox="421 1164 582 1239">-i</td> <td data-bbox="582 1164 1322 1239">uu is set to <math>10^{-i}</math> for both MSCLU and MKL Pardiso (where i is between 1 and 20)</td> </tr> </table>		0 (Default)	uu is set to <ul style="list-style-type: none"> <li>■ MSCLU: <math>10^{-9}</math></li> <li>■ MKL Pardiso: <math>10^{-13}</math> for non-symmetric matrices and <math>10^{-8}</math> for symmetric indefinite matrices. For more general information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a>, iparm(10).</li> </ul>	-i	uu is set to $10^{-i}$ for both MSCLU and MKL Pardiso (where i is between 1 and 20)				
0 (Default)	uu is set to <ul style="list-style-type: none"> <li>■ MSCLU: <math>10^{-9}</math></li> <li>■ MKL Pardiso: <math>10^{-13}</math> for non-symmetric matrices and <math>10^{-8}</math> for symmetric indefinite matrices. For more general information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a>, iparm(10).</li> </ul>									
-i	uu is set to $10^{-i}$ for both MSCLU and MKL Pardiso (where i is between 1 and 20)									
PRDIDPVT	<p>Control of pivoting in Intel MKL Pardiso, when solving symmetric indefinite matrices. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a>, iparm(21).</p> <table border="1" data-bbox="421 1308 1322 1510"> <tr> <td data-bbox="421 1308 582 1355">0</td> <td data-bbox="582 1308 1322 1355">1x1 diagonal pivoting</td> </tr> <tr> <td data-bbox="421 1355 582 1402">1</td> <td data-bbox="582 1355 1322 1402">1x1 and 2x2 Bunch-Kaufman pivoting (Default)</td> </tr> <tr> <td data-bbox="421 1402 582 1449">2</td> <td data-bbox="582 1402 1322 1449">1x1 diagonal pivoting without automatic iterative refinement</td> </tr> <tr> <td data-bbox="421 1449 582 1510">3</td> <td data-bbox="582 1449 1322 1510">1x1 and 2x2 Bunch-Kaufman pivoting without automatic iterative refinement</td> </tr> </table>		0	1x1 diagonal pivoting	1	1x1 and 2x2 Bunch-Kaufman pivoting (Default)	2	1x1 diagonal pivoting without automatic iterative refinement	3	1x1 and 2x2 Bunch-Kaufman pivoting without automatic iterative refinement
0	1x1 diagonal pivoting									
1	1x1 and 2x2 Bunch-Kaufman pivoting (Default)									
2	1x1 diagonal pivoting without automatic iterative refinement									
3	1x1 and 2x2 Bunch-Kaufman pivoting without automatic iterative refinement									

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
PRDITRFN	Control of iterative refinement step for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a> , iparm(8).	
	0	Solver automatically performs two steps of iterative refinement (Default)
	>0	Maximum number of iterative refinement steps, up to a maximum of nine.
	<0	Same as above, but residual is computed using extended precision. Maximum of nine steps.
PRDMTYPE	Control of matrix type for Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso documentation</a> , Input Parameters.	
	1	real structurally symmetric
	2	real symmetric positive definite (SOL 101 default)
	-2	real symmetric indefinite (SOL 400 default)
	3	complex structurally symmetric
	4	complex Hermitian positive definite
	-4	complex Hermitian indefinite
	6	complex symmetric (SOL 108 & 111 default)
	11	real unsymmetric
13	complex unsymmetric (SOL 107 default)	
PRDOOC	Control of out-of-core (OOC) solution algorithm in Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a> , iparm(60).	
	0	In-core mode
	1	Choose between in-core and OOC mode based on memory (opencore) size
	2	Out-of-core mode
PRDWMTCH	Control of weighted matching algorithm in Intel MKL Pardiso. For more information, see <a href="#">Intel MKL Pardiso iparm Parameter Table</a> , iparm(13).	
	0	Disable matching (SOL 101 default)
	1	Enable matching (SOL 107, 108, 111, and SOL 400 default)
PRTELAS	Print list of ID's of CELASi elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
PRTFAST	Print list of ID's of CFAST elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTMASS	Print list of ID's of CMASSi, CONM1 and CONM2 elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTSEAM	Print list of ID's of CSEAM elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
PRTWELD	Print list of ID's of CWELD elements that are deleted.	
	NO (or blank)	Turn off the print. (Default)
	YES	Turn on the print.
QR6ROT	Parameter to determine whether the drilling degrees-of-freedom are to be deactivated for QUADR/TRIAR elements. If the drilling degrees-of-freedom are deactivated, the QUADR/TRIAR become elements similar to QUAD4/TRIA3. QR6ROT has the following values:	
	0	The drilling degrees-of-freedom are active. (Default)
	1	The drilling degrees-of-freedom are deactivated for all QUADR/TRIAR element in the model.
	2	The drilling degrees-of-freedom are deactivated for those QUADR/TRIAR which have membrane stiffness only (MID2 and MID3 are blank on the PSHELL entry)
QRSHEAR	Parameter to select the off-plane shear formulation for the QUADR element. There are two types of off-plane shear formulations: the stiffness method and the flexibility method. The stiffness method is a new method implemented in QUADR. The flexibility method was the method implemented in the QUAD4 element. Therefore, if the flexibility method is selected, the solution results of QUADR are closer to those of QUAD4. QRSHEAR has the following values:	
	0	Use stiffness method if MID3 $\neq$ 0 on the PSHELL Bulk Data entry. Use the flexibility method if MID3 = 0. (Default)
	1	Use flexibility method.
	2	Use the stiffness method.

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values												
RDBOTH	<p>Parameter to select Rayleigh damping approach for rotordynamics (compatibility with MSC Nastran 2005) implementation, Integer. A cumulative sum can be provided in case multiple features are desired in the analysis. The parameter is allowed to take values of 1, 2, 4, 8, 16, or any combination of them except both 4 and 8 are included, such as 12, 13, 14 etc. Default value for this parameter is 0.</p> <table border="1" data-bbox="429 421 1326 999"> <tr> <td data-bbox="429 421 589 496">0</td> <td data-bbox="589 421 1326 496">Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)</td> </tr> <tr> <td data-bbox="429 496 589 666">1</td> <td data-bbox="589 496 1326 666">Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT is set to 0.0.</td> </tr> <tr> <td data-bbox="429 666 589 708">2</td> <td data-bbox="589 666 1326 708">Ignore circulation effects in rotordynamic analysis.</td> </tr> <tr> <td data-bbox="429 708 589 817">4</td> <td data-bbox="589 708 1326 817">Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.</td> </tr> <tr> <td data-bbox="429 817 589 927">8</td> <td data-bbox="589 817 1326 927">Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.</td> </tr> <tr> <td data-bbox="429 927 589 999">16</td> <td data-bbox="589 927 1326 999">This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.</td> </tr> </table>	0	Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)	1	Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT is set to 0.0.	2	Ignore circulation effects in rotordynamic analysis.	4	Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.	8	Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.	16	This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.
0	Uses implementation for Rayleigh Damping as described in RSPINR/RSPINT entry description (Default)												
1	Switch to V2005 implementation of Rayleigh damping where damping coefficients specified in the model through "PARAM, ALPHA1" and "PARAM, ALPHA2" are applied to the complete model and Rayleigh damping specified through "ALPHAR1" and "ALPHAR2" in RSPINR/RSPINT is set to 0.0.												
2	Ignore circulation effects in rotordynamic analysis.												
4	Include effect of stress stiffening using method = 1 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=4 is not supported in SOL 400.												
8	Include effect of stress stiffening using method = 2 (see RFORCE entry). Using RFORCE load step to include differential stiffness in SOL 400 since RDBOTH=8 is not supported in SOL 400.												
16	This option treats rotor structural damping with an imaginary stiffness matrix for complex eigenvalue analysis. See RSPINR remark 10.												
RELAXF	<p>If there are SPRELAX entries, RELAXF=1 will result in the GI module outputting the GPGK datablock without relaxation while the GDGK datablock will include the relaxation effects.</p> <p style="padding-left: 40px;">=0 (Default) Relaxation is applied to all splines =1 Relaxation is only applied to the splining of displacements.</p>												
REUPSE	<p>Choose method for handling rigid elements in superelements when RIGID=LAGRAN or LGELIM</p> <table border="1" data-bbox="429 1281 1326 1590"> <tr> <td data-bbox="429 1281 589 1558">0</td> <td data-bbox="589 1281 1326 1558">                     (Default) Automatic selection:                     <ol style="list-style-type: none"> <li>a. If ALPHA is zero for ALL r-elements in ALL superelements (SEID&gt;0) then keep of these r-elements in the superelements with RIGID=LINEAR formulation.</li> <li>b. If ALPHA is not zero for ANY r-element in ANY superelement (SEID&gt;0) then transfer ALL r-elements in ALL superelements to the residual structure.</li> </ol> </td> </tr> <tr> <td data-bbox="429 1558 589 1590">1</td> <td data-bbox="589 1558 1326 1590">Transfer all r-elements in the superelements to the residual structure.</td> </tr> </table>	0	(Default) Automatic selection: <ol style="list-style-type: none"> <li>a. If ALPHA is zero for ALL r-elements in ALL superelements (SEID&gt;0) then keep of these r-elements in the superelements with RIGID=LINEAR formulation.</li> <li>b. If ALPHA is not zero for ANY r-element in ANY superelement (SEID&gt;0) then transfer ALL r-elements in ALL superelements to the residual structure.</li> </ol>	1	Transfer all r-elements in the superelements to the residual structure.								
0	(Default) Automatic selection: <ol style="list-style-type: none"> <li>a. If ALPHA is zero for ALL r-elements in ALL superelements (SEID&gt;0) then keep of these r-elements in the superelements with RIGID=LINEAR formulation.</li> <li>b. If ALPHA is not zero for ANY r-element in ANY superelement (SEID&gt;0) then transfer ALL r-elements in ALL superelements to the residual structure.</li> </ol>												
1	Transfer all r-elements in the superelements to the residual structure.												

Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
RMRBE3RT	Parameter to control if remove rotational dof from REFC of any RBE3 which reference grid point(REFG) is used by solid elements only and REFC has rotational dof..	
	0(default)	Don't remove.
	1	Remove rotational dof.
RSTIGNDP	A flag to determine if ignoring duplicate PARAM entries between a restart run and its initial run.	
	0(default)	Don't ignore duplicating.
	>0	Ignore duplicating.
SHEARP	Parameter to select the new HARDER shear panel over the classic GARVEY panel. The HARDER panel through testing appears to be superior to the GARVEY panel especially in cases of thermal loading. (Character-all required)	
	GARVEY	GARVEY SHEAR PANEL (Default)
	HARDER	HARDER SHEAR PANEL - NEW
SHRTOQ4	Flag to determine whether to convert a CSHEAR element to a CQUAD4 element for nonlinear analysis. Cannot be used with the PSHEARN entry.	
	0	Do not convert CSHEAR to CQUAD4. (Default)
	1	Convert CSHEAR to CQUAD4.
	2	Do converting and print out converted Bulk Data entries on f06 file.
SOLCTLBC	Backward compatibility in SOL 400 with older Nastran versions. (Integer; Default = 0). Each BIT of this flag can be turned on individually.	
	0	Updating the calculation of residual force with damping effect; releasing residual immediately in the beginning of subsequent load steps.
	-1	Activate all of following flags (back to 2020 SP1).
	1	Old logic which misses NDAMP effect in residual force calculation.
	2	Ramping down residual gradually in subsequent load steps.
SPBLNDX	Factor to be applied to D1 and D2 blend depths on the SPBLND1 and SPBLND2 bulk data entries for determining the structural grids in the blended region. (Real, $\geq 1.0$ , default = 1.0)	
STREQCNT	Flag to control calculation method for equivalent von Mises stress/strain at element center for advanced nonlinear solid elements in SOL 400.	
	0	Average integration point equivalent stress/strain (Default).
	1	Average integration point stress/strain components then compute equivalent value at element center.



Table 20 PARAMi Names and Descriptions

Name	Description, Type, and Default Values	
TWBRBML	Parameter to select method for computing properties of PBARL/PBEAML.	
	0	Select Finite Element Method. (Default)
	1	Select Beam Library Equations.
BCSOL129	Controls the convergence parameter computation method, iteration information printing format, and divergence solution sequence checking for SOL 400.	
	0	The current format. (Default) Two formats are available: static and dynamic.
		Static format will be used for static analysis.
		Dynamic format will be used for dynamic analysis.
-1	Using the format and algorithm similar to SOL 129.	

## MDMIAUX

## Matrices from Marc - SOL 600

Specifies the DOMAINSOLVER command to be used in conjunction with secondary spawned jobs when MDMIOUT is used. SOL 600 only. (See the MDMIOUT Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MDMIAUX	ID								
	"FMS"	STRF1							
		STRF2							
	"EXEC"	STRE1							
		STRE2							

Example:

MDMIAUX	2									
	FMS	ASSIGN OUTPUT2=MODEL1.OP2,UNIT=50,DELETE								
	EXEC	DOMAINSOLVER ACMS (PRINT=YES,UPFACT=3.5)								

Describer	Meaning
ID	ID of a matching IAUX field on the corresponding MDMIOUT entry. (Integer; no Default)
"FMS"	Enter the string FMS to start a series of FMS strings that should appear before the SOL entry for the secondary spawned job. (Character; no Default)
STRFi	Series of FMS strings, entry as many as desired. (Character; no Default)
"EXEC"	Enter the string EXEC to start a series of strings that should appear after the SOL entry for the secondary spawned job. (Character; no Default)
STREi	Series of Executive Control strings, enter as many as desired. (Character; no Default)

Remarks:

1. This entry is not active unless a matching MDMIOUT entry is found in the input.
2. STRFi and STREi may occupy fields 3-9. (columns 9 to 72).
3. Standard FMS and Executive Control continuation rules apply to the strings.

## MDMIOUT

## Matrices from Marc - SOL 600

Defines full or reduced stiffness and mass matrices to be output from the Marc portion of SOL 600. This entry may be used to generate External Superelements using DMIG Matrices or an MSC Adams MNF File from the Marc portion of a SOL 600 analysis. SOL 600 only. (See the MNF600 and DMIGOUT Bulk Data entries.)

Format:

1	2	3	4	5	6	7	8	9	10
MDMIOUT	ID	IDOF	G1	THRU	G2	ITYPE	NAME	IAUX	
	IDOF2	G3	THRU	G4	IDOF3	G5	THRU	G6	
	etc.								

Example:

MDMIOUT	100	123456	1	THRU	5456	1	m343	1	
---------	-----	--------	---	------	------	---	------	---	--

Describer	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. (Integer; Default = 1)
IDOF, IDOF2	List of DOF's to be output (any or all of the integers 1-6 are acceptable). (Integer; Default = 123456)
G1, G3	Starting grid ID for reduced matrices. (Integer; Required; no Default)
G2, G4	Ending grid ID for reduced matrices. (Integer; Required; no Default)
ITYPE	Type of file to generate. (Integer; Default = 1) 1           DMIG Matrices 2           Adams MNF
NAME	Name of file containing the output (Character; no Default; limited to 8 characters, should be in lower case, <b>the extension .dmi is added automatically</b> )
IAUX	ID of a MDMIAUX entry if the secondary run requires the use of file management entries or additional Executive Control statements such as the DOMAINSOLVER option for parallel processing. (Integer; Default = 0) A value of zero means that no auxiliary entries are required.
ISOL	Solution sequence to run using the DMIG matrices. To speed up the solution, use DOMAINSOLVER ACMS (PARTOP=DOF) for eigenvalues and set ISOL to the negative value of the solution sequence desired (-103, -111 or -112). (Integer absolute value > 100; Default = 0 which means do not run any solution sequence using the DMIG's created by Marc in this execution)

## Remarks:

1. The continuation line(s) are not required.
2. This entry corresponds to Marc's entry, SUPERELEM with a value of 1 in the second line 4th field and produces DMIG's or an MDF file for the initial geometry prior to any nonlinear iterations.
3. DMIG output will be in `jid.marc_dmigst_0001`.
4. The reduced matrices may be used in the Nastran analysis for eigenvalue extraction or any other purpose by invoking the `CONTINUE=5` option on the SOL 600 entry.
5. If the SOL 600 CONTINUE options is invoked, case control commands and a bulk data entry include statements to receive the matrices will be automatically added to the original input data file. A second Nastran execution will be spawned from the original Nastran execution after completion of the Marc execution.
6. ID must be 106 or 129 in the Executive Control statement, SOL 600, ID.
7. Only one MDMIOUT entry should be entered per run. If more are entered, only the first will be used.
8. MNF controls for other solution sequences are ignored for SOL 600.
9. For the case where DMIG's are generated and a continuation option is used, the following Bulk Data parameters are usually required in addition to the MDMIOUT entry:  

```
$2345678x234567890123456x34567890123456  
param   marcfile           nastb.rc  
param   marcfil1          m3356
```
10. For a more general form of the DMIG output, see Bulk Data entry, DMIGOUT.

## MDMIR1

## Module Reposition by Mirroring - Option 1

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMIR1	MVID	P1	P2	P3					

Example:

MDMIR1	110	12	45	1125					
--------	-----	----	----	------	--	--	--	--	--

Describer	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
Pi	GRID or POINT identification numbers of three non-collinear points in Module 0. (Integer>0).

Remarks:

1. MDMIR1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRIDs or POINTs referenced on this entry must be defined in the main Bulk Data Section (Module 0).

## MDMIR2

## Module Reposition by Mirroring - Option 2

Defines a Module mirror by specifying a pair of coordinate system axes on the mirror plane in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMIR2	MVID	CID	RID	AXES					

Example:

MDMIR2	110		45	XZ					
--------	-----	--	----	----	--	--	--	--	--

Describer	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
CID	Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be mirrored about CID's origin if RID is blank. (Integer ≥ 0, default=0)
RID	GRID or POINT identification number or coordinates of reference point through which mirror plane will be applied. If RID is blank, then the origin of CID will be used (Integer ≥ 0, default=0)
AXES	Coordinate system axes pair: "XY", "XZ", "YZ", "YX", "ZX", or "ZY" (Character).

Remarks:

1. MDMIR2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDMIR2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDMOVE**

Defines a Module repositioning sequence

Defines a Module repositioning sequence by referencing MDMIRi, MDROTi, and MDTRAN entries.

Format:

1	2	3	4	5	6	7	8	9	10
MDMOVE	MODID	MVID1	MVID2	MVID3	-etc.-				

Example:

MDMOVE	40	10	11	12					
--------	----	----	----	----	--	--	--	--	--

Describer	Meaning
MODID	Module identification number of a primary or copied module (Integer>0)
MVIDi	Move identification number of an MDTRAN, MDROTi or MDMIRi Bulk Data entry (Integer>0).

Remarks:

1. MDMOVE can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. In order to use MDMOVE, an MDBULK entry with the same MODID must be specified with TYPE="MOVE".
3. If MDMIRi is referenced on an MDMOVE then its MVID must be specified in the MVID1 field and not in the second or subsequent MVIDi fields.

## MDMPC

## Multipoint Constraint Between Two or More Modules

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where  $u_j$  represents degree-of-freedom  $C_j$  at grid or scalar point  $G_j$ .  $G_j$  may be contained in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDMPC	SID	MOD1	G1	C1	A1				
	MOD2	G2	C2	A2	MOD3	G3	C3	A3	
	MOD4	G4	C4	A4	-etc-				

Example:

MDMPC	3	10	28	3	6.2				
	11	2		4.29	21	1	4	-2.91	

Descriptor	Meaning
SID	Set identification number. (Integer > 0)
MODj	Module IDs that contain the GRID entries for $G_j$ . (Integer ≥ 0).
$G_j$	Identification number of grid or scalar point. (Integer > 0)
$C_j$	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
$A_j$	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

1. The MDMPC entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the MPC entry description. All remarks and figures under MPC apply to MDMPC.



**MDMPLN****Module Reposition by Mirroring**

Defines reposition of a Module with a mirror plane defined by three non-collinear points in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDMPLN	MODID	P1	P2	P3					

Example:

MDMPLN	110	12	45	1125					
--------	-----	----	----	------	--	--	--	--	--

Describer	Meaning
MODID	Module identification number of a primary or secondary (copied) module (Integer>0).
Pi	GRID or POINT identification numbers of three non-collinear points in Module 0 (Integer>0).

Remarks:

1. MDMPLN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MDMPLN=n command.
2. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

## MDRBE2

## Rigid Body Element Between Two or More Modules, Form 2

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points. All grid points may be defined in different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDRBE2	EID	MODN	GN	CM	MOD1	GM1	MOD2	GM2	
	MOD3	GM3	MOD4	GM4	MOD5	GM5	MOD6	GM6	
	MOD7	GM7	MOD8	GM8	-etc.-	ALPHA	TREF		

Example:

MDRBE2	9	101	8	12	201	10	202	12	
	202	14	203	15	203	16	6.5-6		

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000).
MODN	Module ID that contains the GRID entry for GN. (Integer ≥ 0).
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. See Remark 12. under RBE2. (Integers 1 through 6 with no embedded blanks.)
MODi	Module IDs that contain the GRID entries for GMj. (Integer ≥ 0).
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. See Remark 11. under RBE2. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

Remarks:

1. The MDRBE2 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE2 entry description. All remarks and figures under RBE2 apply to MDRBE2.

**MDRBE3****Interpolation Constraint Element Between Two or More Modules**

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

Format:

1	2	3	4	5	6	7	8	9	10
MDRBE3	EID	REFMOD	REFGRID	REFC	WT1	C1	MOD1,1	G1,1	
	MOD1,2	G1,2	MOD1,3	G1,3	WT2	C2	MOD2,1	G2,1	
	MOD2,2	G2,2	-etc.-	WT3	C3	MOD3,1	G3,1	MOD3,2	
	G3,2	-etc.-	WT4	C4	MOD4,1	G4,1	MOD4,2	G4,2	
	-etc.-								
	“UM”	MOD1	GM1	CM1	MOD2	GM2	CM2		
		MOD3	GM3	CM3	-etc.-				
	“ALPHA”	ALPHA	TREF						

Example:

MDRBE3	14	1000	100	1234	1.0	123	101	1	
	102	3	103	5	4.7	1	101	2	
	102	4	103	6	5.2	2	101	7	
	102	8	103	9	5.1	1	104	15	
	105	16							
	UM	101	100	14	102	5	3		
		103	7	2					
	ALPHA	6.5-6							

Describer	Meaning
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFMOD	Module IDs that contain the GRID entry for REFGRID. (Integer ≥ 0).
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WT <sub>i</sub>	Weighting factor for components of motion on the following entry at grid points G <sub>i,j</sub> . (Real)
C <sub>i</sub>	Component numbers with weighting factor WT <sub>i</sub> at grid points G <sub>i,j</sub> . (Any of the integers 1 through 6 with no embedded blanks.)
MOD <sub>i,j</sub>	Module IDs that contain the GRID entries for G <sub>i,j</sub> . (Integer ≥ 0).

Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
“UM”	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
MODk	Module IDs that contain the GRID entries for GMk. (Integer ≥ 0).
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
“ALPHA”	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark 14. under RBE3. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

Remarks:

1. The MDRBE3 entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RBE3 entry description. All remarks and figures under RBE3 apply to MDRBE3.

## MDRJNT

## Rigid Joint Between Two Modules

Defines a rigid joint element connecting two coinciding grid points in two different Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDRJNT	EID	MODA	GA	MOdB	GB	CB			

Example:

MDRJNT	5	11	1	21	2	12345		1	
--------	---	----	---	----	---	-------	--	---	--

Describer	Meaning
EID	Element identification number. (Integer > 0)
MODA,MOdB	Module IDs that contain the GRID entries for GA and GB in Modules MODA and MOdB, respectively. (Integer ≥ 0).
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. under RJOINT. (Integers 1 through 6 with no embedded or blank.)

Remarks:

1. The MDRJNT entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RJOINT entry description. All remarks and figures under RJOINT apply to MDRJNT.

## MDROT1

## Module Reposition by Rotation - Option 1

Defines a Module rotation by specifying a rotation vector and reference point in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDROT1	MVID	RID	IDS	IDE	MAG				

Example:

MDROT1	110		45	72	90.				
--------	-----	--	----	----	-----	--	--	--	--

Describer	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
RID	GRID or POINT identification number of a reference point at which rotation will occur. If blank or 0 then Module will be rotated about rotation vector defined from IDS to IDE. (Integer ≥ 0, Default=0).
IDS	GRID or POINT identification number of rotation vector's starting point. (Integer>0)
IDE	GRID or POINT identification number of rotation vector's end point. (Integer>0)
MAG	Magnitude of rotation in degrees. (Real ≠ 0.0).

Remarks:

1. MDROT1 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE=n command.
2. MDROT1 must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDROT2****Module Reposition by Rotation - Option 2**

Defines a Module rotation by specifying a coordinate system axis for the rotation vector.

Format:

1	2	3	4	5	6	7	8	9	10
MDROT2	MVID	CID	RIDS	AXIS	MAG				

Example:

MDROT2	40		45	Y	90.				
--------	----	--	----	---	-----	--	--	--	--

Describer	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
CID	Coordinate system identification number of a CORDij entry. Blank or zero means basic coordinate system. Module will be rotated about CID's origin if RID is blank. (Integer ≥ 0, default=0).
RID	GRID or POINT identification number of a reference point at which rotation will occur. If RID is blank or 0 then the origin of CID will be used. (Integer ≥ 0, Default=0)
AXIS	Coordinate system axis: "X", "Y", "Z", "MX", "MY", or "MZ". The latter three values indicate the negative side of the axis. (Character)
MAG	Magnitude of rotation in degrees. (Real ≠ 0.0).

Remarks:

1. MDROT2 can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDROT2 must be referenced by an MDMOVE entry in order to reposition a Module.
3. CORDij, GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

## MDRROD

## Rigid Pin-Ended Element Connection Between Two Modules

Defines a pin-ended element that is rigid in translation between two Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDRROD	EID	MODA	GA	MODB	GB	CMA	CMB	ALPHA	
	TREF								

Example:

MDRROD	14	11	1	21	2	2		6.5-6	
--------	----	----	---	----	---	---	--	-------	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
MODA,MODB	Module IDs that contain the GRID entries for GA and GB (Integer ≥ 0).
GA, GB	Grid point identification numbers of connection points in Modules MODA and MODB, respectively. (Integer > 0)
CMA,CMB	Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. See Remark 3. under RROD. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear MDRROD. For Lagrange MDRROD, both CMA and CMB can be blank.)
ALPHA	Thermal expansion coefficient. See Remark 11. under RROD. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

Remarks:

1. The MDRROD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the RROD entry description. All remarks and figures under RROD apply to MDRROD.



**MDSEAM****A Shell Patch SEAM Connection Between Two Modules**

Defines a SEAM connecting two surface patches in two Modules.

Format:

1	2	3	4	5	6	7	8	9	10
MDSEAM	EID	PID	SMLN	CTYPE	IDAS	IDBS	IDAE	IDBE	
	GS	GE					MODA	MODB	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MDSEAM	EID	PID		CTYPE	IDAS	IDBS	IDAE	IDBE	
	XS	YS	ZS	XE	YE	ZE	MODA	MODB	

Example:

MDSEAM	552	297			43	48			
	30422	77987					15	32	

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
PID	Property identification number of a PSEAM entry. (Integer > 0)
SMLN	SEAM line identification. See Remark 2. under CSEAM. (CHAR or blank)
CTYPE	Connectivity search type. (Character)
	If CTYPE = "PSHELL", IDAS and IDBS are property identification numbers of PSHELL's. (Default)
	If CTYPE = "ELEM", IDAS and IDBS are element identification numbers.
IDAS, IDBS	Used to define patch A and B in Modules MODA and MODB or the start of patch A or B for a tailored blank in Modules MODA and MODB. See Remark 2. under CSEAM (Integer > 0)
	If CTYPE = "PSHELL", required property id defining patches A and B in Modules MODA and MODB. If CTYPE = "PSHELL" and IDAS = IDBS or IDBS = blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom. See Remark 6. under CSEAM.
	If CTYPE = "ELEM", required element id defining patches A and B in Modules MODA and MODB. IDAS ≠ IDBS.

Describer	Meaning
IDAE, IDBE	<p>Used to define the end of patch A and the end of patch B for a tailored blank in Modules MODA and MODB. See Remark 4. under CSEAM. (Integer &gt; 0 or blank)</p> <p>If CTYPE = "PSHELL", property id defining patches A and B. If CTYPE = 'PSHELL' and IDAE = IDBE or IDBE=blank the patch will be considered as two-sided and the property identification numbers of PSHELL's will be the same for both the top and bottom.</p> <p>If CTYPE = "ELEM", element id defining patches A and B in Modules MODA and MODB. IDAE ≠ IDBE.</p>
GS, GE	Grid ids of piercing points on patches A and B of the Start and End of the SEAM. Their GRID entries must be defined in the main Bulk Data section (Module 0). (Integer > 0)
XS, YS, ZS	Location of the SEAM Start. (Real or blank)
XE, YE, ZE	Location of the SEAM End. (Real or blank)
MODA, MODB	Module IDs that contain the surface patches defined by property or element entries IDAS/IDAE and IDBS/IDBE, respectively (Integer ≥ 0).

Remarks:

1. The MDSEAM entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CSEAM entry description. All remarks and figures under CSEAM apply to MDSEAM.

## MDSSET

### Set Definition for Structural Elements or Grid Points in Modules

Defines a set of structural grid points or element identification numbers in a Module to be referenced by the ACMODL Bulk Data entry.

Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>
MDSSET	MODID	ID1	ID2	ID3	ID4	ID5	ID6	ID7
	ID8	-etc.-						

Example 1:

MDSSET	22	31	62	93	124	16	17	18
	19							

Example 2:

MDSSET	22	29	32	THRU	50	61	THRU	70
	17	57						

Describer	Meaning
MODID	Module identification number (Integer ≥ 0)
IDI	List of structural grid point or element identification numbers. (Integer > 0 or “THRU”; for the “THRU” option: ID1 < ID2).

Remarks:

1. The MDSSET entry can only be used in the presence of Modules and can only be specified in the main Bulk Data section or Module 0.
2. The MDSSET entry is used when SSET is set to -1 on the ACMODL entry.
3. Multiple MDSSET entries may reference more than one Module.

## MDTRAN

## Module Reposition by Translation

Defines a Module translation by specifying a vector in Module 0.

Format:

1	2	3	4	5	6	7	8	9	10
MDTRAN	MVID	IDS	IDE	MAG					

Example:

MDTRAN	110	72	45						
--------	-----	----	----	--	--	--	--	--	--

Describer	Meaning
MVID	Move identification number to be referenced on an MDMOVE entry. (Integer>0).
IDS	GRID or POINT identification number of translation vector's starting point. (Integer>0).
IDE	GRID or POINT identification number of translation vector's end point. (Integer>0).
MAG	Magnitude of translation. If blank or zero, then the length of the vector will be used. (Real, Default=0.0)

Remarks:

1. MDTRAN can only be specified in the main Bulk Data Section (Module 0) and will cause a fatal error message if it appears after the BEGIN MODULE = n command.
2. MDTRAN must be referenced by an MDMOVE entry in order to reposition a Module.
3. GRID or POINT entries referenced on this entry must be defined in the main Bulk Data Section (Module 0).

**MDWELD****Weld or Fastener Element Connection Between Two Modules**

Defines a weld or fastener connecting two surface patches or points between two Modules.

Format PARTPAT:

1	2	3	4	5	6	7	8	9	10
MDWELD	EWID	PWID	GS	"PARTPAT"	GA	GB		MCID	
	PIDA	PIDB			MODA	MODB			
	XS	YS	ZS						

Example:

MDWELD	101	8	203	PARTPAT					
	21	33			11	21			

Alternate formats and examples:

Format ELPAT:

1	2	3	4	5	6	7	8	9	10
MDWELD	EWID	PWID	GS	"ELPAT"	GA	GB		MCID	
	SHIDA	SHIDB			MODA	MODB			
	XS	YS	ZS						

Example:

MDWELD	103	5	403	ELPAT					
	309	511			11	21			

Format ELEMID:

MDWELD	EWID	PWID	GS	"ELEMID"	GA	GB		MCID	
	SHIDA	SHIDB	MODS		MODA	MODB			

Example:

MDWELD	103	5	403	ELEMID					
	309	511			11	12			

Format GRIDID:

MDWELD	EWID	PWID	GS	"GRIDID"	GA	GB	SPTYP	MCID	
					MODA	MODB			
	GA1	GA2	GA3	GA4	GA5	GA6	GA7	GA8	
	GB1	GB2	GB3	GB4	GB5	GB6	GB7	GB8	

Example:

MDWELD	7	29	233	GRIDID			QT		
					11	21			
	15	28	31	35	46	51	55	60	
	3	5	8						

Format ALIGN:

MDWELD	EWID	PWID		"ALIGN"	GA	GB		MCID	
					MODA	MODB			

Example:

MDWELD	7	29		ALIGN	103	259			
					11	21			

Describer	Meaning	Type	Default
EWID	MDWELD element identification number. See Remark 1 under CWELD.	0 < Integer < 100,000,000	Required
PWID	Property identification number of a PWELD entry.	Integer > 0	Required
GS	Identification number of a grid point which defines the location of the connector. GRID entry for GS must be defined in main Bulk Data section only (Module 0) except for a point-to-patch connection with OPTION="ELEMID" in which case GS must be defined in Module MODS. See Remarks 2. and 3. under CWELD.	Integer > 0 or blank	

Describer	Meaning	Type	Default
	Character string indicating the type of connection. The format of the subsequent entries depends on the type. "PARTPAT", for example, indicates that the connectivity of surface patch A to surface patch B is defined with two property identification numbers of PSHELL entries, PIDA in Module MODA and PIDB in Module MODB, respectively. The "PARTPAT" format connects up to 3x3 elements per patch. See Remark 4. under CWELD.	Character	Required
GA, GB	Grid point identification numbers of piercing points on surface A and surface B, respectively. GRID entries for GA and GB must be defined in the main Bulk Data section only (Module 0) except for OPTION="ALIGN". See Remark 5. under CWELD.	Integer > 0 or blank	Blank
MCID	Specifies the element stiffness coordinate system. See Remark 16. under CWELD	Integer > -1 or blank	Default = -1
MODS	Module ID that contains GRID entry for GS. Only used for point-to-patch connection with OPTION="ELEMID".	Integer ≥ 0	Default = 0
MODA, MODB	Module IDs that contain the surface patches defined by (1) property entries PIDA and PIDB, (2) element entries SHIDA and SHIDB or (3) GRID entries GAI and GBi, respectively.	Integer ≥ 0	Default = 0
PIDA, PIDB	Property identification numbers of PSHELL entries defining surface A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for "PARTPAT"
XS, YS, ZS	Coordinates of spot weld location in basic. See Remark 2. under CWELD.	Real	Required if GS and GA are not defined.

For the alternate formats, the describer meaning are described below:

Describer	Meaning	Type	Default
“ELPAT”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The “ELPAT” format connects up to 3x3 elements per patch. See Remark 6. under CWELD.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for “ELPAT”
“ELEMID”	Character string indicating that the connectivity of surface patch A to surface patch B is defined with two shell element identification numbers, SHIDA in Module MODA and SHIDB in Module MODB, respectively. The “ELEMID” format connects one shell element per patch. See Remark 7. under CWELD.	Character	Required
SHIDA, SHIDB	Shell element identification numbers of elements on patch A in Module MODA and B in Module MODB, respectively.	Integer > 0	Required for “ELEMID”
“GRIDID”	Character string indicating that the connectivity of surface patch A in Module MODA to surface patch B in Module MODB is defined with two sequences of grid point identification numbers, GAi in Module MODA and GBi in Module B, respectively. The “GRIDID” format connects the surface of any element. See Remark 8. under CWELD.	Character	Required
SPTYP	Character string indicating types of surface patches A and B. SPTYP = “QQ”, “TT”, “QT”, “TQ”, “Q” or “T”. See Remark 9. under CWELD.	Character	Required for “GRIDID”



Describer	Meaning	Type	Default
GAi	Grid identification numbers of surface patch A. GA1 to GA3 are required. Their GRID entries must be defined in Module MODA only. See Remark 10. under CWELD.	Integer > 0	Required for "GRIDID"
GBi	Grid identification numbers of surface patch B. Their GRID entries must be defined Module MODB only. See Remark 10. under CWELD.	Integer > 0	
"ALIGN"	Character string indicating that the connectivity of surface A to surface B is defined with two shell vertex grid points GA in Module MODA and GB in Module MODB, respectively. See Remark 11. under CWELD.	Character	Required
GA, GB	Vertex grid identification number of shell A and B, respectively. GA and GB GRID entries must be defined in Module MODA and Module MODB, respectively.	Integer > 0	Required for "ALIGN"

#### Remarks:

1. The MDWELD entry must be defined in the main Bulk Data section only (Module 0).
2. See all remarks and figures under the CWELD entry description. All remarks and figures under CWELD apply to MDWELD.

## MESH

## Mesh Generator

Defines a mesh. Used in SOL 700 only.

Format:

1	2	3	4	5	6	7	8	9	10
MESH	MID	TYPE	DXEL	DYEL	DZEL	XREF	YREF	ZREF	
	X0	Y0	Z0	DX	DY	DZ			
	NX	NY	NZ	SUBMESH	NSTGP	NSTEL	PROP	PID	
	RESIZE	TID-X	TID-Y	TID-Z	METHOD				
	BIAS	GROWX	GROWY	GROWZ	IBIDX	IBIDY	IBIDZ		
	XOBY	YOBY	ZOBY	DXBY	DYBY	DZBY			
	CID								
	NELCUBE	NBX	NBY	NBZ	PROCDIR	NPX	NPY	NPZ	

Example:

MESH	1	ADAPT	0.1	0.2	0.3				
	SCALE	101			ALL				
	CENTER	1.2	1.2	1.2					

Descriptor	Meaning
MID	Unique MESH number. See Remark 1. (Integer > 0; no Default)
TYPE	Type of mesh generation: See Remark 1. (Character; Required)
ADAPT	An Euler mesh will be created around a coupling surface. This option is only valid for PROP=EULER, and requires that the MID of the MESH is referenced from the MESHID of a COUPLE card. During the simulation, when the coupling surface moves or deforms, the Euler mesh will adapt itself by adding and removing elements. The adapt algorithm ensures that the coupling surface is contained inside the Euler mesh at all times with the minimum amount of elements. The Euler elements are aligned with the basic coordinate.
BOX	A rectangular mesh will be created, that is aligned with the basic coordinate system. The mesh will use CHEXA elements.
DXEL,DYEL, DZEL	Euler element sizes. See Remark 1. (Real)

Describer	Meaning
XREF,YREF,ZREF	Coordinates of reference point. For TYPE=ADAPT, these coordinates provide control over the location of the Euler mesh, to avoid that faces of the Euler mesh are initially at the same location as faces of the coupling surface. For TYPE=BOX, these coordinates will be used as the origin of the mesh. They are the default setting for (X0, Y0, Z0). (Real; Default = -1e-6)
X0,Y0,Z0	Coordinates of point of origin. (Real, XREF, YREF, ZREF) Not used for TYPE=ADAPT
DX,DY,DZ	Width of mesh in different directions. For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Real)
NX,NY,NZ	Number of elements in the different directions. For TYPE=ADAPT, these values will only be used if (DXEL, DYEL, DZEL) are left blank. See Remark 1. for more detail. (Integer > 0)
SUBMESH	Allows using smaller mesh sizes for a part of the mesh. SUBMESH is the MESH ID of a finer mesh that is to replace part of the mesh. See Remark 8. (Integer ≥ 0, default=0)
NSTGP	Starting grid-point number. Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting grid-point number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)
NSTEL	Starting element number. Not used for TYPE=ADAPT. If there are multiple couple surfaces then the starting element number can only be specified if param,flow-method,facet has been activated. See Remark 2. (Integer > 0)
PROP	Property type: (Character; default=EULER) EULER            An Eulerian mesh will be created.
PID	ID of a PEULER or PEULER1 entry. (Integer > 0, Required.)
RESIZE	Only valid for TYPE=ADAPT. See Remark 6. (Character, NONE) Option to change the element size during the simulation: NONE            No resizing of DX,DY,DZ during simulation. SCALE           The elements are resized by a scale-factor as a function of time. LENGTH        The elements are resized by specifying the length as a function of time.
TID-X	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; Blank)
TID-Y	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)
YID-Z	ID of a TABLED1. See RESIZE for contents of table. It must define a step function. See Remarks 4. and 5. (Integer > 0; TID-X)
METHOD	Method for determining when to create Euler elements: (Character; ALL)

Describer	Meaning
	<p>ALL Always re-mesh any existing Euler element. Maintains existing void regions. Only used for resizing.</p> <p>MATERIAL Only re-mesh those Euler elements that contain material. Removes void regions. See Remark 7.</p>
BIAS	Adds bias to the mesh. (Character; Blank)
	<p>CENTER Starting at the center of the BOX the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center.</p> <p>REF Starting at the reference point the mesh size gradually changes such that the mesh size at the boundaries of the BOX is GROWX times the mesh size at the center</p>
GROWX, GROWY, GROWZ	Total grow factor. Is the ratio between finest and coarsest element size. (Real > 0; Only required if BIAS is not blank)
IBIDX, IBIDY, IBIDZ	BIAS ID reference. See Remarks 9. and 10. (Integer; Default = 0)
X0BX,Y0BX,Z 0BX,DXBX,D YBX,DZBX	Definition of an auxiliary box for output purposes. By defining an auxiliary box all adaptive elements that are within the box for one of the cycles requested are stored in the archive. This allows multiple cycles in one Euler archive. This box should be sufficiently large such that it contains all elements. The fields X0BX, Y0BX and Z0BX specify the start point and DXBX, DYBX and DZBX specify the width of box. If there are adaptive elements outside the box the run is terminated and a larger box needs to be specified. Only used for TYPE=ADAPT. (Real; Blank) See Remark 14.
CID	ID - number of a local coordinate system. (Integer ≥ 0; Default=0)
NELCUBE	The number of elements per cube. This number is used as a guideline. The actual number used per cube can differ and can be found in the OUT file. See Remark 12. (Integer ≥ 1; Default=1)
NBX	Overrules NELCUBE. The number of cubes in the x-direction. (Integer ≥ 0; Default=1)
NBY	The number of cubes in the y-direction.(Integer ≥ 0; Default=NBX)
NBZ	The number of cubes in the z-direction. (Integer ≥ 0; Default=NBX)
PROCDIR	This directive controls the way cubes are distributed across processors. The effect can be checked by checking the Eulerian output variable PARTITION. (Char, Default=X)
	<p>X Partition in global X direction first.</p> <p>Y Partition in global Y direction first.</p> <p>Z Partition in global Z direction first.</p> <p>USER Define user defined partitioning.</p>

Describer	Meaning
	SIMPLE Partition Euler cubes in a simple pattern.
NPX	The number of cubes in the x-direction. Required for PROCDIR=USER. (Integer, Default=1)
NPY	The number of cubes in the y-direction. (Integer, Default=NPX)
NPZ	The number of cubes in the z-direction. (Integer, Default=NPX)

#### Remarks:

1. The grid-points of the mesh are generated at following locations:

Type=ADAPT:  $(x,y,z)=(XREF+i*DXEL, YREF+j*DYEL, ZREF+k*DZEL)$

Grid-points and elements located a certain distance outside the coupling surface will not be created. This saves memory and CPU time.

When (XREF, YREF, ZREF) are outside the coupling surface, no grid-point will be created at this location, but the mesh will be shifted appropriately.

Type=BOX:  $(x,y,z) = (X0+i*DXEL, Y0+j*DYEL, Z0+k*DZEL)$

Nodes and Elements will always be created, even if the MESH is referenced from the MESHID of a COUPLE entry.

One of the following input combinations is required:

TYPE=ADAPT

- a. (DXEL, DYEL, DZEL)

or

- b. 1

- c. (DX, DY, DZ) and (NX, NY, NZ)

➔  $DXEL = DX/NX$  ;  $DYEL=DY/NY$  ;  $DZEL=DZ/NZ$

TYPE=BOX

- a. (DXEL, DYEL, DZEL) and (NX, NY, NZ)

or

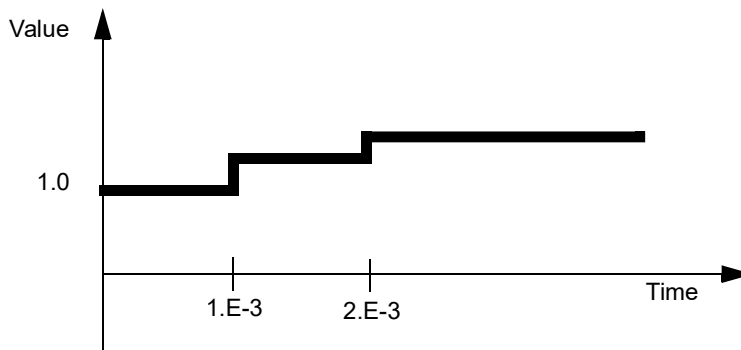
- b. (DX, DY, DZ) and (NX, NY, NZ)

➔  $DXEL = DX/NX$  ;  $DYEL=DY/NY$  ;  $DZEL=DZ/NZ$

2. When the starting grid point and/or element number is left blank, then the default start number for the elements and grid-points is equal to the maximum number used +1. For simulations with multiple coupling surfaces two methods of treating transport between the Euler meshes are available. One method supports meshes of TYPE = ADAPT, but does not allow the specification of starting element or starting grid-point number.
3. The PID should refer to an existing property id, which can handle the property type given by PROP.

4. To avoid that the Euler mesh will be resized every time-step, the functions defined by TID-X, TID-Y, TID-Z must describe a 'step-function', like in this example:
- i. TABLED1,1,,,,,,,,,+
  - ii. +,0.0,1.0,,,,,,,,,+
  - iii. +,1.E-3,1.0,,,,,,,,,+
  - iv. +,1.E-3,1.1,,,,,,,,,+
  - v. +,2.E-3,1.1,,,,,,,,,+
  - vi. +,2.E-3,1.2

Which specifies following function:



5. Care must be taken when refining the Euler mesh. To avoid instabilities, it is advised to stay within the following guidelines:
  - a. Each refining step, use a scale factor larger than 0.5
  - b. Allow the solution to become smooth again after each refining step. For airbag simulations, use an interval larger than  $5 * \text{diameter\_airbag} / \text{soundspeed}$
6. Resizing is not available for the Multi-material solver.
7. In most cases METHOD = ALL is the preferred method. Using METHOD=MATERIAL may be helpful in case of instabilities due to presence of void regions.
8. SUBMESH glues a fine mesh into a coarse mesh and uses the same gluing functionality as PARAM, GRADED MESH. If the fine mesh is completely contained inside the coarse mesh no restrictions apply. Then to avoid any restrictions the grid points of the fine mesh are slightly displaced. But if parts of the fine mesh are outside the coarse mesh a restriction applies. In that case an Euler element of the coarse mesh has to be fully active or fully inactive. This means that the coarse element should not intersect elements of the fine mesh or it should be fully covered by the fine elements. Fine elements are not allowed to cover any part of the coarse elements. In practice, this means that the fine mesh has to fit nicely in the coarse mesh. For details refer to the section "Graded meshes" in the MSC Nastran Explicit Nonlinear User's Manual.

When running on one cpu the elements of the Euler mesh and the submesh will be put into one euler archive. But when running with multiple cpus, the mesh and sub mesh will be put in different euler archives. To distinguish the archives, the name `_FVX` is added to the Euler archive names. Here X is the MESH-ID MID. The Euler archive of the mesh and the submesh can be read simultaneously into Patran.

9. A biased mesh has nonconstant element sizes in selected directories. Neighbor element size can have a constant ratio or have identical size. In literature, this type of mesh is also referred to as a nonuniform mesh or a locally refined mesh.
10. A block mesh consists of a number of planes in all three directions. For a nonbiased mesh, these planes are at fixed distance from each other. In a biased mesh, the distance between subsequent planes can differ. The varying element size is determined by:
  - a. IBIDX
  - b. GROWX
  - c. The constant step size specified by X0, NX, DX.

Here, IBIDX overrules GROWX and GROWX overrules the X0, NX DX specification. Likewise, for the other directions. The locations of the planes are written out in the OUT file. Intersecting an x-plane with a y-plane and z-plane will give a grid point. By carrying out all intersections, the grid points are constructed.

11. Defining the CID allows for positioning the mesh box arbitrarily in space. When active, the following restrictions apply:
  - MESH,ADAPT cannot be used
  - The use of single material Euler with strength elements is not supported
12. Setting NELCUBE, NBX, NBY, NBZ allow to divide the Euler mesh into a number of cubes. By setting NELCUBE equal to 2000, optimal use is made of memory caching during Euler computation. This can give a speedup of 1.5. If NBX is defined, also NBY and NBZ need to be defined. Defining NBX overrules the definition of NELCUBE.

When using multiple Euler cubes the BARRIER and FLOW definitions only support geometric conditions like boundary face direction and square definition.

13. There are several ways to distribute cubes across processors. Some ways may lead to bad load balancing. To avoid this it is possible to control the way Euler cubes are distributed across processors by defining PROCDIR.

When option PROCDIR=USER, the values for NBX, NBY, and NBZ must be such that NBX is equal or a multiple of NPX, NBY is equal or a multiple of NPY and NBZ is equal or a multiple of NPZ. Also for this option,  $NPX \cdot NPY \cdot NPZ$  must be equal to the number of processors used in the cluster.

For option PROCDIR=SIMPLE, the values NBX, NBY, and NBZ on DYPARAM,EULERCB must be such that  $NBX \cdot NBY \cdot NBZ$  is equal or a multiple of the number of processors used. For instance, if the number of processors in the cluster is 4,  $NBX \cdot NBY \cdot NBZ$  must be equal to either 4 or 8 or 12, etc. Otherwise, the calculation will terminate prematurely with an error message.

When using these options the Barrier and FLOW definitions only support geometric conditions like boundary face direction and a square definition

14. To determine a suitable size for the static output box, the simulation can be run first without the fields X0BX, Y0BX, Z0BX, DXBX, DYBX, and DZBX set. In the ,OUT file each summary of adaptive meshing gives the smallest box surrounding all adaptive elements so far. The last summary then yields the dimension of the static output box. To accommodate for elements that are not completely inside the box, the actual static output box is automatically extended a little. The actual dimensions are written in the out file after the first adaptive meshing summary. It is not needed to set the XREF, YREF, and ZREF option. If they are set the static output box will be compatible with the defined reference point.



**MESUPER****Defines External Superelement DMIG Input for SOL 600 Residual Analyses**

Superelement DMIG matrices are created by Nastran and used when Marc is spawned from Nastran, SOL 600 only.

**Format:**

1	2	3	4	5	6	7	8	9	10	
MESUPER	ID	Fname								

**Example:**

MESUPER	1	super1.pch								
MESUPER	2	super2.pch								

Describer	Meaning
ID	Superelement ID. (Integer; no Default)
Fname	Filename containing external superelement data from the creation run (using Case Control EXTSEOUT (ASMBULK, DMIGPCH,EXTID=ID) (Character; no Default). Left justify in field, see Remark 8.

**Remarks:**

1. Enter as many MESUPER lines as necessary to define all external superelements.
2. This entry can presently only be used with SOL 600,106, SOL 600,101, SOL 600,nlstatic, or SOL 600,sestatic. External Superelements are not presently available for other types of SOL 600 analyses such as nonlinear (or linear) transient dynamics, eigenvalue analysis or buckling.
3. Fname is limited to 56 characters.
4. Include entries with the same Fname must be specified as include files in the Nastran input file. The include specifications must appear at the end of the Bulk Data portion of the file.
5. External superelement creation runs should use the Case Control command:  
EXTSEOUT(ASMBULK,DMIGPCH,EXTID=N)  
where N is the external superelement ID number. All creation runs must have the same number of subcases and use the same subcase IDs.
6. The SOL 600 residual input file must have the same number of subcases and subcase numbers as the creation runs.
7. This entry acts like an element, in other words it is not controlled by a case control command. It is always active if entered.

8. The MESUPER entries should normally be coded in small fixed field format. If coded in small format free field, Fname is limited to 8 characters. If coded in large format free field, Fname is limited to 16 characters. The filename may be extended to a continuation line. All filenames should be entered in lower case. Nastran will convert to upper case, and the SOL 600 translator will reconvert to lower case. The creation runs should also use lower case for all external superelement punch filenames for case-sensitive computer systems.
9. If there are no elements in the residual (that is, all elements are in the external superelements, PARAM,MARCND99,-1 is required to output the displacements in the Marc .out file regardless of the specified Case Control request.
10. The ASMBULK option in the creation runs is required for SOL 600 when outr options are specified. It is recommended whether or not outr options are specified. All .asm files (created by the ASMBULK option) for all external superelements should be included in the Bulk Data before any punch files (produced using the DMIGPCH option). See the following input file examples.
11. At present, an OP2 with results datablocks only can be produced by a SOL 600 External Superelement residual execution. OP2 files which combine geometry and results datablocks cannot be produced. Other outr options such as xdb, f06 and punch are also not available for SOL 600 External Superelement residual runs.
12. For the External Superelement Initial run(s) to generate punch and asm files, no Case Control output requests should be made and the following DMAP should be inserted in the Executive Control in order to prevent extra information (which will cause an error) for being inserted into the punch file:

```
compile extout
alter 'sdr2' $
delete /iug1,,,, $
alter 'sdr2' (2) $
delete /igulo,,,, $
```

13. If data exists after ENDDATA, including any characters on the ENDDATA line, after the word ENDDATA in the creation run input, this data must be removed.

Typical File Setup for External Superelement Creation Run for SOL 600 (same as for other solution sequences)

```
SOL 101
compile extout
alter 'sdr2' $
delete /iug1,,,, $
alter 'sdr2' (2) $
delete /iug1o,,,, $
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO.
EXTSE2A
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
EXTSEOUT (ASMBULK,DMIGPCH,EXTID=100)
SPC = 100
BEGIN BULK
aset1,123456,840,thru,848
CORD2R,1001,1002,,,,,1.0
```

(rest of file same as any other Nastran run)

## Typical File Setup for External Superelement Residual Run for SOL 600

```
SOL 600,101 outr=op2
CEND
TITLE = 2 SUPERELEMENTS AND THE RESIDUAL -- TEST PROBLEM NO. EXTSE2R
SUBTITLE = 8 X 8 MESH OF QUAD4 ELEMENTS; GM-CMS PROJECT
param,mextsee,1
SPC = 100
LOAD = 1000
DISP = ALL
K2GG=KAAX
M2GG=MAAX
BEGIN BULK
param,marcnd99,-1
force, 1000, 844, , 0.1, 0., 0., 1.
SPC1 100 12346 840 848
$2345678 2345678 2345678
mesuper 100 extse2a.pch
mesuper 200 extse2b.pch
include 'OUTDIR:extse2a.asm'
include 'OUTDIR:extse2b.asm'
include 'OUTDIR:extse2a.pch'
include 'OUTDIR:extse2b.pch'
ENDDATA
```

## METADATA

## Metadata for the input section

Defines the entry metadata.

Format:

1	2	3	4	5	6	7	8	9	10
METADATA	META								
	METAINFOR								

Example:

METADATA	Rear_Fender							
	This is line 1 of metadata							
	This is line 2 of metadata							
	This is line 3 of metadata...							

Describer	Meaning
META	Character handle of up to 24 characters identifying the metadata.
METAINFO	A open-ended list of strings of 64 characters that represents the user information.

Remarks:

1. META can include any character from defined bulk character set
2. METAINFO can contain basic Nastran characters and special symbols.

## MFLUID Fluid Volume Properties

Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix.

Format:

1	2	3	4	5	6	7	8	9	10
MFLUID	SID	CID	ZFS	RHO	ELIST1	ELIST2	PLANE1	PLANE2	
	RMAX	FMEXACT							

Example:

MFLUID	3	2	15.73	1006.	3	4	S	N	
		100.							

Describer	Meaning
SID	Set identification number. (Integer > 0)
CID	Identification number of rectangular coordinate system used to specify the orientation of the free surface (normal to $X_3$ ) and of planes of symmetry, if any. (Integer $\geq 0$ or blank)
ZFS	Intercept of the free surface on the $X_3$ axis of the coordinate system referenced by CID. If $X_3$ of a grid > ZFS then there is no fluid. See Remark 3. (Real; Default means that the free surface is located at an infinitely large positive value of XFS.)
RHO	Density of the fluid (Real).
ELIST1	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on one side by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. See Remarks 3. and 5. (Integer $\geq 0$ )
ELIST2	Identification number of an ELIST entry that lists the IDs of two-dimensional elements that can be wetted on both sides by the fluid. Only those elements connected to at least one grid point below ZFS are wetted by the fluid. (Integer $\geq 0$ ; ELIST1 + ELIST2 > 0)
PLANE1, PLANE2	Planes of symmetry, antisymmetry, or no symmetry. "S" means that plane 1, which is the plane containing the $X_1$ and $X_3$ axes of CID, is a plane of symmetry. "A" means that plane 1 is a plane of antisymmetry. "N" means that it is neither. See Remark 5. Plane 2 uses "S", "A", or "N" for the $X_2$ and $X_3$ plane. (Character: "S", "A", or "N")
RMAX	Characteristic length. Interactions between elements with separation that is greater than RMAX will be neglected. (Real > 0.0; Default = 1.0E 10)
FMEXACT	Exact integration is used if the distance between two elements is less than FMEXACT times the square root of the area of the larger element. Otherwise, center point integration is used by default. (Real; Default = 1.0E 15)

## Remarks:

1. The MFLUID entry must be selected with the Case Control command MFLUID = SID.
2. Several MFLUID entries corresponding to different fluid volumes can be used simultaneously.
3. The wetted side of an element in ELIST1 is determined by the presence or absence of a minus sign preceding the element's ID on the ELIST entry. A minus sign indicates that the fluid is on the side opposite to the element's positive normal, as determined by applying the right-hand rule to the sequence of its corner points. The same element can appear on two ELIST entries, indicating that it forms a barrier between the unconnected fluids.
4. The fluid volume may be finite (interior) or infinite (exterior). The volume may be bounded by a free surface and one or two planes of structural symmetry. If structural symmetry is used, the structure must have the symmetric or antisymmetric boundary corresponding to the selection in fields 8 and 9. Interior fluids must have ELIST1 data and a free surface or plane of antisymmetry.
5. The planes of symmetry and/or antisymmetry defined in fields 8 and 9 must be planes of symmetry for the entire analysis. The user may apply appropriate structural boundary conditions at all grid points lying in these planes.
6. The current list of elements that may be placed in ELIST1 and ELIST2 include CTRIA3, CQUAD4, CTRIAR, and CQUADR.
7. The continuation entry is optional.
8. If there is ELIST1 data and no free surface nor plane of antisymmetry, the program assumes a special form of external fluid. These special external fluids must have a CID (field 3) such that the origin of the fluid coordinate system is near the center of the enclosed volume, since the singularity for volume change will be placed at the origin. Special external fluids are supported only in SOLs 103 and 107 through 112. If used in conventional solution sequences, System Fatal Message 3001 results for file 205.
9. See PARAM,VMOPT in [Parameters](#). VMOPT controls when the virtual mass is included in the mass matrix.
10. If any MFLUID entry is changed or added on restart then a complete re-analysis may be performed. Therefore, MFLUID entry changes or additions are not recommended on restart.
11. A tolerance is computed for each wetted element, with the value of  $0.01 \sqrt{2A}$ , where A is the area of the element. If any grid point connected to the element lies within TOL below the free surface it is moved to the free surface.
12. Any element that has all grids on or above the free surface, after the grid points are moved by the procedures given in Remark 11, is removed from the ELIST. It is not included in the VM effects, and will produce no pressure output.

## MGRSPR Defines Grids to Add Soft Spring to Ground - SOL 600

This entry is used to add soft springs to ground at selected grids to stabilize the structure in a nonlinear analysis. It is most often used with contact to stabilize free-free bodies before they come into contact. The spring rates selected should be stiff enough to allow matrix decomposition but weak enough so they are not significant once full contact is achieved. Values on the order of 1.0E-6 to 1.0E-4 times the average main diagonal terms in the stiffness matrix for the grids selected are recommended.

Format:

1	2	3	4	5	6	7	8	9	10
MGRSPR	ID1	THRU	ID2	IDIR	K				

Example:

MGRSPR	100	THRU	200	123	10.0				
	500	THRU	520	123456	1.0				

Describer	Meaning
ID1	Starting grid ID. (Integer; Required; no Default)
THRU	Enter the character string THRU if more than one grid is desired.
ID2	Ending grid ID of the range. (Integer; or blank. If blank; ID2=ID1)
IDIR	Directions that the spring(s) will be added in, any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required)
K	Spring rate. (Real; Required)

Remarks:

1. If springs are to be added to all grids, PARAM,MRSRING may be used instead of this entry.
2. Do not use PARAM,MRSRING and this entry in the same model.
3. Grids that do not exist in the range ID1 to ID2 will automatically not have springs in the Marc input.

**MINSTAT**                      Option to Define Initial State Variables for SOL 600

This option is used to enter initial (stress free) temperatures calculated from a previous heat transfer analysis and saved on a t16 or t19 file. (MCHSTAT is used to define the temperatures that cause thermal strains). This entry may also be used to initialize other state variables if required.

Format:

1	2	3	4	5	6	7	8	9	10
MINSTAT	IDV	IOPT		INCR		IFORM	IPRT	NPST	
	"FILE"	Name							
	"ELEM"	ELE1	ELE2	INT1	INT2	LAY1	LAY2	VAL	
	"STATE"	NS	IS1	IS2	IS3	IS4	IS5	IS6	
		IS7	IS8	IS9	etc.				

Example:

MINSTAT	1	3		1		0		9	
	FILE	initial_state_example							

Descriptor	Meaning
IDV	State variable identifier (1=temperature). (Integer; Default = 1) (2,1) If more than one state variable is required, enter -1.
IOPT	Option of how to enter the data. (Integer; Default = 3) (2,2) 1        Use the "ELEM" continuation line for as many elements as necessary 2        Enter the data using user subroutine INITSV 3        Read the data from a t16 or t19 file (see IFORM)
INCR	Increment number on t16 or t19 file defining the new state values if IOPT=3. (Integer; no Default) (2,5)
IFORM	Designates whether a binary (t16) or formatted (t19) post file is used if IOPT=3. (Integer; Default = 0) (2,7) 0        Use binary (t16) file 1        Use formatted (t19) file
IPRT	Enter a value of 1 to suppress printing of state variable values defined in user subroutine INITSV (only applicable if IOPT=2). (Integer; Default = 0) (2,8)
NPST	Post Code ID to be read into this state variable. (Integer; Default = 9 [temperature]) (2,9) See MARCOUT for a list of the post codes.
FILE	Enter the character string FILE if IOPT=3. (Character; no Default; Required if IOPT=3)



Descriptor	Meaning
NAME	Enter the file name without the extension (.marc.t16 or .marc.t19). (Character; no Default) The file name must be entirely in lower case for case-sensitive computer systems and is limited to 56 characters. This file must be in the same directory as the Nastran input file.
ELEM	Enter the character string ELEM to start a list of elements and associated values if IOPT=1. (Character)
ELE1	First element with value VAL. (Integer; no Default; Required) (3,1)
ELE2	Last element with value VAL. (Integer; Default = ELE1) (3,2)
INT1	First integration point with value VAL. (Integer; no Default; Required) (3,3)
INT2	Last integration point with value VAL. (Integer; Default = INT1) (3,4)
LAY1	First cross-section layer with value VAL. (Integer; no Default; Required) (3,5)
LAY2	Last cross-section layer with value VAL. (Integer; Default = LAY1) (3,6)
VAL	New state value for these elements. (Real; no Default; Required) (4,1)
STATE	Enter the character string STATE to start a list of state variables. (Character)
NS	Number of state variables to be defined. (Integer; no Default; limited to 16 maximum)
ISi	State variable post codes. (Integer; no Default) (9,i) See MARCOUT entry for applicable post codes.

#### Remarks:

1. Only one MINSTAT entry may be entered in the input. If additional entries are found, the first will be used.
2. This entry maps to Marc's INITIAL STATE entry
3. (i,j) refer to Marc's INITIAL STATE (data block, field)
4. MINSTAT (and/or MCHSTAT) cannot be the only applied "loads". At least one standard load such as FORCE, PLOAD4 or a standard TEMP entry must be entered with a LOAD Case Control command that references the standard load(s). If there are no standard loads, please enter a dummy load with a very small magnitude and a LOAD Case Control command to reference it.

## MISLAND

## SOL 600

Defines an island of connected elements that will be completely removed if the number of elements within the island becomes smaller than a specified value in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MISLAND	NELEM								

Example:

MISLAND	20								
---------	----	--	--	--	--	--	--	--	--

Describer	Meaning
NELEM	Number of elements to use as a limit for element deactivation. If the number of elements in an island is less than or equal to this number, all elements in this island are deactivated. (Integer; no Default; Required field)

Remarks:

1. If this entry is made in the model, it subdivides the entire mesh into islands of connected regions. If the number of elements in a particular island is less than or equal to the specified value, all elements of this island are deactivated. This check is performed after element deactivation has taken place where the deactivation can be due to model input options, deactivation user subroutines, or through material damage or failure. Two elements are considered connected if they share a node for line elements, an edge for 2-D elements, or a face for 3-D solid elements. This option is useful for cases where unconnected elements or regions of elements might exist after the neighboring elements have been deactivated. There is no check performed to see if the island to be deactivated has enough boundary conditions. Only the number of elements in the island is used for determining if the elements should be deactivated.
2. Only one MISLAND entry should be placed in the input. If more than one is entered, the first will be used.

**MIXTURE****Constituents of "Composite" Material on Original and Potentially Damaged State - SOL 600**

Defines constituents of "composite" material on original and potentially damaged state. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MIXTURE	MID	ITYPE	IALPH	ISET	M1	FRACT1	M2	FRACT2	
	M3	FRACT3	M4	FRACT4	M5	FRACT5	etc.		

Example:

MIXTURE	120	3	1	200	10	.2	20	.2	
	30	.2	40	.2	50	.2			

Describer	Meaning
MID	Material ID (Integer; no Default, required field)
ITYPE	Mixture rule (Integer; Default = 1) 1 Weighted average of material properties based upon volume fraction. 2 Weighted average of Hooke's law based upon volume fraction 3 Weighted average of nonlinear stress strain curve based on volume fraction
IALPH	Flag controlling averaging procedure for thermal expansion (Integer; Default = 1) 1 Weighted average based upon volume fraction 2 Not currently available
ISET	ID of a SET3 containing the all the elements making up this material (Integer; no Default; Required)
Mi	Identification of a previously defined material, defined on a MATxxx entry (for example the MID on a MAT1 entry) (Integer; no Default)
FRACTi	Fraction for the ith component (see ITYPE). The sum of all FRACTi must add to 1.0 (Real; no Default)

Remarks:

1. Enter as many Mi, FRACTi components as desired. Each may have different properties, and failure criteria.
2. Fields 2-4 of each continuation entry are required, fields 5-9 may be left blank.

3. This option allows one to create a new material comprised of a number of other materials. The material behavior is based upon a "mixture" of the individual components using a mixture rule. Several of these mixture rules are only appropriate for linear elastic materials but also allow for temperature dependent material properties. The most sophisticated model (ITYPE=3) allows for the mixture of materials which undergo elastic-plastic behavior.
4. If void ratio or porosity is defined, it applied to all components in a uniform manner.
5. ITYPE 1 and 2 only support linear elastic material.
6. ITYPE 3 is not available using updated Lagrange - use PARAM,MARCPLAS,1 or PARAM,MARCPLAS,2 if any of the component materials have plasticity. The mixture will be limited to small strains, but large displacement, large rotation and follower forces and follower pressures can be included.
7. ITYPE 3 may not include the following material laws in any of the components.
  - Thermo-pore
  - Gurson damage
  - Simplified damage models 9 and 10
  - Gasket material
  - Shape memory material
  - Soils
  - User defined generalized stress strain
  - ORNL
  - Rigid-Plastic
  - Grain size effects
  - Rubber material (mooney, ogden, foam, gent, arruda-boyce)
  - Cohesive
8. Rebar elements and shell elements with offsets or non-composites with different properties for bending vs membrane (which use the Marc PSHELL option) may not be used for mixtures.
9. Within a layer, if the components are orthotropic or anisotropic, the preferred directions must be aligned.
10. The material identification number must be unique for all COHESIV, MAT1, MAT2, MAT3, MAT8, MATDIGI, MATG, MATHE, MATHP, MATNLE, MATORT, MATPE1, MATSMA, MATUSR, MCOHE, MIXTURE, and MATD\* entries.

## MKAERO1

## Mach Number - Frequency Table

Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO1	m1	m2	m3	m4	m5	m6	m7	m8	
	k1	k2	k3	k4	k5	k6	k7	k8	

Example:

MKAERO1	.1	.7							
	.3	.6	1.0						

Describer	Meaning
mi	List of from 1 to 8 Mach numbers. (Real $\geq$ 0.0). See remark 8.
kj	List of from 1 to 8 reduced frequencies. (Real > 0.0)

Remarks:

- Blank fields end the list, and thus cannot be used for 0.0.
- All combinations of (mi, kj) will be used.
- The continuation entry is required.
- Multiple MKAERO1 entries are permitted.
- For the lifting surface theories (Doublet-Lattice and Mach Box), the maximum value of kj should be less than one quarter of the number of boxes on a representative chord (i.e.,  $\text{MAX}(kj) < \bar{C}/4\Delta x$  where  $\bar{C}$  is the reference chord and  $\Delta x$  is a typical box chord length).
- In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
- The very low nonzero value of kj required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.
- mi = 1.0 is not supported except when used in conjunction with CAERO4 entry (Strip Theory).

## MKAERO2

### Mach Number - Frequency Table

Provides a list of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.

Format:

1	2	3	4	5	6	7	8	9	10
MKAERO2	m1	k1	m2	k2	m3	k3	m4	k4	

Example:

MKAERO2	.10	.30	.10	.60	.70	.30	.70	1.0	
---------	-----	-----	-----	-----	-----	-----	-----	-----	--

Describer	Meaning
mi	Mach numbers. (Real $\geq$ 0.0). See remark 5.
ki	Reduced frequencies. (Real > 0.0)

Remarks:

1. MKAERO2 will cause the aerodynamic matrices to be computed for the given sets of parameter pairs. Embedded blank pairs are skipped.
2. No continuation entries are allowed, but multiple MKAERO2 entries are permitted.
3. In SOL 146, the program selects only one value of mi to use in computing the dynamic response solution and, by default, the lowest value is selected. The PARAM,MACH,m entry may be specified to select a different value. If PARAM,MACH,m is specified, then the value of mi closest to m will be selected.
4. The very low nonzero value of ki required for aeroelastic divergence analysis of restrained vehicles with the K- and KE-methods of flutter analysis must be included on this entry.
5. mi = 1.0 is not supported except when used in conjunction with CAERO4 entry (Strip Theory).

## MLAYOUT

## SOL 600

Selects layered composite shell output to be placed in Marc's t16 and/or t19 files and (if requested) to be transferred from Marc to the Nastran Database. The MARCOUT entry with LAYCODE of 3 or 103 must be used in conjunction with this entry. Used in SOL 600 only.

### Format:

1	2	3	4	5	6	7	8	9	10
MLAYOUT	L1	THRU	L2	BY	L3				
	L4	THRU	L5	BY	L6				

### Example:

MLAYOUT	2	THRU	5						
	11	THRU	51	BY	10				
	52	THRU	54						
	21	THRU	20	BY	10				

Descriptor	Meaning
L1, L4	Starting layer of a sequence. (Integer; Required; no Default)
L2, L5	Ending layer of a sequence. (Integer; Default = L1 or L4)
L3, L6	Increment value for the sequence. (Integer; Default = 1)

### Remarks:

1. This entry must be used in conjunction with MARCOUT with LAYCODE=3 or 103.
2. L2 must be larger than L1.
3. L1, L4, etc. must be positive integers of 1 or larger.
4. L2, L5 should not exceed the largest number of layers in the model.
5. Selection of all layers can lead to extremely large output files.
6. In the example, L1 could also be 1 instead of 2 and the largest layer could be 55 instead of 54 with causing an error.
7. If the layers designated on this entry are to apply to selected elements, enter the elements on the T16SEL entry. It is not currently possible to process different layered output for multiple groups of elements (i.e., it is not currently possible to request output of layers 1-20 for some elements and 1-10 for others.)

## MNF600

## Defines Auxiliary Data for MSC Adams MNF Files - SOL 600

Generated by the Marc portion of a SOL 600 execution. Used in SOL 600 only. (See also the MDMIOU Bulk Data entry.)

Format:

1	2	3	4	5	6	7	8	9	10
MNF600	ID	ISTRESS	ISTRAIN	ISHELL	MASSU	LENGU	TIMEU	FORCU	

Example:

MNF600	100	1	1	3	2	7	3	2	
--------	-----	---	---	---	---	---	---	---	--

Describer	Meaning
ID	Subcase for which the reduced matrices will be output. ID must correlate to a SUBCASE Case Control ID, for example, if the case control contains SUBCASE 20, ID would be 20. Must match the MDMIOU entry. At present, ID is ignored and the first entry will apply to all subcases. (Integer; Default = 1)
ISTESS (2)	Flag to compute stress and place in the MNF file (0=no stress, 1=stress) (Integer; Default = 0)
ISTRAIN (3)	Flag to compute strain and place in the MNF file (0=no strain, 1=strain) (Integer; Default = 0)
ISHELL (4)	For shell elements, this entry describes which location the stresses or strains will be output to the MNF file. 1=top, 2=center, 3=bottom (Integer; Default = 1)
MASSU [2,1]	Mass units for MNF file. (Integer; Default = 1: kilogram) The following possible values may be entered: 1:kilogram (Default) 2:pound mass 3:slug 4:gram 5:ounce mass 6:kpound mass 7:megagram 8:dozen slug



Describer	Meaning
LENGU	Length units for MNF file (Integer; Default = 2: meter) The following possible values may be entered: 1:kilometer 2:meter (Default) 3:centimeter 4:millimeter 5:mile 6:foot 7:inch
TIMEU [2,3]	Time units for MNF file (Integer; Default = 3: second). The following possible values may be entered: 1:hour 2:minute 3:second (Default) 4:millisecond
FORCEU [2,4]	Force units for MNF file (Integer; Default = 1: newton). The following possible values may be entered: 1:Newton (Default) 2:pound force 3:kilogram force 4:ounce force 5:dyne 6:kNewton 7:kpound force

#### Remarks:

1. The MDMIOUT entry is the primary entry which generates an MNF file. This entry, MNF600 is only necessary if one or more of the fields is required to define non-default values, for example to generate stresses or strains or to specify the units.
2. The ID must be the same as the MDMIOUT ID.
3. Only one MNFDAT entry is allowed in an input file. If more than one is entered, the first will be used.
4. All remarks concerning MNF files for the MDMIOUT entry are also applicable to this entry.
5. (i) Indicates the corresponding field of Marc's MNF Parameter.
6. [i,j] Indicates the corresponding datablock and field of Marc's MNF units entry.

## MODTRAK

## Mode Tracking Parameters

Specifies parameters for mode tracking in design optimization (SOL 200).

Format:

1	2	3	4	5	6	7	8	9	10
MODTRAK	SID	LOWRNG	HIGHRNG	MTFILTER					

Example:

MODTRAK	100	1	26	0.80					
---------	-----	---	----	------	--	--	--	--	--

Describer	Meaning
SID	Sets identification number that is selected in the Case Control Section with the MODTRAK command. See Remark 1. (Integer; no Default)
LOWRNG	Lowest mode number in range to search. See Remark 2. (Integer $\geq 0$ ; Default = 0. If nonzero, LOWRNG < HIGHRNG.)
HIGHRNG	Highest mode number in range to search. See Remark 2. (Integer > 0; Default = number of eigenvalues extracted. If nonzero, LOWRNG < HIGHRNG.)
MTFILTER	Filtering parameter used in mode cross-orthogonality check. See Remark 3. (Real; Default = 0.9)

Remarks:

1. Only the designed modes for the subcase will be tracked. A designed mode is one that is used in the design model (in connection with either objective or constraints) and, therefore, identified on a DRESP1 entry.
2. The range of modes LOWRNG through HIGHRNG, inclusive, will be used to track the designed modes. If LOWRNG and HIGHRNG are both blank, then all computed modes will be used to search for the designed modes. Since large numbers of computed modes will result in higher computational costs, limiting the search range with LOWRNG and HIGHRNG is recommended.
3. Modes are considered to correlate if their mass normalized cross orthogonalities are greater than MTFILTER.

## MOMAX Conical Shell Static Moment

Defines a static concentrated moment load on a ring of a conical shell.

Format:

1	2	3	4	5	6	7	8	9	10
MOMAX	SID	RID	HID	S	MR	MP	MZ		

Example:

MOMAX	1	2	3	1.0	0.1	0.2	0.3		
-------	---	---	---	-----	-----	-----	-----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
RID	Ring identification number. See the RINGAX entry. (Integer > 0)
HID	Harmonic identification number or a sequence of harmonics. See Remark 5. (Integer $\geq 0$ or Character)
S	Scale factor. (Real)
MR, MP, MZ	Moment components in the r, $\phi$ , z directions. (Real)

Remarks:

1. MOMAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD=SID.
3. A separate entry is needed for the definition of the moment associated with each harmonic.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.
5. If a sequence of harmonics is to be placed in HID, the form is as follows: "Sn1Tn2", where n1 is the start of the sequence and n2 is the end of the sequence; i.e., for harmonics 0 through 10, the field would contain "S0T10".

**MOMENT**

## Static Moment

Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT	SID	G	CID	M	N1	N2	N3		

Example:

MOMENT	2	5	6	2.9	0.0	1.0	0.0		
--------	---	---	---	-----	-----	-----	-----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
CID	Coordinate system identification number. (Integer ≥ 0 or blank)
M	Scale factor. (Real)
Ni	Components of the vector measured in the coordinate system defined by CID. (Real; at least one Ni ≠ 0.0 unless M is zero)

Remarks:

1. The static moment applied to grid point G is given by

$$\vec{m} = M\vec{N}$$

where  $\vec{N}$  is the vector defined by (N1, N2, N3). The magnitude of  $\vec{m}$  is equal to M times the magnitude of  $\vec{N}$ .

2. In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. A CID of zero or blank references the basic coordinate system.
4. For scalar points see SLOAD.
5. For TYPE=12 or TYPE=13 on the TLOAD1, G is the ID of a rigid body; the MID of a rigid material (MATRIG) or the EID of a RBE2D. The MID of a rigid material and the EID of RBE2 must be different when both of a RBE2D and a rigid material are used with these TYPEs. SOL 700 only.

## MOMENT1

## Follower Moment, Alternate Form 1

Defines a concentrated moment at a grid point by specifying a magnitude and two grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT1	SID	G	M	G1	G2				

Example:

MOMENT1	6	13	-2.93	16	13				
---------	---	----	-------	----	----	--	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
G1, G2	Grid point identification numbers used to define the unit vector $\vec{n}$ . (Integer > 0; G1 and G2 cannot be coincident.)

Remarks:

- The concentrated moment applied to grid point G is given by

$$\vec{m} = M\vec{n}$$

where  $\vec{n}$  is a unit vector parallel to a vector from G1 to G2.

- In the static solution sequences, SID must be selected by the LOAD Case Control command. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 835](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400, if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

## MOMENT2

### Follower Moment, Alternate Form 2

Defines a concentrated moment at a grid point by specification of a magnitude and four grid points that determine the direction.

Format:

1	2	3	4	5	6	7	8	9	10
MOMENT2	SID	G	M	G1	G2	G3	G4		

Example:

MOMENT2	6	13	-2.93	16	13	17	13		
---------	---	----	-------	----	----	----	----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number at which the moment is applied. (Integer > 0)
M	Magnitude of moment. (Real)
Gi	Grid point identification numbers used to determine the unit vector $\vec{n}$ . (Integer > 0; G1 and G2 cannot be coincident; G3 and G4 cannot be coincident.)

Remarks:

1. The concentrated moment applied to grid point G is given by

$$\vec{m} = M\vec{n}$$

where  $\vec{n}$  is the unit vector parallel to the cross product of the vectors from G1 to G2, and G3 to G4.

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 835](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, and 400 if geometric nonlinear effects are turned on with PARAM,LGDISP,1. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

## MONCARL

### Parameters for Monte-Carlo simulation

Parameters for conducting Monte-Carlo simulation using the non-parametric variability method (NPVM).

Format:

1	2	3	4	5	6	7	8	9	10
MONCARL	SID	VARTYP							
	STRVAR	MSVAR	KSVAR	BSVAR					
	FLUVAR	MFVAR	KFVAR	BFVAR					

Example:

MONCARL	100	DELTAVAR							
	STRVAR	0.001	0.05	0.002					
	FLUVAR	0.0	0.001	0.0					

Describer	Meaning
SID	Set identification number referenced by MONCARL case control. (Integer > 0; Required)
VARTYP	Select the type of variabilities that follow (Character; Default = ALPHAVAR)  ALPHAVAR Variabilities in this entry are Alpha variabilities.  DELTAVAR Variabilities in this entry are Delta variabilities.
“STRVAR”	Indicates Structural variability parameters follow. (String)
MSVAR	Structural mass variability. (Real >= 0.0; Default = 0.0)
KSVAR	Structural stiffness variability. (Real >= 0.0; Default = 0.0)
BSVAR	Structural damping variability. (Real >= 0.0; Default = 0.0)
“FLUVAR”	Indicates Fluid variability parameters follow (String)
MFVAR	Fluid mass variability. (Real >= 0.0; Default = 0.0)
KFVAR	Fluid stiffness variability. (Real >= 0.0; Default = 0.0)
BFVAR	Fluid damping variability. (Real >= 0.0; Default = 0.0)

#### Remarks

1. Please use below as the guideline for selecting above mentioned structural and fluid alpha variabilities (which should be > 0.0):
  - a. Low variability level = 0.001
  - b. Medium variability level = 0.01

- c. High variability level = 0.05
2. Alpha variabilities are related to delta variabilities through the order of the matrix to be randomized,  $n$  (or equivalently the number of modes),

$$a = \delta \sqrt{\frac{2}{n+1}}$$

3. There is a limitation in the admissible values for the variability level. The maximum value for delta variability equals,

$$\delta_{max} = \sqrt{\frac{n+1}{n+5}}$$

On a model with a large amount of modes ( $n \gg 1$ ), the maximum value of delta tends to 1 ( $\delta_{max} \approx 1$ ).

The maximum value of alpha variability using the above equations equals,  $a_{max} = \sqrt{\frac{2}{n+5}}$

4. An unphysical (larger than permissible or negative value) selection of a variability would result in a FATAL.



## MONCNCM

## Nondimensional Normal Force and Pitching Moment

Provides a stripwise aerodynamic normal force and pitching moment coefficients for CAERO1 based aerodynamics.

Format:

1	2	3	4	5	6	7	8	9	10	
MONCNCM	NAME	LABEL								
	MREF	CAERID1	CAERID2	...	CAERIDn					

Example:

MONCNCM	LEFT	Normal force and pitching moment values for strips on the left wing.							
		2001	3001						

Describer	Meaning
NAME	Unique character string of up to 8 characters identifying the family of chord-wise strips. (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the component. (Optional)
MREF	Fractional chord location of the aerodynamic strip about which the pitching moment is to be calculated. (Real; $0.0 \leq MREF \leq 1.0$ ; Default = 0.25)
CAERID1	ID of a CAERO1 entry that contains aero boxes for which strip results are to be produced. (Integer > 0 or "ALL"). See Remarks 2. and 3.

Remarks:

1. The LABEL is optional.
2. Output is produced for all chordwise strips on the referenced CAERO1 entries. If a strip spans CAERO1 panels, results are output for the total strip.
3. If CAERID1 is the character string "ALL", then output will be produced for all strips.
4. Strips are identified based on the y and z coordinates in the aerodynamic coordinate system.
5. If strips from separate CAERO1's have the same y and z coordinate and the same strip width and share an xlocation (i.e., the leading edge of one strip equals the trailing edge from another) then they are processed as a single strip.
6. The normal force component is normalized by the dynamic pressure times the surface area of the strip. The moment component is normalized by the dynamic pressure times the surface area of the strip times the chord length at the center of the strip. The moment is calculated about the MREF location of the strip with the quarter-chord of the strip as the default.
7. Only CAERO1 id's are supported.

## MONDSP1

## Displacement Monitor Point

Defines a virtual point displacement response at a user-defined reference location (coordinates and coordinates system) as a weighted average of the motions at a set of grid points.

Format:

1	2	3	4	5	6	7	8	9	10	
MONDSP1	NAME	LABEL								
	AXES	COMP	CP	X	Y	Z	CD	INDDOF		

Example:

MONDSP1	WING195	Wing twist at station 150.							
	5	WING150	1001	120	150.0	17.0	1002		

Describer	Meaning
NAME	Character string of up to 8 characters identifying the monitor point (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer $\geq 0$ ; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system at which the displacement is to be monitored.
CD	The identification number of a coordinate system in which the resulting displacement components are output. (Integer $\geq 0$ ; Default = the coordinate system specified by the CP field)
INDDOF	Component numbers of all the independent grids from which the derived, dependent, monitor DOF's are to be computed. (Any unique combination of the integers 1 through 6 with no embedded blanks.) See Remark 3. (Default = 123)

Remarks:

1. The MONDSP1 is available for SOLs 101, 103, 105, 108, 109, 111, 112, 144, 146 and 200.
2. The entry can create either an aerodynamic or a structural MONDSP1, depending on the data provided in the COMP field. A structural MONDSP1 can have the same name as an aerodynamic MONDSP1, but two structural or two aerodynamic MONDSP1s cannot have the same name.

3. The INDDOF field defines the Ci field on the virtual RBE3; that is, it defines the components of the grids on the AECOMP that will be sampled to define the dependent (monitor point) displacement. Typically, the default is the correct choice. However, if there is only a single grid point, all six DOF's can be used.

## MONGRP

## Specification of a Monitor Group

Defines a collection of monitor points into a group available for postprocessing.

Format:

1	2	3	4	5	6	7	8	9	10
MONGRP	GNAME	LABEL							
	NAME1	CLASS1	REAL1	INT1	STRING1				
	NAME2	CLASS2	REAL2	INT2	STRING2				
	NAME(m)	CLASS(m)	REAL(m)	INT(m)	STRING(m)				

Example:

MONGRP	WING	A COLLECTION OF MONITOR POINTS ON THE WING							
	WING195	SMONPT1		1	WL195				
	WING205	SMONPT1		2	WL205				
	WINGDSP	SMONDP1	20.0		GRID20				

Descriptor	Meaning
GNAME	A character string of up to 8 characters identifying the monitor group. (Character)
LABEL	A String comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point.
NAME(i)	Name of an existing monitor point. (Character)
CLASS(i)	Type of monitor point (either AMONPT1, AMONDP1, SMONPT1, SMONDP1, MONPT2, MONPT3)
REAL(i)	Optional real value to designate a real property of the monitor point.
INT(i)	Optional integer value to designate a integer property of the monitor point.
STRING	A label of the current monitor type. (Character string of no more than 32 character)

Remarks:

1. The MONGRP entry provides the user a means of specifying groups of monitor point data but does not produce any output on its own.
2. Aerodynamic (AMONPT1 and AMONDP1) and structural (SMONPT1, SMONDP1, MONPT2, MONPT3) cannot be mixed in a given group.
3. The REAL(i), INT(i) and STRING(i) data are provided to facilitate user display of the monitor data and can be used, for example, to label xy plots.
4. The LABEL should be unique across MONGRP entries.

## MONPNT1 Integrated Load Monitor Point

Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system. The integrated loads about this point over the associated nodes will be computed and printed for statics, dynamics and static aeroelastic trim analyses and from integrated loads on the nonlinear static aeroelastic database.

Format:

1	2	3	4	5	6	7	8	9	10	
MONPNT1	NAME	LABEL								
	AXES	COMP	CP	X	Y	Z	CD			

Example:

MONPNT1	WING155	Wing Integrated Load to Butline 155							
	34	WING		0.0	155.0	15.0			

Describer	Meaning
NAME	Character string of up to 8 characters identifying the monitor point (Character; no Default)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the monitor point. (Character; optional)
AXES	Component axes to monitor. (Any unique combination of the integers 1 through 6 with no embedded blanks.)
COMP	The name of an AECOMP or AECOMPL entry that defines the set of grid points over which the monitor point is defined. (Character; no Default)
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer ≥ 0; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer ≥ 0; Default = the coordinate system specified by the CP field)

Remarks:

1. The MONPNT1 is available for SOLs 101, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The entry can create either an aerodynamic or a structural MONPNT1, depending on the data provided in the COMP field. A structural MONPNT1 can have the same name as an aerodynamic MONPNT1, but two structural or two aerodynamic MONPNT1s cannot have the same name.

3. In SOL 144, static aeroelasticity, the structural MONPNT1 integration is modified to account for the redistribution of loads and inertia caused by MPCs and rigid elements. In all other supported solution sequences, the integration is done using the g-set loads on all grid points specified on the referenced SET1 entry.

## MONPNT2

## Internal Load Monitor Point

Element Monitor Output Results Item.

Format:

1	2	3	4	5	6	7	8	9	10	
MONPNT2	NAME	LABEL								
	TABLE	TYPE	NDDLitem	EID						

Example:

MONPNT2	SB100	Leading edge stringer at root							
	STRESS	CBAR	SX2A	100					

Describer	Meaning
NAME	Unique character string of up to 8 characters identifying the monitor point (Character; no Default)
LABEL	An optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point. (Character; Default = Blank)
TABLE	Type of output to be monitored. Options are STRESS, FORCE or STRAIN. (Character; no Default)
TYPE	Element type (Character; no Default)
NDDLitem	Component for this type to be monitored. This is the NDDL label for the particular Table and element type. (Character; no Default)
EID	Element ID. (Integer > 0)

Remarks:

1. The MONPNT2 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146 and 200.
2. Most element types have some items that can be monitored.
3. An assumption is made that the desired component is linear with respect to the displacement vector. If this assumption is not valid, the results will be incorrect.
4. Fictitious Table/Type/NDDLitems/EID generate a warning message and are ignored.
5. NDDL descriptions for Table=FORCE can be found in the [MSC Nastran DMAP Programmer's Guide](#) within the OEF datablock description. Table=STRESS and STRAIN are contained in the OES datablock description.

Once within the datablock description you can search for the element name (or better yet, element number, see the following table) you are interested in. There can be several different descriptions for an element type. For example, real vs. complex, thermal, stress vs. strain (within the OES description), linear vs nonlinear. In addition, the shell class of elements (quad4, quad8, quadr, tria3, tria6, triar) will have a composite form (quad4lc, quud8lc, quadrlc, tria3lc, .., i.e., basename + “LC”), corner or bilin stresses (basename + “C”).

By looking at the comments contained in the text make sure you are reading from the appropriate section. The NDDLitem is labeled as the ‘NAME’ field within the [MSC Nastran DMAP Programmer’s Guide](#).

You can also print the NDDL description for the entire database by running the following 4 statement bulk data file.

```
sol loadnddl  
compile nddl=nddl,list  
cend  
enddata
```



## MONPNT3

## Integrated Load Monitor Point

Sums select Grid Point Forces to a user chosen monitor point.

Format:

1	2	3	4	5	6	7	8	9	10	
MONPNT3	NAME	LABEL								
	AXES	GRIDSET	ELEMSET	CP	X	Y	Z	XFLAG		
	CD									

Example:

MONPNT3	t0	Fuselage station 1105							
	123456	1	2	0	30.0			ASM	

Describer	Meaning
NAME	Unique character string of up to 8 characters identifying the monitor point (Character, Required)
LABEL	A optional string comprising of no more than 56 characters (fields 3 through 9) that identifies the monitor point.
AXES	Component axes about which to sum. Any unique combination of the integers 1 through 6 with no embedded blanks. (Integer; Required)
GRIDSET	Refers to a SET1 entry that has a list of grids to be included in the monitored point. (Integer; Required)
ELEMSET	Refers to a SET1 entry that has a list of elements to include at the monitored point. (Integer; optional)
CP	The identification number of a coordinate system in which the (x,y,z) coordinates are defined. (Integer ≥ 0; Default=0)
X,Y,Z	The coordinates in the CP coordinate system about which the forces are to be summed. (Real; Default = 0.0).
XFLAG	Exclusion flag. Exclude the indicated Grid Point Force types from summation at the monitor point. Default = blank (no type excluded). See Remark 4.
	S            SPCforces
	M            MPC forces
	A, L, or P    applied loads
	D            dmig's (and any other type not described above) at the monitored point.
	C            contact forces (SOL 400 only).
CD	The identification number of a coordinate system in which the results are output. (Integer>=0, Default = the coordinate system specified by the CP field.

**Remarks:**

1. The MONPNT3 is available for SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN). Note that MONPNT3 output to csv file is available only to SOL 144.
2. If ELEMSET is blank, no contributions are made from the set of elements attached to the grid.
3. Fictitious grids or elements do not produce error or warning messages.
4. For linear solution sequences, if the exclusion flags omit some grid point force types but not all of them (i.e; if the field is not blank and is not SMAD) then the following limitations exist:
  - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
  - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
  - It is only available in SOLs 101, 103, 144 and SOL 200 (ANALYSIS = STATICS, MODES, or SAERO).
5. MONPNT3 can be useful in calculating shear, moment and torque from the internal loads in a structure. For example, if a split is made in a fuselage component and all the grids that reside on this split are placed in the GRIDSET, differing internal loads resultants can be obtained based on the ELEMSET and XFLAG values. If a cut is made in a structure, there are three types of loads:
  - A - loads that come from the elements upstream of the cut.
  - B - loads that are applied to the grids on the cut from any other source.
  - C - loads that come from the elements downstream of the cut.Where  $A + B + C = 0$   
Useful options for these two values are:
  - a. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is blank then the internal load is calculated using the elements that connect to the GRIDSET that are NOT included in ELEMSET. The direction of this load is reversed. So actually -C will be calculated which matches A+B. This in effect gives the resultants on the downstream/inboard side of the split pointing into upstream/ outboard direction which will include any loads applied to the GRIDSET from any source.
  - b. If the ELEMSET includes all the elements that connect to the GRIDSET that are on the upstream/outboard part of the split and the XFLAG value is SMAD then the internal load is calculated using the elements listed in ELEMSET. In that case, A will be calculated. This in effect gives the resultants on the upstream/outboard side of the split and does not include loads applied to the GRIDSET from any other source.
6. For SOL 400 (ANALYSIS = NLSTAT or NLTRAN), the following limitations exist:
  - The results for that MONPNT3 will not be exported to a CSV file, for SOL 144 only.
  - That MONPNT3 may not be used on a MONSUM, MONSUM1 or MONSUMT entry.
  - Thermal loads are not supported.

7. For linear solution sequences with XFLAG= blank or SMAD, the program applies a mini-EMA (element matrix assembly) algorithm that bypasses GPFDR (grid point force data recovery) and another modules to improve performance. This algorithm forms a stiffness matrix that just includes those elements specified in MONPNT3 entries. The monitor point results are computed directly from multiplying this matrix by an integration matrix that transfers forces to the monitor points. These results may be slightly different from the GPFDR results, such as clean zero forces versus small residual forces.

## MONSUM

## Linear Combinations of Monitor Point Components

Defines a new monitor result that is the weighted sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (see Remark 5.)

Format:

1	2	3	4	5	6	7	8	9	10
MONSUM	NAME	LABEL							
	NEWAXISA	MTYPE1A	NAME1A	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	etc.			
		MTYPE2A	NAME2A	etc.					
	NEWAXISB	MTYPE1 B	NAME1B	AXES1B	COEF1B	NAME2B	etc.		
		MTYPE1C	etc.						

Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM	AM1PSM3	Adding the wing lift to a free body load							
	3	AMONPNT1	WING	3	1.0				
		MONPNT3	FREEW	3	1400.	FREEW	5	-1000	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MONSUM	NAME	LABEL							
	MTYPE	NEWAXISA	NAME1A	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	etc.			
		NEWAXISB	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	etc.			
		NEWAXISC	etc.						

Alternate Example: Scale an existing monitor point

MONSUM	ROOT	Scale from in-lbs to newton-cm and from lbs to newtons							
	AMONPNT1	123456	ROOT	123	4.482	ROOT	456	11.385	

Describer	Meaning
NAME	Character string of up to 8 characters identifying the monitor result. (Character)
LABEL	A string comprising of no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
NEWAXISj	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 7. and 8.)
MTYPij	Monitor type to be merged. (Character, one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6.

Descriptor	Meaning
MTYPE	Monitor type to be merged in the alternate format. (Character; one of “AMONPNT1”, “AMONDSP1”, “SMONPNT1”, “SMONDSP1”, or “MONPNT3”; no Default). See Remarks 5. and 6.
NAMEij	Name of the monitored quantity that is to be merged
AXESij	Component axes that are to be summed. (any unique combination of the integers 1 to 6 with no embedded blanks)
COEFij	Coefficient to be applied to the component(s) called out on AXESij field. (Real; Default = 1.0)

#### Remarks:

1. The MONSUM is available in SOLs 101, 103, 108, 109, 111, 112, 144, 146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
2. The LABEL is a 56 character string that should be unique.
3. The MONSUM can be used to update an existing monitor result by setting all the NAMEij terms equal to NAME. In this case, the alternate format is available and the NEWAXISj component is a scalar multiple of the original component:

$$MONSUM_j = MR_j COEF_j$$

4. When the NAME differs from the NAMEij values, it must be unique with respect to all monitor quantities. The result of the MONSUM entry is to create new monitor point(s) that are equal to:

$$MONSUM_j = \sum_i^n COEF_{ij} MR_{ij}$$

where  $MR_{ij}$  is the result from the individual component.

5. The merged monitor points must be of a similar type. “Similar” types are defined as:  
Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3  
Average displacement monitor points: AMONDSP1 and SMONDSP1
6. For MONPNT1’s and MONDSP1’s, the MTYPij can be aerodynamic or structural. MTYPij=AMxxx1 designates aerodynamic while SMxxx1 designates a structural monitor point.
7. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are determined from NEWAXISj.
8. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6.
9. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM entry.
10. Structural monitor points may span superelements.

11. If multiple types are specified on a MONSUM, the resulting entry is of a type that appears on the MONSUM with the following order of precedence: smonpnt1, smondsp1, monpnt3, amonpnt1, amondsp1. E.g., a amonpnt1 and a monpnt3 appearing on the same entry will result in a monpnt3 regardless of which appears first.
12. If all the MTYPij values are of the same type, the alternate format provides a simplified interface.
13. A MONSUM can reference another MONSUM (including itself) as long as there is not a circular reference. A MONSUM cannot reference another MONSUM1.
14. MONPNT3's that have one, two or three excluded items (e.g., XFLAG=SMA) cannot be referenced on a MONSUM.
15. Since CP,X,Y,Z and CD are not known with certainty for a MONSUM, they are not printed as part of the monitor point output. Further, in SOL's 101 and 144, COMP=**\*\*SUM\*\***, is used to indicate that the associated monitor point has been derived from a MONSUM.

**MONSUM1**

Linear combination of two or more monitor points.

Defines a new monitor result that is the weighted sum of existing monitor results. The location of the computed MONSUM1 is specified. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3.)

Format:

1	2	3	4	5	6	7	8	9	10
MONSUM1	NAME	LABEL							
	CP	X	Y	Z	CD				
	NEWAXES	MTYP1	NAME1a	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	Etc			
		MTYP2	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	Etc.	.		
		MTYP3	Etc						

Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM1	ROOT	SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS							
		500.	0.0	25.0	20	200			
	123456	MONPNT3	FS_1000	123	-4.482	FS_1000	356	-11.385	
		AMONPNT1	ROOT	123	4.482	R00T	456	11.385	

Alternate Format:

1	2	3	4	5	6	7	8	9	10
MONSUM1	NAME	LABEL							
	CP	X	Y	Z	CD				
	MTYPE	NEWAXISA	NAME1a	AXES1A	COEF1A	NAME2A	AXES2A	COEF2A	
			NAME3A	AXES3A	COEF3A	Etc			
		NEWASIXB	NAME2A	AXES1B	COEF1B	NAME2B	AXIS2B	COEF2B	
			NAME3B	AXES3B	COEF3B	Etc.	.		
		NEWAXISC	Etc						

Alternate Example: Create a new monitor point result by adding an aerodynamic monpnt1 and a monpnt3

MONSUM1	ROOT	SCALE FROM IN-LBS TO NEWTON-CM AND FROM LBS TO NEWTONS							
		500.	0.0	25.0	20				
	MONPNT3	123456	FS_1000	123	-4.482	FS_1000	456	-11.385	
	AMONPNT1	123456	ROOT	123	4.482	R00T	456	11.385	

Describer	Meaning
NAME	Character string of up to 8 characters identifying the monitor result (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)
NEWAXESj	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks, see Remarks 5. and 6.)
MTYPi	Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.
NAMEij	Name of the monitored quantity that is to be summed.
AXESij	Component axes that are to be scaled. (any unique combination of the integers 1 to 6 with no embedded blanks. Must be a subset of NEWAXESj).Default =same as NEWAXESj
COEFij	Scaling coefficient to be applied to the component(s) called out on AXESij field. (Real, Default=1.0)

#### Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The result of the MONSUM1 entry is to create new monitor point(s) as follows:

$$MONSUM1_j = \sum_i^n COEF_{ji} MR_i$$

Where  $MR_i$  is the result from the individual component.

3. The merged monitor points must be of a similar type. “Similar” types are defined as:
  - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1’s the MTYPij can be aerodynamic or structural. MTYPij=AMONPNT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple components are to be summed, the NEWAXISj field must be the union of subsequent AXISij fields. If the subsequent AXISij field is blank, the components are the same as NEWAXISj.



6. If the NEWAXISj field indicates a single output, the AXISij fields must reference a single input, but it can be any value from 1 to 6. There may be as many as 6 AXISij and COEFij for each referenced NAMEij.
7. The same component cannot be referenced multiple times on the NEWAXISj fields for a single MONSUM1 entry.
8. If multiple types are specified on a MONSUM1, the resulting entry is of a type that appears on the MONSUM1 with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
9. A MONSUM1 can reference the results of another MONSUM1 or a MONSUMT as long as there is not a circular reference.
10. The MONSUM1 entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).
11. The CP, CD, and XYZ location coordinates are for reference only and do not affect the calculations.

**MONSUMT** Linear combination of two or more monitor points with moment transfer

Defines a new monitor result that is the sum of existing monitor results. The existing monitor points do not need to be of the same type but they must be of similar type (See Remark 3) 3.)

Format:

1	2	3	4	5	6	7	8	9	10
MONSUMT	NAME	LABEL							
	AXES	CP	X	Y	Z	CD			
	MTYP1	NAME1a	NAME1b	NAME1c	NAME1d	NAME1e	NAME1f	NAME1g	
	MTYP2	NAME2a	NAME2b	NAME2c	NAME2d	NAME2e	NAME2f	NAME2g	

Example: Create a new monitor point result by adding an aerodynamic and a monpnt3

MONSUMT	FS_900	FUSELAGE STATION 900							
	123456		500.	0.0	25.0	20			
	MONPNT3	FS_1000	FS_1100	FS_1200	MONPNT3				
	AMONPNT1	HTP_ROOT	AMONPNT1						

Descriptor	Meaning
NAME	Character string of up to 8 characters identifying the monitor result (Character)
LABEL	A string comprising no more than 56 characters (fields 3 through 9) that identifies and labels the merged monitor result.
CP	The identification number of a coordinate system in which the input (x,y,z) coordinates are defined. (Integer > 0; Default = 0)
X,Y,Z	The coordinates in the CP coordinate system about which the loads are to be monitored. (Real; Default = 0.0).
CD	The identification number of a coordinate system in which the resulting load components are output. (Integer > 0; Default = the coordinate system specified by the CP field)
AXES	The component axis of the newly-created monitor point into which the summed quantity will be stored (integer, any unique combination of the integers 1 to 6 with no embedded blanks.
MTYPi	Monitor type to be merged. (Character, one of “AMONPNT1”, “SMONPNT1”, or “MONPNT3”, no default). See Remarks 3. and 4.
NAMEij	Name of the monitored quantity that is to be summed.

Remarks:

1. The LABEL is a 56 character string that should be unique.
2. The NAME must be unique with respect to all monitor quantities. The result of the MONSUMT entry is to create new monitor point(s) as follows:

$$MONSUMT_j = \sum_i^n T_{ji} MR_i$$

Where  $MR_i$  is the result from the monitor points being summed and  $T_{ji}$  is set of partial rigid body vectors for the locations of the monitor points being summed, with the origin at the X, Y, Z location.

3. The summed monitor points must be of a similar type. “Similar” types are defined as:
  - Force and moment summation monitor points: AMONPNT1, SMONPNT1, MONPNT3
4. For MONPNT1’s the MTYPij can be aerodynamic or structural. MTYPij=AMONPT1 designates aerodynamic while SMONPNT1 designates a structural monitor point.
5. If multiple types are specified on a MONSUMT, the resulting entry is of a type that appears on the MONSUMT with the following order of precedence: SMONPNT1 (structural MONPNT1), MONPNT3, AMONPNT1 (aerodynamic MONPNT1). E.g., a AMONPNT1 (aerodynamic MONPNT1) and a MONPNT3 appearing on the same entry will result in a MONPNT3 regardless of which appears first.
6. A MONSUMT can reference the results of another MONSUMT or a MONSUM1 as long as there is not a circular reference.
7. The MONSUMT entry is available in SOLs 101,103,108,109,111,112,144,146, 200 and 400 (ANALYSIS = NLSTAT or NLTRAN).

## MOTION

Specifies Motion Direction in Brake System for Brake Squeal Analysis Using SOL 400

Defines motion direction of brake system for brake squeal calculations.

Format:

1	2	3	4	5	6	7	8	9	10
MOTION	ID		Velocity						
	FORM	COSX	COSY	COSZ	X	Y	Z		

Example:

1	2	3	4	5	6	7	8	9	10
MOTION	18		50.0						
	ROTAT	0.5	0.2	0.9	12.0	7.5	-2.0		

or

1	2	3	4	5	6	7	8	9	10
MOTION	18		100.0						
	TRANS	0.7	0.3	0.9					

Describer	Meaning
ID	Identification ID of the Motion referred by other entry card like <a href="#">BRKSYS</a>
VELOCITY	Angular velocity of rotation or moving velocity of translation motion (not used now, designed for velocity-related brake analysis in future).
FORM	Type of motion  ROTAT - rotating motion; or TRANS - translation moving
COSX(Y,Z)	For FORM=ROTAT, X (Y, Z) direction cosine (basic coordinate system) of axis of rotation For FORM=TRANS, X (Y, Z) direction cosine (basic coordinate system) of moving direction
X (Y, Z)	X (Y, Z) coordinate in basic coordinate system of rotation center

Remarks:

1. ROTAT and TRANS cannot be defined together in same entry card.

## MPC Multipoint Constraint

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

where  $u_j$  represents degree-of-freedom  $C_j$  at grid or scalar point  $G_j$ .

Format:

1	2	3	4	5	6	7	8	9	10
MPC	SID	G1	C1	A1	G2	C2	A2		
		G3	C3	A3	-etc.-				

Example:

MPC	3	28	3	6.2	2		4.29		
		1	4	-2.91					

Describer	Meaning
SID	Set identification number. (Integer > 0)
$G_j$	Identification number of grid or scalar point. (Integer > 0)
$C_j$	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
$A_j$	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

- Multipoint constraint sets must be selected with the Case Control command MPC = SID.
- The first degree-of-freedom ( $G_1$ ,  $C_1$ ) in the sequence is defined to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPC entry cannot be assigned dependent by another MPC entry or by a rigid element.
- Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
- The m-set degrees-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. See the [Degree-of-Freedom Sets](#) for a list of these entries.
- By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in [Parameters](#). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,3.

## MPCADD

## Multipoint Constraint Set Combination

Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.

Format:

1	2	3	4	5	6	7	8	9	10
MPCADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	-etc.-						

Example:

MPCADD	101	2	3	1	6	4			
--------	-----	---	---	---	---	---	--	--	--

Describer	Meaning
SID	Set identification number. (Integer > 0)
Sj	Set identification numbers of multipoint constraint sets defined via MPC entries. (Integer > 0)

Remarks:

1. Multipoint constraint sets must be selected with the Case Control command MPC = SID.
2. The Sj must be unique and may not be the identification number of a multipoint constraint set defined by another MPCADD entry.
3. MPCADD entries take precedence over MPC entries. If both have the same SID, only the MPCADD entry will be used.
4. By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in [Parameters](#). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,101.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

## MPCAX Conical Shell Multipoint Constraint

Defines a multipoint constraint equation of the form

$$\sum_j A_j u_j = 0$$

for conical shell coordinates, where  $u_j$  represents the degree-of-freedom  $C_j$  at ring  $RID_j$  and harmonic  $HID_j$ .

Format:

1	2	3	4	5	6	7	8	9	10
MPCAX	SID				RID1	HID1	C1	A1	
	RID2	HID2	C2	A2	-etc.-				

Example:

MPCAX	32				17	6	1	1.0	
	23	4	2	-6.8					

Describer	Meaning
SID	Set identification number. (Integer > 0)
RID <sub>j</sub>	Ring identification number. (Integer > 0)
HID <sub>j</sub>	Harmonic identification number. (Integer > 0)
C <sub>j</sub>	Component number. (1 ≤ Integer ≤ 6)
A <sub>j</sub>	Coefficient. (Real; Default = 0.0 except A1 must be nonzero.)

Remarks:

- MPCAX is allowed only if an AXIC entry is also present.
- The first degree-of-freedom in the sequence is assumed to be the dependent degree-of-freedom. A dependent degree-of-freedom assigned by one MPCAX entry cannot be assigned dependent by another MPCAX entry.
- Multipoint constraint sets must be selected with the Case Control command MPC = SID.
- Dependent degrees-of-freedom appearing on MPCAX entries may not appear on OMITAX, SPCAX, or SUPAX entries.
- See [Conical Shell Element \(RINGAX\)](#) (Ch. 3) in the *MSC Nastran Reference Guide* for further discussion of the problem.
- By default, the grid point connectivity created by the MPC, MPCADD, and MPCAX entries is not considered during resequencing, (see the PARAM,OLDSEQ description in [Parameters](#)). In order to consider the connectivity during resequencing, SID must be specified on the PARAM,MPCX entry. Using the example above, specify PARAM,MPCX,32.

**MPCD** Load Selectable Value for  $Y_m$  of Non Homogenous Multipoint Constraint

Defines a load selectable value for  $Y_m$  of a MPCY entry.

Format:

1	2	3	4	5	6	7	8	9	10
MPCD	SID	GM1	CM1	YM1	GM2	CM2	YM2		

Example:

MPCD	700	101	2	.06					
------	-----	-----	---	-----	--	--	--	--	--

Describer	Meaning
SID	Set identification number of the MPCD entry. (Integer > 0)
GMi	Grid or scalar point identification number. Along with Ci it identifies the Equation of a MPCY. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid points, blank, zero or 1 for scalar points.)
YMi	Right hand side value of MPC equation. (Real)

Remarks:

1. In static solution sequences, the SID is selected by the LOAD Case Control command.
2. The Gi,Ci referenced on this entry must also be referenced on a MPCY Bulk Data entry and selected by an MPC Case Control command.
3. Values YMi will override the value specified on an MPCY Bulk Data entry.
4. The LOAD Bulk Data entry will not combine an MPCD load entry.
5. Two separate MPC equations may be pointed to per entry.



**MPCREEP****SOL 600 Advanced Creep Options**

Specifies input values for Marc's creep parameter when creep analysis is performed using SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MPCREEP	ITYPE	IMPLK	IIMPLM	IHOW					

Example:

MPCREEP	1	0	0	0					
---------	---	---	---	---	--	--	--	--	--

Describer	Meaning
ITYPE	Type of explicit creep analysis enter one of the following: (Integer; Default = 0) 0 Normal creep (Maxwell Model) 1 Viscoplastic creep 2 Viscoplastic creep with nonassociative flow rule.
IMPLK	Flag determining if the explicit Kelvin model is to be used. (Integer; Default = 0) 0 Explicit Kelvin model is not used 1 Explicit Kelvin model is used.
IMPLM	Flag determining if the implicit Maxwell creep or implicit viscoplastic model is to be used. (Integer; Default = 0) 0 These models are not used. 1 The implicit Maxwell creep or implicit viscoplastic model is used.
IHOW	Flag specifying how the Maxwell creep model or implicit viscoplastic model is used. (Integer; Default = 0) 0 Use elastic tangent 1 Use secant tangent 2 Use radial return

Remarks:

1. Set IHOW=0 or leave field blank if IMPLM=0.
2. When using the implicit Maxwell creep model, the stress dependence must be in exponential form and the CRPLAW user subroutine cannot be used.
3. Only one MPCREEP entry may be entered in the input. If additional MPCREEP entries are found, the first will be used.
4. If ITYPE > 0, Bulk Data entry, MACREEP or MTCREEP is required in addition to this entry.

5. This entry maps to Marc's CREEP parameter.
6. This entry is not necessary if fields 2-5 are all zero or blank.
7. Kelvin creep requires user subroutines.

## MPCY Non Homogenous Multipoint Constraint

Defines a multipoint constraint equation of the form

$$A_m u_m + \sum_i A_i u_i = Y_m$$

where  $u_m$  represents degree-of-freedom  $C_m$  at grid or scalar point  $G_m$  defined to be the dependent degree-of-freedom.  $u_i$  represents degree-of-freedom  $C_i$  at grid or scalar point  $G_i$  defined to be the independent degree-of-freedom.  $Y_m$  is a value for the equation.

Format:

1	2	3	4	5	6	7	8	9	10
MPCY	SID	GM	CM	AM	YM				
		G1	C1	A1	G2	C2	A2		
		G3	C3	A3	...				

Example:

MPCY	70	205	1	1.0	.02-3				
		1608	1	1.2					

Describer	Meaning
SID	Set identification number of a multi-point constraint set. (Integer > 0)
GM	Identification number of grid or scalar point. (Integer > 0)
Cj	Component number. (Any one of the Integers 1 through 6 for grid points; blank, zero or 1 for scalar points.)
AM	Coefficient. (Real nonzero value)
YM	Right hand side value. (Real; Default = 0.0)
Gi	Identification number of grid or scalar point. (Integer > 0)
Ci	Component number. (Any one of the Integers 1 through 6 for grid point; blank or zero for scalar points.)
Ai	Coefficient. (Real; Default = 0.0)

Remarks:

1. Multipoint constraint sets must be selected with Case Control command, MPC=SID.

2. The  $m$ -set degree-of-freedom specified on this entry may not be specified on other entries that define mutually exclusive sets. The GM term is an equation marker. If PARAM,AUTOMSET,YES is on Nastran may choose internally another  $G_i$  as the actual dependent degree-of-freedom.

**MPHEAT****Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis**

Maps to Marc's HEAT parameter for SOL 600 heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
MPHEAT	IDIST	LHMAX	INTPT	ICONVT	ILAYC	LINR	NONH		

Example:

MPHEAT	1	1	1	0	3	1	0		
--------	---	---	---	---	---	---	---	--	--

Describer	Meaning
IDIST	Temperature distribution in thickness direction of heat transfer shell elements. (Integer; Default = 0) 0 Linear Variation (membrane) 1 Quadratic variation
LHMAX	Maximum number of latent heats associated with any material. (Integer; Default = 0)
INTPT	Controls integration point output. (Integer; Default = 0) 0 No integration point output 1 Gradients and fluxes at integration points 2 Save as 1 plus flux values at grid points
ICONVT	Flag to include convective terms. (Integer; Default = 0) 0 Do not include convective terms 2 Include convective terms
LAYC	Number of through thickness layers for heat transfer shells. (Integer; Default = 1)
LINR	Flag controlling linearization of surface energy and receding surface calculations. (Integer; Default = 1) 1 Linearize the calculations 2 Do not linearize the calculations
NONH	Flag to store nonhomogeneous density for postprocessing. (Integer; Default = 0) 0 Do not store the values 1 Store the values

**MPROCS**

Defines a Processor Entry to be Used in the SOL 600 Marc Parameter Section

Defines a processor entry to be used in the SOL 600 Marc Parameter Section. It controls the use of vectorization and parallelization in the element assembly phase in Marc. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MPROCS	I2	I3	I4	I5	I6	I7			
	IRF	IFGF	IAXIS	WGHT					
	Dx	Dy	Dz	X	Y	Z			

Example:

MPROCS	1	1	1	1	11				
--------	---	---	---	---	----	--	--	--	--

Describer	Meaning
I2	Value of Marc's 2nd processor field. Number of CPUs to use. (Integer; Default = 0)
I3	Value of Marc's 3rd processor field. Optimal vector length (Defaults to 32 or 64 depending on the computer system). (Integer; Default = 0)
I4	Value of Marc's 4th processor field (enter 1 if beta matrices are to be formed in parallel). (Integer; Default = 0)
I5	Value of Marc's 6th processor field (enter 1 to use DDM single file input). (Integer; Default = 0)
I6	Value of Marc's 7th processor field -- Domain Decomposition Method. (Integer; Default = 0) Enter 11 to use Metis Best (best method of 12, 13 or 14) decomposition; Default. Enter 12 to use Metis Element-Based decomposition. Enter 13 to use Metis Node-Based decomposition. Enter 14 to use Vector decomposition. Enter 15 to use Radial decomposition. Enter 16 to use Angular decomposition.
I7	Value of Marc's 8th processor field. Enter 0 to use in-core storage for DDM single file creation. Enter 1 to use out-of-core storage. (Integer; Default = 0)
IRF	Island removal flag for domain decomposition. (Default = 0) 0 Do not remove islands 1 Attempt to remove islands
IFGF	Fine graph flag for domain decomposition. (Default = 0) 0 Coarse graph 1 Fine graph



Describer	Meaning
IAXIS	Control of point on axis of rotation for radial/angular domain decomposition 0        Use centroid of the boundary box of the model 1        User supplied point (supply X, Y, Z below)
WGHT	Element coefficient weight. Controls balance between computational costs of domains, range is 0.0 to 1.0. (Default = 1.0; Use full element weight) 0.0 - Do not use element weight
Dx	First direction cosine of vector used for decomposition method 14, 15, or 16
Dy	Second direction cosine of vector used for decomposition method 14, 15, or 16
Dz	Third direction cosine of vector used for decomposition method 14, 15, or 16
X	X coordinate of point on axis
Y	Y coordinate of point on axis
Z	Z coordinate of point on axis

#### Remarks:

1. This entry should only be made for special cases when using DDM with Marc's single file parallel capability. Do not enter except for SOL 600 parallel executions.
2. Consult the Marc documentation volumes A, B, C for more detailed descriptions of this entry.
3. Enter only one of MPROCS entry in any given file. If more than one is entered, the first encountered will be used.
4. The continuation lines may be omitted if not required.
5. I2, I3, and I4 are no longer used and should be zero or blank.

**MREVERS** Defines Which Elements Require Node Numbering to be Reversed - SOL 600

This entry is only used if the checks built into SOL 600 somehow fail to predict some elements which need node numbering reversed. Nastran allows clockwise as well as counter clockwise node numbering. Marc only allows counter clockwise numbering for most elements. SOL 600 has builtin logic to check for node reversal, however there may be some instances where Marc gives an “inside out” message during increment (before any nonlinear loads have been applied). This rarely happens unless field 7 of the GRID entry is set, in which case it sometimes happens. When this happens, the user can apply this entry to instruct SOL 600 how to renumber elements which have “inside out” messages during increment zero. (If “inside out” messages occur after increment zero, this is normally due to large loads and the element has either failed or the model needs to be re-meshed).

**Format:**

1	2	3	4	5	6	7	8	9	10
MREVERS	N1	M1	N2	M2	N3	M3	N4	M4	
	ID1	ID2	ID3	ID4	ID5	ID6	ID7	ID8	
	ID9	ID10	etc.						

**Example:**

MREVERS		2	3						
	500	510	520	888	889				
	950								

Describer	Meaning
Ni	Node number location to be reversed. (Integer; no Default; see Remark 1.)
Mi	Paired node number location to be reversed. (Integer; no Default; see Remark 1.)
IDI	Element identification number. (Integer; no Default)

**Remarks:**

- Example of node number locations are as follows:  
 For 4-node quads the node number locations are 1, 2, 3, 4  
 For 8-node hexas the node number locations are 1, 2, 3, 4, 5, 6, 7, 8
- The 2<sup>nd</sup> and succeeding lines may be used to make it easy to enter elements with “inside out” messages in a preliminary marc.out file. The user can grep for “inside out”, save the message in a file, easily edit the file to retain only the element numbers and then add these to lines 2 and subsequent of the alternate format.
- Up to four pairs of node locations may be reversed per entry. If additional paris are necessary, repeat the entry and its continuation lines.
- The example will reverse nodes locations 2 and 3 for elements 500, 510, 520, 888, 889 and 950.





5. This entry may only be used in MD Nastran R2.1 and later versions.
6. See PARAM,MAXIREVV for a similar option.

## MRSSCON

### Auxiliary data for RSSCON for use in SOL 600

Defines auxiliary data for RSSCON for use in SOL 600. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MRSSCON	ICONV	IDB	IDP	IDM	A	J	I	E	

Example:

MRSSCON	0	5000	5000	900	20.0	200.	100.	30.0E	
---------	---	------	------	-----	------	------	------	-------	--

Describer	Meaning
ICONV	Flag indicating how RSSCON rigid elements will be converted in SOL 600. (Integer; Default = 0) 0 Convert all RSSCON to CBEAM using values in fields 2-8 1 Convert all RSSCON to RBE2 using 3 degrees of freedom 2 Convert all RSSCON to RBE2 using 6 degrees of freedom
IDB	Starting CBEAM ID for all RSSCON's that are mapped to CBEAM's or RBE2 ID if RSSCON's are mapped to RBE2's (Integer; Default = 99000; see Remark 1.)
IDP	PBEAM ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 5. (Integer; Default = 99000)
IDM	MAT1 ID for use by all RSSCON's in the model mapped to CBEAM's. See Remark 6. (Integer; Default = 99000)
A	Area to be placed in the PBEAM (Real; Default = 10.0)
J	Torsional moment of inertia to be placed in the PBEAM (Real; Default = 20.0)
I	Moment of inertia (in both directions) to be placed in the PBEAM (Real; Default = 10.0)
E	Elastic modulus to be placed in the IDM MAT1 entry (Real; Default = 1.0E6)

Remarks:

1. ICONV=1 or 2 may cause MPC conflicts and/or may fail if AUTOMSET,YES is used.
2. RSSCON type=elem is not presently supported by SOL 600.
3. This entry applies to RSSCON type=grid where ES1, EA1, EB1 all have different coordinates, and if used ES2, EA2 and EB2 do not have the same coordinates as each other (see RSSCON entry for meaning of these values).
4. Only one MRSSCON entry should be include in the input data. If more than one is included, the first as determined by XSORT, will be used.

5. If IDP is entered, a PBEAM with and id of IDP and properties A, J, and I will automatically be created.
6. If IDM is entered, a MAT1 with a Young's modulus of E and Poisson ratio of 0.3 will automatically be created.

**MSTACK**

Defines the Direction in Which 3D Solid Composites are Stacked -SOL 600

Defines the direction in which 3D Solid Composites are stacked. Used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MSTACK	ID1	THRU	ID2	IDIR					

Example:

MSTACK	100	THRU	200	1					
--------	-----	------	-----	---	--	--	--	--	--

Describer	Meaning
ID1	Starting solid element ID (Integer; Required; no Default)
THRU	Enter the character string THRU if more than one element is desired
ID2	Ending solid element in a range of ID1 to ID2 (Integer; optional; no Default)
IDIR	Stacking direction for this range of solid elements (Integer; Required; Default = 1)
	1 Thickness direction is 1-2-3-4 face to 5-6-7-8 face (for CHEXA)
	2 Thickness direction is 1-5-8-4 face to 2-6-7-3 face (for CHEXA)
	3 Thickness direction is 2-6-5-1 face to 3-7-8-4 face (for CHEXA)

Remarks:

1. Enter as many MSTACK lines as necessary to define any solid composites where the defaults do not apply. If IDIR is 1 for all elements in the range, this entry is not required.
2. This entry equates to Marc's EGEOM3 value for solid composite elements (for example element 149).

**MT16SEL**

Limits results to selected elements or grids for t16 and t19 for SOL 600

Limits elements and/or grid results to selected elements or grids for t16 and t19 file results. Used in SOL 600.

**Format**

1	2	3	4	5	6	7	8	9	10
MT16SEL	TYPE	ID1	THRU	ID2	BY	ID3			
	TYPE	ID1	THRU	ID2	BY	ID3			

**Example:**

MT16SEL	GRID	1	THRU	100					
	GRID	2525	THRU	3000	BY	10			
	ELEM	100	THRU	500	BY	2			
	ELEM	1000	THRU	2000					

Descriptor	Meaning
TYPE	Type of output - Enter one of the values “GRID”, or “ELEM”, for “GRID”, nodal results such as displacement, spc force, etc are selected. For “ELEM”, element results such as stress, strain, etc. are selected. This entry should normally be used in combination with MARCOUT unless the MARCOUT defaults are satisfactory for the model.
ID1	Starting ID of above selection. (Integer; no Default required value)
ID2	Ending ID of above selection. (Integer; Default = ID1)
ID3	Increment by value. (Integer; Default = 1)

**Remarks:**

1. It is highly recommended that all grid and element output be placed on the t16 file since it is not usually known where the maximum values will occur and the max/min values could easily be missed. Also, if all grid/element values are not selected, contour plots could be misleading. To output all grid/element results in the t16 file, do not make any MT16SEL entries.
2. This entry may not be used when OUTF options (which requires a t16 to op2 conversion) on the SOL 600 entry are requested. If this entry is made together with any OUTF options, this entry will be ignored and a warning message issued.
3. ID2 must be blank, zero or greater than ID1 (if ID2 is blank or zero, only ID1 will be used)
4. ID3 must not be negative. It can be blank or zero in which case it is reset to one.
5. This entry (with the exception of Remark 3) activates Marc's POST version 13 and overrides any other POST version specifications such as PARAM,MARCPST.
6. This entry can be used to also obtain a t19 file with selected element and/or grid results by placing PARAM,MARCT19,1 in the bulk data.

7. It is not presently possible to only create a t19 file with selected element/grid results using SOL 600. If a t19 file is desired, a t16 file must also be created.

**MT16SPL****Split a Marc t16 file into one or more smaller t16 files in SOL 600**

Determines how to split a Marc t16 file into one or more smaller t16 files in SOL 600. Splitting of a large t16 file is sometimes necessary if the postprocessor aborts due to the large amount of data or if the results need to be transferred to another computer for postprocessing.

Format: This entry is available in small field format only:

1	2	3	4	5	6	7	8	9	10
MT16SPL	IOPT	NVECT	FNAME						
	ID1	START	LAST	INCR	(remark 4)				
	ID2	START	LAST	INCR	LastFew				

Examples:

MT16SPL	0								
	1	5	100	5	3				

MT16SPL	0		old1.mar	c.t16					
	1	10	80	10	3				
	2	90	100	5	4				

MT16SPL	1	1							
	1	100							

MT16SPL	3	2	old2.t16						
	1	10	100	10	3				

Create several new t16 files with 2 increments each using original increments 10, 20, ... 98, 99, 100. This is a restart job and uses old2.t16 (from a standalone Marc run). New files will be created as follows:

jid.0001.t16 with old increments 10, 20  
 jid.0002.t16 with old increments 30, 40  
 jid.0003.t16 with old increments 50, 60  
 jid.0004.t16 with old increments 70, 80  
 jid.0005.t16 with old increments 90, 98  
 jid.0006.t16 with old increments 99, 100

Describer	Meaning
IOPT	Option of how to split up the "old" t16 file. (Integer; Default = 0)
0	Split into as many new t16 files as there are continuation lines of this entry, use the current jid.marc.t16 file, FNAME should not be entered.
1	Split into as many new t16 files with NVECT original increments each, use the current jid.marc.t16 file, FNAME should not be entered.

Describer	Meaning
	2 Split into as many new t16 files as there are continuation lines of this entry, use a previously generated t16 file whose name is specified by FNAME.
	3 Split into as many new t16 files with NVECT original increments each, use a previously generated t16 file whose name is specified by FNAME.
NVECT	Number of increments to be placed on each new t16 file (only used if IOPT is 1 or 3) (Integer; no Default)
FNAME	Original t16 filename - Only used if IOPT is 2 or 3. (Character; no Default) The filename may extend from fields 4-9. If it is more than 8 characters long, this entry must be made in fixed format. the entire t16 filename including the t16 extension should be entered. No upper case letters may be used. (Integer > 0)
IDI	New t16 plot ID (must start with 1 and increase by 1 on each continuation line.) (Integer; no Default)
START	Starting Marc increment to be placed on new t16 file. (Integer; Default = 1)
LAST	Last Marc increment in range of start-last-incr to be n new t16 file. (Integer; Default is last increment on original t16 file if FNAME is blank. If FNAME is not blank, LAST must be an accurate value which can be obtained from the .sts file of the original run that produced FNAME.)
INCR	Increments to be used for start-last-incr to be on new t16 file. (Integer; Default = 1)
Last Few	In addition to START, LAST, INCR the last several increments may be placed on the t16 file. In the first example increments 5, 10, 15, ..., 100, 99, 98 are placed on the new t16 file. LastFew may only be entered on the last line. (Integer; Default = 2)

#### Remarks:

1. Sometimes large SOL 600 models do not converge on a user wishes to examine output at unknown time intervals. To determine what is happening, it is frequently necessary to plot the results at several output intervals. In fact, sometimes the last increment may have bogus results due to divergence. At present, MSC GUI programs sometimes are not capable of postprocessing the large amount of data one would like to include in a single t16 file. This option allows you to break up the t16 file into one or more smaller files.
2. GUI's might only be able to handle one increment per t16 file for extremely large models. To specify this, only enter ID and START as in the 3rd example.
3. All t16 files will have the geometry as well as the specified output increments.
4. LastFew may only be entered on the last line.
5. Nastran may be restarted to perform this step. To do so, enter  

```
SOL 600, ID t16split=fname
```

Where fname is the jid of the original job.
6. The new files will be named jid.ID.t16. Examples are as follows:



Case 1 - Split up t16 as part of current run starting with jid1.dat as the Nastran input SOL 600 will create jid1.marc.t16 with a full set of output increments (unless reduced by you).

jid1.0001.t16

jid1.0002.t16

etc.

Case 2 - Split up a t16 file formed by a previous SOL 600 run named jid1.dat (the t16 file is named jid1.marc.t16). The current Nastran input file to split up the original t16 is named jid2.dat. The new t16 files will be designated:

jid2.00001.t16

jid2.00002.t16

etc.

7. Nastran will spawn Marc's pldump2000 program to split up the original t16 file.
8. If FNAME is entered (IOPT=2 or 3), OUTF options (on SOL600,ID) will be ignored. If FNAME is blank and IOPT=0 or 1 and OUTF options are requested, the t16op2 translator process the full (unsplit) jid.marc.t16 file. Future implementations may allow processing of the split t16 files.
9. Only one MT16SPL (plus many continuation lines as necessary) is allowed per job.

**MTABRV** Defines a List of Tables to Modify and/or Positions of Values - SOL 600

In some cases, a model is built with tables defined backwards. For example, for a stress-strain curve the compression position of the curve might really need to be the tension portion and visa versa. This entry allows them to be reversed. In addition, this entry allows a user to add a point to the lower and/or upper end of the tables in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MTABRV	ID1	THRU	ID2	IREV	XLOW	YLOW	XHIGH	YHIGH	

Example:

MTABRV	1	THRU	10	1	-100.	-1.0E7	0.01	1.0E7	
	21	THRU	25	1					
	45	THRU	48				25.5	66523.	

Describer	Meaning
ID1	Starting table ID to be reversed. (Integer; Required; no Default)
THRU	Enter the character string THRU if more than one grid is desired.
ID2	Ending table ID of the range. (Integer; or blank. If blank; ID2=ID1)
IREV	Option to reverse (flip) positive and negative values and change the sign for tables in the range ID1 to ID2. (Integer; Default = 0)
	0 Do not reverse
	1 Reverse
XLOW	Option to add a point to the lower end of these tables. If so, XNEG is the lower end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)
YLOW	Option to add a point to the lower end of these tables. If so, YNEG is the lower end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)
XHIGH	Option to add a point to the upper end of these tables. If so, XPOS is the upper end X value to be added (after reversal, if IREV=1). (Real; Default = 0.0)
YHIGH	Option to add a point to the upper end of these tables. If so, YPOS is the upper end Y value to be added (after reversal, if IREV=1). (Real; Default = 0.0)

Remarks:

1. Missing tables in the range ID1 to ID2 will be ignored and no error will be produced.
2. This capability is limited to tables defined using TABLES1, TABLED1 and/or TABLEM1 and the behavior in the x and y directions must be linear.

3. Values XLOW, YLOS, XHIGH, YHIGH are not reversed and the signs are not changed even if IREV=1.
4. The first example reverses tables 1-10 and also adds a point at the lower and upper end. The second example reverses tables 21-25, no additional points are added. The third example adds a point to the upper end of tables 45-48, does not reverse the tables or add a point to the lower end.
5. This entry will reverse both X and Y of the specified tables and is not capable of reversing only X or Y.

## MTCREEP

## AUTO THERM CREEP Iteration Control - SOL 600

Controls a transient thermal creep analysis. This entry or the MACREEP entry is required if ITYPE is not zero on the MPCREEP entry in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
MTCREEP	ID	Tchg	Nmax	Iasmb	Ttot	Tincr	Tcrep	Nsub	
	Maxit	Nupd	VV1	VV2	VV3	IABS			

Example:

MTCREEP	11	1.0	99999	1	2000.	10.	1000.	50	
	5	1	.5	.1	.05				

Describer	Meaning
ID	Identification number of a matching NLPARM Case Control command (for statics) or entry (for dynamics). (Integer; no Default; Required field)
Tchg	Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)
Nmax	Maximum number of increments allowed. (Integer; Default = 999999) (2,2)
Iasmb	Reassembly interval for element matrices. (Integer; Default = 1) (2,3)
Ttot	Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)
Tincr	Suggested time increment for creep analysis (Real; no Default) (3,1)
Tcrep	Total creep time to be covered in this creep analysis (Real; no Default) (3,2)
Nsub	Maximum number of subincrements to be allowed during this creep analysis (Integer; Default = 50). (3,3)
Maxit	Maximum number of iterations allowed to modify the time step during any increment. (Integer; Default = 5) (3,4)
Nupd	Number of increments between stiffness matrix updates. (Integer; Default = 1) (3,5)
VV1	Tolerance value #1. (Real; see below for defaults) (4,1)
	IABS=0     Enter the tolerance on the creep strain increment relative to the elastic strain. (Default = 0.5)
	IABS=1     Enter the maximum creep strain increment. (Default = 0.01)
VV2	Tolerance value #2 (Real; see below for defaults) (4,2)
	IABS=0     Enter the tolerance on the stress change per increment divided by the total stress. (Default = 0.1)
	IABS=1     Enter the maximum stress increment. (Default = 100.0)

Describer	Meaning
VV3	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05) (4,3)
IABS	Flag controlling relative or absolute convergence testing. (Integer; Default = 0) (4,5)
	0           Relative checking is used
	1           Absolute checking is used

Remarks:

1. This entry maps to Marc's AUTO THERM CREEP entry.
2. This entry will be used to instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. Bulk Data entries, MPCREEP and MCHSTAT, must also be entered in addition to this entry.
4. (i,j) refer to Marc's AUTO THERM CREEP (data block, field).
5. Bulk Data entries MACREEP and MTCREEP should not be entered in the same input file.

**M THERM** Iteration Control for Automatic Thermal Loading for Structural Analysis Following a SOL 600 Heat Transfer Analysis

Iteration control for automatic thermal loading for structural analysis following a SOL 600 heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
M THERM	ID	Tchg	Nmax	Iasmb	Ttot	Tincr			

Example:

M THERM	21	1.0	99999	1	2000.	10.			
---------	----	-----	-------	---	-------	-----	--	--	--

Describer	Meaning
ID	Identification number of a matching Case Control NLPARM command (for statics) or entry (for dynamics). (Integer; no Default; Required field)
Tchg	Maximum temperature change to be used per step of the stress analysis. (Real; no Default; Required value) (2,1)
Nmax	Maximum number of increments allowed. (Integer; Default = 999999) (2,2)
Iasmb	Reassembly interval for element matrices. (Integer; Default = 1) (2,3)
Ttot	Total transient time from corresponding heat transfer analysis. (Real; no Default) (2,4)
Tincr	Suggested time increment for creep analysis. (Real; Default = 0.0; leave blank if analysis is not a creep analysis) (2,5)

Remarks:

1. This entry maps to Marc's AUTO THERM entry.
2. This entry will be used to instead of AUTO STEP or AUTO INCREMENT entries in the Marc file. It is suggested that if this entry is used, NLAUTO and NLSTRAT should not be specified (and will be ignored if entered).
3. (i,j) refer to Marc's AUTO THERM (data block, field).
4. This entry may be omitted in which case an AUTO THERM entry as follows will be used  
 AUTO THERM  
 1., 9999, 0, 1.0

## NHRMPRM

## Parameters for Nonlinear frequency response analysis (SOL 128)

Format:

1	2	3	4	5	6	7	8	9	10
NHRMPRM	PARAM1	VAL1	PARAM2	VAL2					

Example:

NHRMPRM	MNCSF	3	NLHDIAG	1					
---------	-------	---	---------	---	--	--	--	--	--

Var Name	Type/ Default	Description
NLHRED	I = -1	Use reduction for nonlinear analysis.
NHPLUS	I = 20	Number of extra points to avoid aliasing.
NLHTOL	R = 1.0E-5	Tolerance for convergence.
NLHTWK	R = 1.1	Push-off factor for line search in case convergence fails.
MXICODE0	I = 5	If solution fails to converge for mxicode0 steps, new trial displacement is used.
TICPNCH	I = 0	If TICPNCH = -1 then write a TIC Bulk Data entry for each degree-of-freedom in the d-set to be included in a subsequent transient analysis run.
MNCSF	I = 1000	Max number of non-converged sequential frequencies for RESET.
MFRINT	R = 10.	Maximum frequency interval for RESET.
NLHDIAG	I = 0	Output additional diagnostic messages.
CONTP	I = 0	Continuation procedure.

Remarks:

1. Following parameters can also be specified using PARAM option: NLHRED, NHPLUS, NLHTOL, NLHTWK, TICPNCH and MXICODE0.  
See description under PARAM for more details about these variables.  
In case a parameter is specified in NHRMPRM and also using PARAM, the value specified using PARAM entry is used.
2. Parameters MNCSF or MFRINT can be used to reset initial condition in case of non-convergence. If both these parameters are specified, the condition which is satisfied first is used.
3. CONTP = 1 is used to initiate continuation procedure method. In this case, following parameters are not used in the analysis: NLHRED, MNCSF, MFRINT. Currently, the CONTP option is available for frequency-independent analysis only.  
Here, NLHRED is set to 0 to not use the reduction method. And initial condition for the continuation procedure is determined using the results from two previously converged solutions.

## NLADAPT

## Additional Parameters for Automatic Load or Time Stepping

Defines additional parameters for automatic load or time stepping used with enhanced nonlinear in SOL 400. *NLADAPT is an obsolete option from MD Nastran 2010 onwards. It is recommended that NLADAPT be replaced by NLSTEP.*

Format:

1	2	3	4	5	6	7	8	9	10
NLADAPT	ID								
	“STEP”	RSMALL	RBIG	TSMIN	TSMAX	NSMAX	NRECYC	SFACT	
		IDAMP	DAMP						
	“CREEP”	RAC	TCSTRN	TCSTRC	TCOFF				

Example:

NLADAPT	700								
	STEP								
		4							
	CREEP	1							

NLADAPT	800								
	CREEP	1							
	STEP								
		4							

Describer	Meaning
ID	Identification number of a NLPARM or entry. (Integer > 0)
“STEP”	Keyword indicating the following entries are for enhanced general load step or time step convergence. (Character)
RSMALL	Smallest scale factor for time step changes. See Remark 5. (Real; Default= 0.1)
RBIG	Largest scale factor for time step changes. See Remark 5. (Real; Default = 1.5)
TSMIN	Smallest ratio of a time step to the total time. (Real; Default = total time divided by number of time steps)
TSMAX	Largest ratio of a time step to the total time. (Real; Default = 0.5)
NSMAX	Maximum number of steps allowed. (Integer; Default = 99999)
NRECYC	Number of recycles per increment. (Integer; Default = 10)
SFACT	Scale factor for time step changes. See Remark 6. (Real; Default = 1.2)
IDAMP	Enter 4 to activate artificial damping. (Integer 0 or 4; Default = 0)
DAMP	Damping factor for activated artificial damping. (Real; Default = 2.E-4)



“CREEP”	Keyword indicating the following entries are for enhanced creep analysis. (Character)
RAC	Flag controlling relative or absolute convergence. (Integer 0 or 1; Default = 0)
	0           Relative checking used.
	1           Absolute checking used.
TCSTRN	Creep strain tolerance. (Real)
	RAC = 0     Enter the tolerance on the creep strain increment divided by the elastic strain. (Real; Default = 0.5)
	RAC = 1     The maximum creep strain increment allowed. (Real; Default = 0.1)
TCSTRS	Creep stress tolerance (Real)
	RAC = 0     Enter the tolerance on the stress increment divided by the total stress. (Real; Default = 0.1)
	RAC = 1     enter the maximum stress increment. (Real; Default = 100.0)
TCOFF	Tolerance on low stress point cutoff. Points with a stress lower than this ratio relative to the maximum stress in the structure are not used in the creep tolerance checking. (Real; Default = 0.05)

**Remarks:**

1. This entry must point to an existing NLPARM or entry and is only used in SOL 400.
2. The keywords and their associated entries may occur in any order.
3. This entry computes an initial time step  $TINIT=1.0/NINC$  if pointing to a NLPARM entry and  $TINT=1.0/DT$  if pointing to a entry.
4. If this entry is used, it is preferred KMETHOD on the NLPARM or METHOD on the be set to PFNT, FNT, ITER, or AUTO.
5. The scale factor is defined as the new step size divided by the time step size.
6. SFACT is only used when artificial damping is activated.

## NLAUTO Parameters for Automatic or Fixed Load/Time Stepping - SOL 600

Defines parameters for automatic or fixed load/time stepping used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLAUTO	ID	TINIT	TFINAL	RSMALL	RBIG	TSMIN	TSMAX	NSMAX	
	NRECYC	IENHAN	IDAMP	NSTATE	NCUT	LIMTAR	IFINISH	FTEMP	
	SFACT	IFLAG	IDTAB	DAMP	IDMPFLG		IPHYS	I313	
	CRITERIA	SETID	Y1	X1	Y2	X2	Y3	X3	
		Y4	X4						

Example:

NLAUTO	1	.01	1	1	10	-1.5	.5		
	5		1			-1			
	1.2								

Describer	Meaning
ID	Identification number referenced by the NLPARM, , or TSTEP Case Control command for the applicable subcase. See Remarks 1. and 2. Include a NLPARM or entry for the subcase in addition to the NLAUTO entry. See Remarks 1. and 2. (Integer > 0; no Default)
TINIT (2,1)	Initial time step. (Real; Default is determined by NLPARM NINC or DT, see Remark 8.)
TFINAL (2,2)	Total time period. If OUTR options are used, for a static analysis, TFINAL must be 1.0. (Real; Default is 1.0 or Remark 8.)
RSMALL (2,3)	Smallest ratio between steps. (Real; Default = 0.1 or Remark 7.)
RBIG (2,4)	Largest ratio between steps. (Real; Default = 10.0 or Remark 7.)
TSMIN (2,5)	Minimum time step. (Real; Default is total time divided by number of time steps or Remark 7.)
TSMAX (2,6)	Maximum time step. (Real; Default is total time or Remark 7.)
NSMAX (2,7)	Maximum number of steps allowed. If IENHAN=2, NSMAX is the number of fixed time steps. (Integer; Default = 999999 or Remark 7.)

Describer	Meaning
NRECYC (2,8)	Desired number of recycles per increment. For beam or shell bending NRECYC should be set to 10. If large deformation is anticipated, NRECYC = NO is necessary for plate and shell buckling. PARAM,MARCDEF,0 can be used to set NRECYC=10 without entering NLAUTO. (Integer; Default = 5 or Remark 7.)
IENHAN (2,9)	Enter 1 to activate the enhanced scheme, 0 otherwise. Enter 2 to use fixed time stepping (enter the number of time steps using NSMAX) (Integer)
IDAMP (2,10)	Artificial damping control. (Integer; Default = 1, or Remark 7.) Enter 0 to not use artificial damping Enter 1 to use artificial damping if time step is less than minima time step. Reduce the minimum time step by 1000. IDAMP=1 is usually necessary (in combination with NRECYC=10) for plate and shell buckling models. Enter 2 to always use artificial damping. The value to use is given by DAMP which scales the damping matrix. Enter 4 to always use artificial damping which is determined by the strain energy in the first increment of the present load case. DAMP is used to scale the strain energy. Enter 5 to set the time step as if artificial damping was being used, but do not actually add artificial damping. Enter 6 to add artificial damping when the minimum time step has been reached.
NSTATE (3,1)	Number of states for post file. (Integer; enter only if ienhan = 1)
NCUT (3,2)	Maximum number of times to cut down time step in an increment. (Integer) (Default = 10 or Remark 7.; enter only if ienhan = 1)
LIMTAR (3,3)	Enter 0 to create criteria as limits, 1 to treat criteria as targets. (Integer, Default = 0 or Remark 7.; enter only if ienhan = 1)
IFINISH (3,4)	Enter 1 to finish time period when all nodal temperatures fall below FTEMP. Enter -1 if all nodal temperatures should exceed FTEMP. Enter 0 to omit temperature check. (Integer; enter only if ienhan = 1, Default = 0 or Remark 7.)
FTEMP (3,5)	Finish temperature, use with IFINISH. (Real; enter only if ienhan = 1, Default = 0.0 or Remark 7.)
SFACT (3,6)	Scale factor for time step changes other than changes due to user criteria. (Real) (Default = 1.2 or Remark 7.; enter only if ienhan = 1)
IFLAG (3,7)	Enter flag to override CREEP and DYNAMIC parameters as specified in the Marc input parameter section for this load case. (Integer) 0 Do not override parameters. 1 Turn off CREEP and DYNAMICS. 2 Turn off CREEP. 3 Turn off DYNAMICS.
IDTAB (3,8)	Table ID scaling damping factor (see next item) (Integer)

Descriptor	Meaning
DAMP (3,9)	Damping factor for artificial damping. The number entered here depends on the IDAMP option. If IDAMP is 1, the damping matrix is scaled by setting this factor to be the ratio of the initial damping energy to the initial strain energy (Defaults to 1e-5). If IDAMP is 2, the damping matrix is directly scaled by this factor. If IDAMP is 4, the estimated total damping energy in the subcase will be this factor times the estimated total strain energy. (Default value of 2.e-4 or Remark 7.)
IDMPFLG (3,10)	Enter 1 to put states reached by the above IDMPFLG flag on the post file. (Integer)
IPHYS (3,12)	Flag to determine if automatic physical criteria should be added and how analysis should proceed if they are not satisfied. (Integer) 2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied. 1 Add automatic physical criteria. Stop when any user criteria are not satisfied. -1 Add automatic physical criteria. Continue when any user criteria are not satisfied. -2 Do not add automatic physical criteria. Stop when any user criteria are not satisfied.
I313 (3,13)	Flag to check if dynamic integration error checks should be made while determining the timestep (single step Humbolt and Newmark-Beta only) (Integer, Default = 0 or Remark 7). 0 Skip error check. 1 Include error check.
CRITERIA (4,1)	Enter an integer corresponding to the criteria desired. See Remark 1. (Integer; no Default)
SETID (4,2)	Case Control Set ID of nodes or elements for which this criteria will apply. Restriction: Must be one of first 25 sets entered in the case control. Leave blank if "ALL" is desired. (Integer; Default is all or Remark 7.)
Y1 (4,3)	Time step adjustment value. See Remark 10. (Real)
X1 (4,4)	Time step adjustment value. See Remark 10. (Real)
Y2 (4,5)	Time step adjustment value. See Remark 10. (Real)
X2 (4,6)	Time step adjustment value. See Remark 10. (Real)
Y3 (4,7)	Time step adjustment value. See Remark 10. (Real)
X3 (4,8)	Time step adjustment value. See Remark 10. (Real)

Describer	Meaning
Y4 (4,9)	Time step adjustment value. See Remark 10. (Real)
X4 (4,10)	Time step adjustment value. See Remark 10. (Real)

#### Remarks:

1. Enter the following index in the CRITERIA field (a limit of 9 criteria may be specified and the usual option is to specify none).

1	Strain Increment	2	Plastic Strain Increment
3	Creep Strain Increment	4	Normalized Creep Strain Increment
5	Stress Increment	7	Strain Energy Increment
8	Temperature Increment	9	Displacement Increment
10	Rotation Increment	12	Stress
13*100+n	State Variable n		

2. Values entered on NLAUTO override values with the same meaning if entered elsewhere (for example, on the NLPARM or entry).
3. If the NLAUTO entry is used, there should also be a corresponding NLPARM or . The matching NLPARM entry must have KMETHOD=AUTO or ITER. If is the matching entry, then field 6 must be blank or have the value ADAPT.
4. Values such as (3,7) indicated corresponding item on Marc's AUTOSTEP data block 3 field 7.
5. Items (3,7) to (3,13) were implemented starting with MSC Nastran 2004.0.4 and are not in previous versions.
6. The continuation lines may be omitted if not needed. If one of the continuation lines is needed, all proceeding continuation lines must be entered and at least one value per line is specified (no blank lines are allowed).
7. If an NLAUTO field is blank for the second and following subcases, the value will be assumed to be the same as that of the proceeding subcase for the same field. If this is not the behavior that is desired, be sure not to leave fields blank that should vary between the current and previous subcases.
8. Dynamics TINIT and TFINAL are determined from and TABLED1 entries. They are ignored if entered using NLAUTO. For statics, TINIT and TFINAL, if specified on NLAUTO override the values from NLPARAM.
9. Fixed time stepping is triggered by setting NSMAX equal to the number of steps (N) and IENHAN=2. It may also be triggered by using bulk data PARAM,MARCITER,N.
10. The time step is adjusted based upon:

if  $X < X1$              $Y(\text{calculated})/Y1$   
 if  $X1 < X < X2$          $Y(\text{calculated})/Y2$   
 if  $X2 < X < X3$          $Y(\text{calculated})/Y3$   
 if  $X3 < X < X4$          $Y(\text{calculated})/Y4$

where

Criterion	X	Y
1	Strain	Strain increment
2	Plastic strain	Plastic strain increment
3	Creep strain	Creep strain increment
4	Creep strain	Creep strain increment/Elastic strain
5	Stress	Stress increment
7	Strain energy	Strain energy increment
8	Temperature	Temperature increment
9	Displacement	Displacement increment
10	Rotation	Rotation increment
12	Stress	Stress increment/Stress
13*100+n	State variable n	Increment in state variable n

**NLBSH3D****Nonlinear 3-dimensional rotor loading**

Defines connectivity of a three-dimensional nonlinear load as a function of relative deflection and velocity of a grid pair which in turn is a function of rotor speed.

Format:

1	2	3	4	5	6	7	8	9	10
NLBSH3D	SID	GA	GB	Blank	LID-1	LID-2			

Describer	Meaning
SID	Nonlinear load set identification number. (Integer > 0; Required)
GA	Inner grid. (Integer > 0; Required)
GB	Outer grid. (Integer > 0; Required)
LID-1	Load vs deflection group ID of a LBSH3DG (Integer > 0; Required)
LID-2	Load vs velocity group ID of a LBSH3DG (Integer > 0; Required)

**Remarks:**

- NLBSH3D are selected with the Case Control command `NONLINEAR = SID`.
- In order to ensure correct results, GRID points GA and GB must satisfy the following three conditions:
  - They must both be grid points
  - They must be coincident
  - They must have parallel displacement coordinate systems

The program checks for the above conditions. If they are not satisfied, the program will terminate the run with an appropriate fatal message.
- Multiple NLBSH3D may reference the same SID but require different GA, GB grid pairs.
- Multiple NLBSH3D entries may refer to the same LID independent of SID.
- Non-linear load is calculated as a function of relative deflection and velocity, which in turn are functions of rotor speed, between GRID points GA and GB. Till version 2019FP1, non-linear load = load (deflection (rotor speed)).
- The loads are applied to the DOFs of GA and GB, based on the difference in displacement ( $u_b - u_a$ ) or velocity ( $v_b - v_a$ ) at the 2 GRID points, according to the tables referenced on the LBSH3DG entries. A positive difference in the displacement/velocity results in a positive force being applied to GA and a negative force being applied to GB
- GA and GB must have all 6 DOF in D-set. Both grids should not be dependent on any RBE or MPC and must not be constrained by an SPC. The program checks for active DOFs of both the grids. If they are not satisfied, the program will terminate the run with an appropriate fatal message.

**NLCYSYM** Defines Information to Perform Nonlinear Cyclic Symmetry Analysis using SOL 600

A limited capability is available to analyze structures with a geometry and a loading varying periodically about a symmetry axis. This capability is termed Nonlinear Cyclic Symmetry. This is done by developing a special set of tying constraints on the boundaries of the “pie-sliced” segment being analyzed. For more details on the type of structures that may be analyzed using this method and its restrictions, please see the Remarks.

Format:

1	2	3	4	5	6	7	8	9	10
NLCYSYM	ID	DC1	DC2	DC3	X	Y			
	ANG	TOL	IRB	LinFlag					

Examples:

NLCYSM		0.0	0.0	1.0	0.0	0.0	0.0		
	30.0		-1	-1					

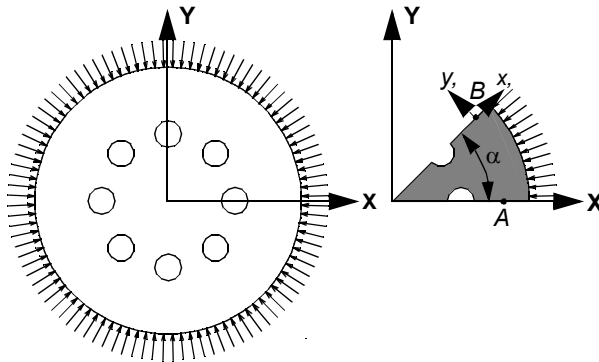
Describer	Meaning
ID	Identification number not presently used. (Integer or blank, Default = 0)
DC1	First component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
DC2	Second component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
DC3	Third component of the direction cosine of the symmetry axis. (Real; Default = 0.0)
X	X coordinate of a point on the symmetry axis. (Real; Default = 0.0)
Y	Y coordinate of a point on the symmetry axis. (Real; Default = 0.0)
Z	Z coordinate of a point on the symmetry axis. (Real; Default = 0.0)
ANG	Cyclic angle in degrees. (Real; no Default)
TOL	Cyclic symmetry tolerance. (Real; Default = 0.5 times the minimum element size)
IRB	Rigid body mode suppression flag. (Integer; Default = -1)
	-1 Automatically suppress rigid body mode.
	0 Do not suppress the rigid body mode.
	N Suppress rigid body mode at grid point N. (N > 0)
LinFlag	Linearization flag to be used if a cyclic symmetric structure consists of quadratic elements. (Integer; Default = 1)



Describer	Meaning
1	The outer boundary of the structure is described based on the corner nodes only. Multipoint constraints due to cyclic symmetry are not assigned to midside nodes. Instead, they are linearly tied to the corresponding corner nodes.
-1	The outer boundary of the structure is described using a quadratic field. Due to cyclic symmetry, full quadratic multipoint constraints are set up; they are assigned both to corner and midside nodes.

Remarks:

1. A full and “pie-shaped” structure to be analyzed using nonlinear cyclic symmetry shown below:



Looking at points and on this segment, the displacement vectors should fulfill:

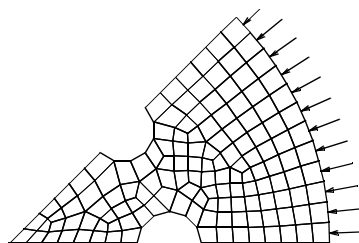
$$u'_B = u_A$$

which can also be written as:

$$u_B = Ru_A$$

where the transformation matrix R depends on the symmetry axis (which, in the example above, coincides with the global Z-axis) and the sector angle. The input for the nonlinear cyclic symmetry option consists of the direction vector of the symmetry axis, a point on the symmetry axis and the sector angle. The following items should be noted:

- a. The meshes do not need to line up on both sides of a sector (for example, see the following figure)



- b. Any shape of the sector sides is allowed provided that upon rotating the sector  $360/\alpha$  times about the symmetry axis over the sector angle will result in the complete model.
- c. The nonlinear cyclic symmetry option can be combined with standard contact. In this case, both sides of the cyclic symmetry sectors need to belong to the same contact body.
- d. The nonlinear cyclic symmetry option can be combined with global remeshing (not currently available).
- e. In a coupled thermo-mechanical analysis, the temperature is forced to be cyclic symmetric (TA=TB) (not currently available).
- f. A nodal point on the symmetry axis is automatically constrained in the plane perpendicular to the symmetry axis.
- g. Possible rigid body motion about the symmetry axis can be automatically suppressed.
- h. Cyclic Symmetry is valid for:
  - Primarily for continuum elements. However, the presence of beams and shells is allowed, but there is not connection of shells to shells across the symmetry plane, so the shell part can, for example, be a turbine blade and the volume part is a turbine rotor. The blade is connected to the rotor and if there are 20 blades, 1/20 of the rotor is modeled plus one complete blade.
  - Nonlinear static analysis including remeshing as well as coupled analysis
  - Valid for all analysis involving contact
  - Valid also for: eigenvalue analysis such as buckling or modal extraction, and linear transient dynamic analysis.
  - Cyclic Symmetry is invalid for pure heat transfer.
  - The contact status can be viewed. Grids tied across the symmetry plane will have a value of 2.

To prevent confusion, it must be emphasized that the cyclic symmetry feature described above is different than linear cyclic symmetry commonly used in modal analysis where physical quantities such as  $x_n$ , displacements, forces, stresses and temperature in the n-th segment are expanded in a Fourier

series with terms of the cyclic components,  $u^k$ , in the fundamental region, like:

$$x_n = \frac{1}{N}u^0 + \sqrt{\frac{2}{N}} \sum_{k=1}^K [u^{k,c} \cos(n-1)k\alpha + u^{k,s} \sin(n-1)k\alpha] + \frac{(-1)^{n-1}}{\sqrt{N}}u^{N/2}$$

where  $k$  is the harmonic order;  $N$  is the total number of sectors;  $\alpha$  is the fundamental inter-sector phase shift defined as  $2\pi/N$ ; and  $K$  is defined as:

$$K = \begin{cases} \frac{N-1}{2} & \text{if } N \text{ is odd} \\ \frac{N-2}{2} & \text{if } N \text{ is even} \end{cases}$$

There are considerable savings in both computing time and data storage associated with the use of the linear cyclic symmetry concept. Assuming a finite element model with a sector size of  $m$  degrees of freedom, a real-valued cyclic symmetry approach leads, in the worst case, to one eigenvalue problem of size  $m$  and  $(N-1)/2$  eigenvalue problems of size  $2m$ . A complex approach leads to  $N$  eigenvalue problems of size  $m$ ; while the full analysis leads to a single, but very costly, eigenvalue problem of size  $Nm$ .

Although linear cyclic symmetry can reduce the problem size greatly, it is restricted to linear analysis, and the sector must have its surface mesh on the symmetry planes to be identical on each side of the sector. The nonlinear cyclic symmetry implemented in SOL 600 can be used for nonlinear problems, such as contact, and the nodes do not need to line up on both symmetry planes of the sector.

## NLDAMP

## Damping Constants - SOL 600

Defines damping constants for nonlinear analysis when Marc is executed from SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLDAMP	EID1	EID2	ALPHA	BETA	GAMMA				

Examples:

NLDAMP	1	2000	.025	4.5	1.0				
--------	---	------	------	-----	-----	--	--	--	--

Describer	Meaning
EID1	First element for which the damping values will be used. (Integer $\geq 0$ ; Required)
EID2	Last element for which the damping values will be used. (Integer $\geq 0$ or blank)
ALPHA	Mass Matrix Multiplier. (Real; Default = 0.0)
BETA	Stiffness Matrix Multiplier. (Real; Default = 0.0)
GAMMA	Numerical Damping Multiplier. (Real; Default = 0.0)

Remarks:

1. This entry matches Marc's Damping definition.
2. NLDAMP is recognized only when Marc is executed from SOL 600.

## NLFREQ Frequency List for Nonlinear Harmonic Response

Defines explicit forcing frequencies for nonlinear harmonic response.

Format:

1	2	3	4	5	6	7	8	9	10
NLFREQ	ID	F1	F2	F3	F4	F5	F6	F7	
	F8	F9	-etc.-						

Examples:

NLFREQ	17	3.	7.	21.	14.				
--------	----	----	----	-----	-----	--	--	--	--

Describer	Meaning
ID	Identification number referenced by the NLFREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0)
F1...Fn	Forcing frequency values in cycles per unit time. (Real $\geq$ 0.0)

Remarks:

1. The frequencies may be specified in any order; they do not have to be in ascending or descending order.
2. Only one NLFREQ or NLFREQ1 entry with the same ID is allowed.

## NLFREQ1 Frequency List for Nonlinear Harmonic Response, Alternate Form 1

Defines a set of forcing frequencies for nonlinear harmonic response by specification of a starting frequency, frequency increment, and the number of increments/decrements desired.

Format:

1	2	3	4	5	6	7	8	9	10
NLFREQ1	ID	F1	DF	NDF					

Examples:

NLFREQ1	17	3.	2.	6					
---------	----	----	----	---	--	--	--	--	--

Descriptor	Meaning
ID	Identification number referenced by the NLFREQ field (field 5) of an NLHARM Bulk Data entry. (Integer > 0).
F1	First forcing frequency in the set. (Real $\geq$ 0.0)
DF	Frequency increment. See Remark 1. (Real $\neq$ 0.0; Required)
NDF	Number of frequency increments/decrements. (Integer > 0; Default = 1)

Remarks:

1. If DF is positive, it defines a frequency increment. In this case, the first excitation frequency will be F1 with all subsequent excitation frequencies being of larger value than F1. If DF is negative, it defines a frequency decrement. In this case, the first excitation frequency will still be F1, but now all subsequent excitation frequencies will be of smaller value than F1. In both cases, the initial conditions for a frequency are the response configuration at the previous excitation frequency. The initial conditions for the first frequency are zero.
2. Only one NLFREQ or NLFREQ1 entry with the same ID is allowed.

## NLHARM Parameters for Nonlinear Harmonic Response Control

Defines parameters for nonlinear harmonic response analysis.

Format:

1	2	3	4	5	6	7	8	9	10
NLHARM	ID	SUBFAC	NHARM	NLFREQ					

Examples:

NLHARM	41		3	15					
--------	----	--	---	----	--	--	--	--	--

Describer	Meaning
ID	Identification number referenced by the NLHARM Case Control command. (Integer > 0)
SUBFAC	Factor for capturing sub-harmonic response. See Remark 3. (Integer ≥ 1; Default = 1)
NHARM	The number of harmonics to include in the solution. See Remark 2. (Integer > 0)
NLFREQ	Identification number of the NLFREQ or NLFREQ1 entry specifying the forcing frequency list. (Integer > 0)

Remarks:

1. The NLHARM ID must be unique among all NLHARM entries.
2. The greater the degree of nonlinearity, the larger the number of harmonics required to find a solution (if one exists). The number of harmonics (NHARM) used may influence the ability of the algorithm to find solutions to the nonlinear problem posed. However, the larger the number of harmonics, the more computational effort is required to find a solution.
3. The response frequencies in linear harmonic analysis are the same as the forcing frequency. Permanent oscillations whose frequencies are a fraction of the forcing frequency ( $\frac{1}{2}$ ,  $\frac{1}{4}$ , ...) may occur in a nonlinear system; these oscillations are known as sub-harmonic response.

**NLHEATC** Defines Numerical Analysis Parameters for SOL 600 Heat Transfer Analysis

Format:

1	2	3	4	5	6	7	8	9	10
NLHEATC	ID	MLS	MRC	MMIN	NPOS	IASMB			
	TCHG	TEVAL	TERR						

Examples:

NLHEATC		9999	10	1	1	1			
	10.	999999.	20.						

Describer	Meaning
ID	Not presently used, leave blank.
MLS	Maximum number of load steps in the run. (Integer; Default = 4). [2,1]
MRC	Maximum number of recycles during an increment due to temperature dependent material properties. (Integer; Default = 3) [2,2]
MMIN	Minimum number of recycles during an increment. This value is forced even if convergence appears to occur in a fewer number of cycles. (Integer; Default = 1) [2,3]
NPOS	Nonpositive definite flag. (Integer; Default = 0) [2,7]
	0 Solution of nonpositive definite matrices will fail.
	1 Solution of nonpositive definite matrices is forced (computer time is greater if nonpositive matrices do not exist)
IASMB	Flag to assemble conductivity matrix. (Integer; Default = 0) [2,12]
	0 Conductivity matrix is not assembled each iteration
	1 Conductivity matrix is assembled each iteration
TCHG	Maximum nodal temperature change allowed. (Real; Default = 20.0) [3,1]
TEVAL	Maximum temperature change before properties are re-evaluated and matrices reassembled. (Real; Default = 100.0) [3,2]
TERR	Maximum error in temperature estimate used for property evaluation. It provides a recycling capability to improve accuracy for highly nonlinear heat transfer such as latent heat and radiation. (Real; Default = 0.0) [3,3]

Remarks:

1. Only one NLHEATC entry should be entered.
2. This entry maps to Marc's CONTROL history definition entry for heat transfer. [i,j] indicates the datablack and field of this Marc entry.



## NLMOPTS

## Nonlinear Multiple Options

Specifies nonlinear material options and composite options. The nonlinear material options are for advanced materials used in SOL400. The composite options (TSHEAR and INLAM, CPROJ) are applicable to all solution sequences in which Layered Solid Elements (PCOMPLS) are available, namely, SOL600, SOL400, SOL200 (analysis only), and all linear solution sequences between SOL101 and SOL112.

Format:

1	2	3	4	5	6	7	8	9	10
NLMOPTS	"CREEP"	valc1	valc2	valc3	valc4				
	"ASSM"	vala							
	"TSHEAR"	vals							
	"LRGSTRN"	valle							
	"HEMICUBE"	VALUE	NPIXEL		CUTOFF	FRACTION	FACCNT	FACTOL	
	"TEMPP"	valtd							
	"TEMGO"	vmaptg							
	"SPROPMAP"	PROPMAP	PROPBEH	DIRECT	THICKOP	IPRINT			
	"SPCRMPT"	vramp							
	"DEACTEL"	vald1	vald2						
	"ENTHALP"	valclu	valen1						
	"MAPTOL"	vmptol							
	"INLAM"	vcoord	cproj						

Examples:

NLMOPTS	CREEP	0							
	HEMICUBE	1	500		0	0.01			

Describer	Meaning
"CREEP"	<p>Keyword that indicates that creep is possible during one or more steps in the job. It also indicates the formulation for creep analysis. It should be noted that this keyword is only valid for elements with extended property extensions (PSLDN1, PSHLN1, etc.). (Character Default CREEP). Creep data should be entered using MATVP, not the CREEP bulk data entry.</p>
	<p>There are 2 kinds of creep models</p>
	<p>(a) Explicit Creep - ValC1 and ValC2 are used for this.</p>
valc1	<p>0 : Maxwell Model Creep (default) 1 : Explicit Viscoplastic Creep (only supported via user subroutine CRPLAW)</p>
valc2	<p>(Blank or 0) : (Default) 1: (Reserved)</p>
	<p>(b) Implicit Creep - ValC3 and ValC4 are used for this.</p>
valc3	<p>0 : Implicit Creep off 1 : Implicit Creep on</p>
valc4	<p>(Blank or 0) : elastic tangent (Default) 1: secant tangent 2: radial return</p>
	<p>Note that:</p>
	<p>(1) ValC3 = 0 means Explicit Creep (see ValC1). ValC3 = 1 means Implicit Creep.</p>
	<p>(2) ValC4 is only used in conjunction with Valc3=1. The recommended value for ValC4 is 1 (secant tangent).</p>
	<p>The CREEP option on NLMOPTS is generally used in conjunction with the following bulk data entries - the user can refer to each of the bulk data entries identified below for more information.</p>
NLSTEP	<p>The "CREEP" field on this entry indicates that creep is active for the particular step using the NLSTEP card. Note that the permanent creep flag through the NLMOPTS,CREEP keyword should be turned on along with the NLSTEP,CREEP keyword.</p>
MATVP	<p>The material data for creep are entered through this card. It should be noted that the data through MATVP is only for elements with extended property extensions. For elements without extended property extensions, material data for creep should be specified through the CREEP entry.</p>
MATEP	<p>Optional data for plasticity in combination with implicit or explicit creep is specified through this entry.</p>

Describer	Meaning
“ASSM”	Keyword indicating that the item following applies to assumed strain. See Remark 3. (Character; Default = See Remark 3.)
vala	ASSUME for assumed strain formulation D OFF for no assumed strain formulation.
“TSHEAR”	Keyword indicating that the item following applies to a parabolic shear distribution through the shell thickness (or ply thickness). See Remark 4.
vals	TSHEAR for parabolic distribution. (Character; Default = blank)
“LRGSTRN”	Keyword indicating that the item following applies to a formulation for large strain. (Default; Integer = 0)
valle	-1 No large strain formulation. 0 Mean normal return. 1 Hypoelasticity and additive plasticity with mean normal return. 2 Hyperelasticity and multiplicative plasticity with radial return.
“HEMICUBE”	Keyword to select the view factor calculation method. See Remarks 2. and 7.
	Field 3 VALUE=0 - Use Nastran finite difference, contour integration, or Gaussian integration method (Default). =1 - (HEMI) Use pixel based modified hemicube method. This method is not available for axisymmetric view factor calculations.
	Field 4 NPIXEL - Enter the number of pixels (Default = 500).
	Field 6 CUTOFF - Enter the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored (Default = 0).
	Field 7 FRACTION - Enter the fraction of the maximum view factors that is to be treated implicitly. View factors values smaller than this cutoff are treated explicitly (Default = 1.0E-2).
	Field 8 FACCNT - Set 1 to activate explicit treatment of reflection matrix. Default = 0)
	Field 9 FACTOL - Tolerance to be used on the above iteration on the Poljak equations.
“TEMPP”	Keyword indicating temperature distribution across the thickness of advanced heat shell elements. Internal invisible scalar points will be created to store these extra variables. LINE and QUAD options require shell elements to have full 3-D thermal material data. When the temperature results of these internal points are applied to calculating thermal loading effect in a separate mechanical analysis, this keyword must also be specified in the structural model (See Remarks 12. ,13. and 14.).
valtd	CONS Constant distribution. (Default)

Describer	Meaning
	LINE      Linear distribution. (Character)
	QUAD      Quadratic distribution.
“TEMGO”	Keyword indicating user request identified in .f06 file, the mapping of user supplied and internally generated grids for the linear and quadratic thermal shell elements. See Remark 8.
vmaptg	NO          Do not show grid mapping. (Character; Default)
	YES          Show mapping.
“SPROPMAP”	Keyword controlling automated inclusion of nonlinear property options for SOL 400. See Remark 9.
PROPMAP	(Integer $\geq -1$ and $\leq 3$ ; Default 0)
	-1          No mapping considered
	0          Map to full integration elements associated with nonlinear property extension entries when necessary
	1          Map to reduced integration elements associated with nonlinear property extension entries when necessary
	2          Map to full integration elements associated with nonlinear property extension entries when possible
	3          Map to reduced integration elements associated with nonlinear property extension entries when possible
	<b>Note:</b> The nonlinear property extension entries refer to: PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2 and PSLDN1 entries. To ensure that the automatic mapping is unique for an element ID, whenever advanced elements are detected / specified by the user, ELTPRT will be automatically called.
PROPBEH	Property behavior flag. Valid values are 1D, 3D, PLST, PSTR, AXI. (Default = None for 1D, 3D. PLST for 2D; Character)
DIRECT	Layer direction flag. Valid values; 1, 2, 3. (Integer > 0; Default 1)
THICKOP	For 2D solids: out of plane thickness. (Real $\geq 0.0$ ; Default 1.0)
IPRINT	Print flag for generated property flags.
	0          Blank (Default)
	1          Print
	2          Punch
	3          Print and Punch
“SPCRMPT”	Keyword indicating heat transfer SPC is to be applied instantaneously or by linear step. (Character Default SPCRMPT)
vramp	0          SPC applied instantaneously. (Integer or blank; Default = blank)
	1          SPC applied by linear step.

Describer	Meaning
“DEACTEL”	Keyword controlling default behavior and output diagnostics for element deactivation
vald1	Select default deactivation rules for each physics pass in the analysis. See Remark 10. -1 Default deactivation rules are always ignored. (Default in single physics analysis) 0 Default deactivation rules are always applied. (Default in coupled analysis) 1 Default deactivation rules are applied in a mechanical pass only. 2 Default deactivation rules are applied in a thermal pass only.
vald2	Select diagnostic output behavior for active and deactivated elements. 0 A summary of all deactivated elements is printed. If no elements get deactivated output is skipped. (Default) 1 A detailed list of all deactivated elements is printed. If no elements get deactivated output is skipped. 2 A summary of all deactivated and all remaining active elements is printed. 3 A detailed list of all deactivated and all remaining active elements is printed.
“ENTHALP”	Keyword controlling the scheme used to form the capacitance matrix and the enthalpy vector for elements with nonlinear property extensions in transient heat transfer. See Remark 10.
valclu	Select option to form copacitance matrix for elements with nonlinear property extensions. 0 Use same scheme as that for linear elements without property extensions 1 Use consistent capacitance matrix 2 Use coupled capacitance matrix 3 Use lumped capacitance matrix
valen1	Select option to form enthalpy vector for elements with nonlinear property extensions 0 Form enthalpy using automatic scheme. (Default) 1 Form enthalpy explicitly through time integration for material behavior 2 Form enthalpy via multiplication of temperature with capacitance
“MAPTOL”	Keyword controlling the tolerance used to check if a structural node lies inside a thermal element.
vmptol	Value of MAPTOL. (Real > 0.0; Default = 0.2)
INLAM	(Character; Default Blank)
vcoord	Selects coordinate system for interlaminar stress output. Blank Using the original coordinate system for interlaminar stress output

Describer	Meaning
cproj	BOTT      Using the ply coordinate system of bottom layer of the interlaminare
	TOP        Using the ply coordinate system of above layer of the interlaminare
	Activates projection system for projecting X-axis of CORDM onto layer plane. See remark 5.
	OFF        Does not project (Default).
	ON         Projection scheme is activated.

Remarks:

1. The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
2. The material and property related keywords associated with the entry (CREEP, ASSM, TSHEAR, LRGSTRN, TEMPP and ENTHALP) are only applicable to elements associated with PAXISYM, PBARN1, PBEMN1, PRODN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMPLS, and PCOHE entries. Other keywords (SPCRMPT, HEMICUBE, DEACTEL, MAPTOL) are more generally applicable. Keyword SPCRMPT applies to all SPC temperature boundary conditions, HEMICUBE applies to radiation boundary conditions and DEACTEL applies to all elements on the job.
3. The setting of ASSUMED STRAIN in SOL 400 is automatic for the following three elements with some conditions.
  - a. 4 nodes full integration Plane Stress (PLPLANE + PSHLN2 with PSTRS -- BEH4=PSTRS, INT4=L);
  - b. 4 nodes full integration Plane Strain (PLPLANE + PSHLN2 with PSTRN -- BEH4=PLSTRN, INT4=L); and
  - c. 8 nodes full integration HEXA (PSOLID + PSLDN1 -- BEH8=SOLID, INT8=L,)

The application conditions are summarized here.

OFF                      Assumed strain always turned off.

- ASSUMED      Assumed strain will be turned on manually for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:
- a. Elastic / Elasto-Plastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1). This includes isotropic and non-isotropic materials.
  - b. Elastic / Elasto-Plastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1). This includes isotropic and non-isotropic materials.

Assumed strain flag will be turned off for all materials using the Multiplicative Formulation (NLMOPTS,LRGSTRN,2).

- Default      Assumed Strain will be turned on automatically for 3 element types (PSTRS, PSTRN, PSLDN1) under the following cases:
- a. Isotropic Elastic materials using small strain formulation (i.e., PARAM,LGDISP,-1 / NLMOPTS,LRGSTRN,-1)
  - b. Isotropic Elastic materials using large strain additive formulation (i.e., PARAM,LGDISP,1 / NLMOPTS,LRGSTRN,0 or 1)

Note that this turns on Total Lagrange Formulation currently by default for the isotropic elastic materials.

Assumed Strain will be turned off automatically for the following cases:

- a. Any material that uses Updated Lagrange in either Additive or Multiplicative Formulation
  - b. Any non-isotropic material
4. TSHEAR allows a parabolic shear distribution for the BEH4=DCT, INT4=L or LRIH (shell) elements, and BEH8 or BEH20=SLCOMP, INTi=L, Q (Layered Solid) or ASTN (Layered Solid Shell) elements.

5. In Figure 9-102, plane ABCD is the layer plane of interest.  $Z_m$  is normal to the layer plane (or is the thickness direction).  $X_m$  is the projection of the X axis of the MCID coordinate system on the layer plane.  $Z_m \times X_m$  gives the  $Y_m$ . Angle  $\theta$  is measured counter-clockwise from  $X_m$  axis about the  $Z_m$  axis.

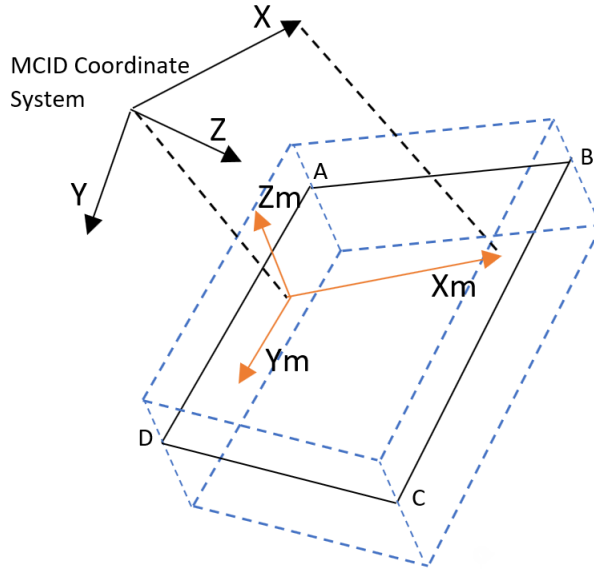


Figure 9-102 Projection Scheme

6. NLMOPTS,LRGS,valle can be used separately or in conjunction with the PARAM,LGDISP option. Two separate cases can be identified:
- Only NLMOPTS,LRGS specified. In this case, valle = -1 and valle = 0 mean the same, i.e., no large strain flags are turned on for the applicable elements identified in remark 2. Valle = 1 flags the additive framework and valle = 2 flags the multiplicative framework.
  - Both NLMOPTS,LRGS and PARAM,LGDISP,N ( $N > -1$ ) specified. In this case, valle = -1 supercedes the PARAM,LGDISP option and turns off all large displacement / large strain flags for the applicable elements identified in remark 2. Also, valle = 0 behaves the same way as valle = 1 (i.e., for the applicable elements, both options flag the additive framework).
7. Definition of the radiation exchange matrix as in the *MSC Nastran Thermal User's Guide*, Eq. 0-15.

$$R = \sigma[A\varepsilon - A\alpha(A - F(I - \alpha))^{-1}F\varepsilon]$$

in which the reflection matrix is:

$$[A - F(I - \alpha)]^{-1}$$

Note that the reflection term is costly, since it involves the factorization of a dense matrix.



For the SOL 400 Newton’s method, the previous exchange matrix multiplied by a function of the temperature is added to the stiffness. If the exchange matrix is dense, which is generally the case, the sparse stiffness matrix consequently also becomes dense, and the factorization of the stiffness matrix becomes much more expensive.

The input options are as follows. All options with the exception of the “faccnt” option, are available in SOLs 153, 159, and 400. The “faccnt” option is only available in SOL 400:

hemi, value, npixel, cutoff, fraction, faccnt, factol

where

- value            1 to flag hemicube method. Default is 0. The following options are available only if value = 1.
- npixel           Number of pixels per quarter section, where total number of pixels =  $(2 \cdot \text{npixel})^2$ . Default is 500.
- cutoff            Factor below which viewfactors will be set to zero. This one is applied during the last viewfactor calculation. It results in a less dense exchange matrix and therefore a faster calculation of the reflection term and of the stiffness matrix factorization. Default is 0.0.
- fraction        Factor below which radiation exchange matrix terms are not added to the stiffness matrix. For each equation in the radiation exchange matrix, the fraction is multiplied by the diagonal term, and all terms in the equation less than this value is not added to the stiffness matrix. This procedure does not affect the density of the radiation exchange matrix or the cost of calculating it, including the reflection term, but reduces the density of the stiffness matrix and the cost of factorizing the stiffness matrix. Note that the full radiation exchange matrix still is used to calculate fluxes. Default is 0.01.
- faccnt           Set to 1 to activate explicit treatment of reflection matrix. Default is 0. In this procedure, the radiation exchange matrix is constructed to be:  

$$R = \sigma[A\varepsilon - A\alpha F\varepsilon]$$

Note that the expensive reflection term is absent. This reduced form of the radiation exchange matrix is added to the stiffness matrix. Since the reflection matrix never is calculated and factorized, the calculation of the radiation exchange matrix is significantly cheaper. Since this reduced radiation exchange matrix is less dense, the factorization of the stiffness matrix also is significantly faster.

For the flux calculation, an iterative procedure is used based on iterating towards a solution simultaneously satisfying both Poljak equations in the *MSC Nastran Thermal User’s Guide*, Eqs. 6-11 and 6-12, respectively.

$$A\{q\}_e^{\text{IN}} = [F]\{q\}_e^{\text{OUT}}$$

$$\{q\}_e^{\text{OUT}} = \sigma[\varepsilon]\{u_e\}^4 + [I - \varepsilon]\{q\}_e^{\text{IN}}$$

The value of “facnt” also can be set to the maximum allowable number of iterations to be used in this procedure. If the value is set to 1, the maximum allowable number is internally set to 100. This procedure currently is available only in SOL 400 and is not available for wavelength-dependent emissivities.

The “cutoff” and “fraction” parameters can be specified concurrently with this option.

factol tolerance to be used on the previous iterative procedure on the Poljak equations.

8. TEMGO,YES results in the following type of print.

USER GRID POINT(TOP) MAPPING TO BOT/MID INTERNAL GRID										
TOP	1	2	3	4	5	6	7	8	9	10
BOT	101000001	101000002	101000003	101000004	101000005	101000006	101000007	101000008	101000009	101000010
MID	0	0	0	0	0	0	0	0	0	0
...										

9. The “SPROPMAP” keyword provides a convenient option to automatically flag secondary properties like PBARN1, PBEMN1, PRODN1, PSHEARN, PSHLN1, PSHLN2, and PSLDN1. Note that these secondary property entries expose the user to a set of sophisticated 2-D continuum and 3-D beam, shell and continuum elements in SOL 400. The rules governing the generation of the additional properties are many and are a function of the problem dimension, material type, and procedure. All these rules have been incorporated into the automatic generation option. These are briefly summarized in the table below for the default SPROPMAP = 0 case:

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PBARN1	PBARL	1-D	MAT4	Note 4	Tapered Sections
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PBEMN1	PBEAML	1-D	MAT4	Note 4	
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PRODN1	PROD	1-D	MAT4	Note 4	J,C
			MATS1	Note 1	
			MATEP		
			MATF		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHEARN	PSHEAR	3-D	MAT4	Note 4	F1,F2
			MAT8		
			MATS1	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHLN1	PSHELL	3-D	MAT4	Note 4	TS/T, nondefault Z1 and Z2, 121/T <sup>3</sup>
			MAT5	Note 4	
			MATS1	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
MATVP	Note 3				

Secondary Property	Primary Property	Dimension	Material	Notes	Unsupported Features of Primary Entry
PSHLN1	PCOMP/ PCOMPG	3-D	MAT4	Note 4	FT, GE, LAM options other than BLANK and SYM, SOUTi
			MAT5	Note 4	
			MAT8		
			MATS1	Note 1	
			MATS2	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 2	
			MATVE		
			MATVP	Note 3	
PSHLN2	PLPLANE	2-D	MAT4	Note 4	
			MAT5	Note 4	
			MATG	Note 5	
			MATS1	Note 1	
			MATS3	Note 1	
			MATS8	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA	Note 6	
			MATVE		
			MATVP	Note 7	
PSLDN1	PSOLID	3-D	MAT4	Note 4	IN, ISOP, FCTN
			MAT5	Note 4	
			MATS1	Note 1	
			MATEP		
			MATF		
			MATORT		
			MATSMA		
			MATVE		
MATVP					

Note:	1.	Only when NLMOPTS, LRGSTRN, 1 is flagged or HGENPLAS is provided NLSTEP entry for coupled analysis.
	2.	For shape memory materials, define through MATSMA only the thermo-mechanical model is available.
	3.	For creep, define through MATVP, VALC=0 must be set on NLMOPTS for explicit formulation.
	4.	Only when phase changes are flagged.
	5.	Gasket materials for BEH4=COMPS or AXCOMP,INT4=L.
	6.	For shape memory materials, define through MATSMA and BEH=PLSTRS only the thermo-mechanical model is available.
	7.	For creep, define through MATVP and BEH=PLSTRS VALC=0 must be set on NLMOPTS for explicit formulation.
	8.	The "SPROPMAP" does not support MATDIGI.

In the previous table, the Secondary Property (Column 1) is automatically flagged depending on the Primary Property (Column 2) and the Primary/Secondary Materials (Column 4). The Comments column (Column 5) indicates any special conditions that are followed by the program while doing this mapping. The Unsupported Features column (Column 6) indicates the list of options on the Primary Property entry that are not supported by the Secondary Property.

It should be noted that the Secondary Property additions offer a wide range of element formulations that primarily cater to nonlinear extension but can also serve for linear applications. By default, the PROPMAP field is set to 0 and maps to the full integration elements of the secondary property only if necessary. For a less memory intensive element, the PROPMAP field can be manually set to 1 and this then maps to the equivalent reduced integration element of the secondary property only if available and necessary. PROPMAP field can be manually set to 2 or 3 when secondary properties need to be tagged to all elements in the model. In this case, the special conditions given in the 5th column above are ignored and the mapping is carried out for both linear and nonlinear materials (if allowed). PROPMAP = 2 maps to full integration elements whenever possible and PROPMAP = 3 maps to reduced integration elements whenever possible.

- a. Property mapping is considered for all possible elements (i.e., PROPMAP = 0/1 is automatically converted to PROPMAP = 2/3) for the following conditions:
  - When IDAMP (damping scheme specified through the NLSTEP entry) > 0 is used for static analysis. Note that the damping scheme is only available for elements with nonlinear property extensions.
  - When CRITID (user criteria specified through the NLSTEP entry) and the associated TABSCTL entry refer to displacement / stress / strain criteria for all elements in the model. Note that the user criteria are only supported for elements with nonlinear property extensions.

- When a small displacement analysis is flagged for thermo-mechanical coupled analysis specified through SUBSTEP or for real-time thermo-mechanical chained analysis specified through TEMP(Load,HSub,HStep,HTime). Note that for real-time temperatures to be transferred from the heat pass to the structural pass, all elements need to be processed in the nonlinear program flow and this is accomplished by flagging all possible elements with nonlinear property extensions.
- b. Property mapping is provided for elements that are associated with special procedures / flags:
- Elements whose grids are specified on the VCCT Bulk Data entry. VCCT is only supported for grids of elements with nonlinear property extensions.
  - Elements that are specified on the IPSTRN / ISTRESS commands. These are only supported for elements with nonlinear property extensions.
  - Elements that are specified on the BOLT1 entry. BOLT1 is only supported for elements with nonlinear property extensions.

It should be noted also that while the NLMOPTS,SPROPMAP option is powerful and allows a user-friendly way to map to advanced elements, it does not provide a substitute for all cases. Defaults are provided for the most common cases on the NLMOPTS,SPROPMAP entry itself. This includes PROPBEH - especially useful to distinguish between plane strain and plane stress, DIRECT - the layer thickness direction for gaskets, THICKOP - the out-of-plane thickness for the planar case. It should be noted that these defaults are global defaults and apply to all the elements for which the secondary properties are added. If such defaults are not applicable (for e.g., different out-of-plane thicknesses), it is the user's responsibility to add individual secondary property entries. No mapping to advanced nonlinear elements takes place for PSHELL entries with MID greater than 100,000,000 to avoid conflicts with internally generated material IDs.

IPRINT can be used to get additional information on the secondary properties that have been added. Default is 0 in which case, there is no print-out. IPRINT = 1 allows the print-out in the .f06 file, IPRINT = 2 allows the print-out in the .pch file, IPRINT = 3 allows print-out in both the .f06 and .pch files.

Example 1: 8 noded CHEXA, PSOLID, MAT1	
SPROPMAP	Added Secondary Property
-1	No mapping considered
0	No mapping done - not necessary
1	No mapping done - not necessary
2	PSLDN1: C8, BEH8 = SOLID, INT8 = L, BEH8H = SOLID, INT8H = L
3	PSLDN1: C8, BEH8 = SOLID, INT8 = LRIH, BEH8H = SOLID, INT8H = LRIH

Example 2: 4 noded CQUAD4, PSHELL, MATF	
SPROP MAP	Added Secondary Property
-1	No mapping (note that MATF will not work in this case)
0	PSHLN1: C4, BEH4 = DCT, INT4 = L, BEH4H = DCT, INT4H = L
1	PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L
2	PSHLN1: C8, BEH8 = DCT, INT4 = L, BEH4H = DCT, INT4H = L
3	PSHLN1: C4, BEH4 = DCT, INT4 = LRIH, BEH4H = DCT, INT4H = L

- The **vald1** value in “DEACTEL” determines in which physics pass the default deactivation rules are applied. If **vald1** is positive the deactivation rules are applied in the specified pass only and in all other passes the defaults are switched off, meaning that in those passes all elements remain active unless a DEACTEL Bulk Data input deactivates some of them.

The default deactivation rules are summarized in the following table:

Element Types	By Default Deactivated in Pass
Connector elements CWELD, CFAST, CSEAM	Thermal
Rigid body elements Rxxx	Thermal
CONMi and CMASSi elements	Thermal
Interface elements CIFxxx	Thermal
Surface elements CHBDYx	Mechanical
Heat convection elements CONVx	Mechanical
Radiation boundary elements RADBC	Mechanical
CDAMP4 elements originating from CMB or CMS fields in BCBODY input	Mechanical

The default for elements of all other types is that they are active in every pass.

- The “ENTHALP” keyword is applicable to elements associated with the nonlinear property extensions PRODN1, PSHEARN, PSHLN1, PSLDN1 and for axisymmetric heat transfer shells specified through PAXISYM. It provides different options to form the capacitance matrix and enthalpy vector for these elements through **valclu** and **valenl** respectively.

The **valclu** value allows the choice of 3 different capacitance formulations: consistent, coupled (average of consistent and lumped) and lumped. The default value of **valclu = 0** allows the nonlinear elements to follow the same scheme as that for linear elements; lumped for rods and 3D shells, coupled for axisymmetric shells and linear hexes, pentas, tetras and consistent for quadratic hexes, pentas and tetras. Values of **valclu > 0** allow the user to pick specific capacitance formulations for all elements.

The **valenl** value allows the user the choice of 2 different enthalpy vector formulations: time integration of the enthalpy at each integration point or product of the nodal temperature with the capacitance. The default value of **valenl = 0** allows the program to pick automatically: use the time integration option for nonlinear specific heat / latent heat, use the nodal product option for linear specific heat. Values of **valenl > 0** allow the user to pick the approach. Note that for nonlinear problems, the time integration approach is more accurate.

12. To compute the thermal loads of 3D thermal shells in a separate mechanical run, a consistent temperature distribution type must be specified in mechanical models, i.e. LINE must be defined if the thermal model has a linear temperature distribution using LINE, or QUAD must be specified for the thermal model with a quadratic temperature distribution defined by QUAD. If LINE or QUAD is specified, the program will switch the structural shell elements to advanced elements automatically by generating PSHLN1 entries internally if no PSHLN1 or PSHLN2 entries are specified in the original model.
13. MSC NASTRAN thermal analysis does not differentiate between top and bottom faces while applying loads or boundary conditions on shell, even though the top and bottom temperatures may be different through specifying TEMPP = LINE or QUAD.
14. TEMPP keyword is supported in coupled analysis or in separate runs of thermal and mechanical analyses. This keyword is not supported in chained thermal and mechanical analyses.



**NLOUT****Selects Additional Nonlinear Output Quantities as Referenced By NLSTRESS Case Control Command**

Selects additional nonlinear output quantities as referenced by NLSTRESS Case Control Command in SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
NLOUT	ID	AIO1	AIO2	AIO3	AIO4	AIO5	AIO6	AIO7	
	AIO8	AIO9	etc.						
	ESV	SV2	SV3	etc.					

Example:

NLOUT	1	TOTTEMP							
-------	---	---------	--	--	--	--	--	--	--

NLOUT	1	TOTTEMP							
	ESV	SV2	SV3						

Describer	Meaning
ID	Identification number of a NLOUT entry. Selected by the NLOUT keyword on the NLSTRESS Case Control command. (Integer > 0)
AIO <sub>i</sub>	Additional Output request. See Remark 3. (Character)
ESV	Keyword indicating the following are user state variables, it is optional.
SV <sub>i</sub>	State variable names selected for output.

Remarks:

1. The keywords can appear in any order.
2. Any item selected that is inappropriate for the element will be ignored.
3. The request codes, meaningful only to elements who refer to PRODN1, PBARN1, PBEMN1, PSHLN1, PSHLN2, PSLDN1, PLCOMP, PCOMLS, or PCOHE entries, are as follows:

Additional Output Code Keywords	
Keyword	Description
CCASTRSS	Components of Cauchy Stress
CTOTSTRN	Components of Total Strain
CELASTRN	Components of Elastic Strain
CPLASTRN	Components of Plastic Strain

Additional Output Code Keywords	
Keyword	Description
CCRPSTRN	Components of Creep Strain
CTHMSTRN	Components of Thermal Strain
TSTRNPS	Thickness Strain for Plane Stress
MAJESTRN	Major Engineering Strain
MINESTRN	Minor Engineering Strain
CURVOL	Current Volume
ORGVOL	Original Volume
TOTTEMP	Total Temperature
INCTEMP	Incremental Temperature
EQVMSTRS	Equivalent von Mises Stress
EQSTRSA	Equivalent Stress/Yield Stress Ratio
EQELSTRN	Equivalent Elastic Strain
EQPLSTRN	Equivalent Plastic Strain
EQCRSTRN	Equivalent Creep Strain
TTSTRNED	Total Strain Energy Density
ELSTRNED	Elastic Strain Energy Density
PLSTRNED	Plastic Strain Energy Density
PLSTRNRT	Plastic Strain Rate
ILNMSTRS	Interlaminar Normal Stress. See Remark 5.
ILSHSTRS	Interlaminar Shear Stress. See Remark 5.
ILSHTKCX	X-component of the interlaminar shear stress for thick composite shells.
ILSHTKCY	Y-component of the interlaminar shear stress for thick composite shells
CSTRSCRD	Components of Stress Preferred System
GSKTCLST	Gasket Pressure
GSKTCLSR	Gasket Closure
PGSKTCLS	Plastic Gasket Closure
FAILINDX	Failure Index (%)
TOTVSV1	Total Value of First State Variable
TOTVSV2	Total Value of Second State Variable
TOTVSV3	Total Value of Third State Variable

Additional Output Code Keywords	
Keyword	Description
EQPHSTRN	Equivalent phase transformation strain
EQTWSTRN	Equivalent TWIN strain
EQTPSTRN	Equivalent TRIP strain 75
CPHSTRN	Phase transformation strain tensor
VOLFMART	Volume fraction of Martensite

- User state variable name following the ESV keyword can be the default nominal name or user defined name in UDSESV.
- ILNMSTRS and ILSHSTRS are stress tensors formed in the basic coordinate system, their principal directions are the directions of stress vector of interlaminar normal and shear stresses respectively.

More precisely, the tensor components  $\sigma_{ij}$  satisfy below equation:

$$(\sigma_{ij} - \delta_{ij}\sigma)n_j = 0$$

where;

$n_j$  is the stress vector direction,

$\sigma$  is the normal of stress vector, and

$\delta_{ij} = 0$  when  $i \neq j$  and  $\delta_{ij} = 1$  when  $i = j$ .

## NLOUTUD

### User Defined Output Requests For Nonlinear Explicit Analysis.

User defined output requests for elements or Lagrangian grid points. Use in SOL700 only.

Format:

1	2	3	4	5	6	7	8	9	10
NLOUTUD	NID	GROUP	UNAME	DTOUT	STEPS	OUTNAME			
+	EL1/G1	EL2/G2	Eli/Gli						

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myuds 'SCA.MDSolver.Obj.Uds.Dytran.InitOut '
```

*In Bulk Data:*

NLOUTUD	4	MYUDS	EEXOUT	0.01		OUTPUTR1			
+	1	2	3	4					

Field	Contents
NID	Unique output number. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	Name User subroutine name associated with the entry. (Character; default=EEXOUT)  EEXOUT: Output for elements  GEXOUT: Output for Lagrangian grid points
DTOUT	Output time interval. (Real > 0.0; default=blank)
STEPS	Output step interval. (Integer > 0; default=blank)
OUTNAME	Name for output request. (Character; required.)
Eli	Element id for user output. (Integer > 0; required)
Gi	Grid point id for user output. (Integer > 0; required)

Remarks:

1. Only can be used for SOL 700.
2. UNAME can be EEXOUT or GEXOUT.
3. Only one of DTOUT and STEPS is available.

4. When UNAME is set to EEXOUT, element id's are required for user output and when UNAME is set to GEXOUT, grid point id's are required for user output.
5. The file names that will be generated are:  
`{jobname}_{OUTNAME}_{cycle number}.[THS,ARC]`

## NLPARM

## Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy. NLSTEP is the preferred option for SOL 400.

Format:

1	2	3	4	5	6	7	8	9	10
NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	MAXBIS				MAXR		RTOLB	MINITER	

Example:

NLPARM	15	5		ITER					
--------	----	---	--	------	--	--	--	--	--

Describer	Meaning
ID	Identification number. (Integer > 0)
NINC	Number of increments. See Remark 16. (Integer > 0)
DT	Incremental time interval for creep analysis. See Remark 3. (Real $\geq$ 0.0; Default = 0.0 for no creep.)
KMETHOD	Method for controlling stiffness updates. See Remark 4. (Character = "AUTO", "ITER", "SEMI", "FNT", or "PFNT"; Default = "AUTO" for SOL 106, "AUTO" for SOL 400 with non-contact analysis, and "FNT" for SOL 400 with contact analysis.)
KSTEP	Number of iterations before the stiffness update for ITER method. See Remarks 5. For the FNT and PFNT usage of KSTEP, please see Remark 19. (Integer $\geq$ -1; Default = 5 for SOL 106 and 10 for SOL 400)
MAXITER	Limit on number of iterations for each load increment. See Remark 6. (Integer $\neq$ 0; Default = 25)
CONV	Flags to select convergence criteria. See Remarks 7., 21., and 22. (Character = "U", "P", "W", "V", "N", "A" or any combination; Default = "PW". (See Remark 4 for additional default comment.)
INTOUT	Intermediate output flag. See Remark 8. (Character = "YES", "NO", "ALL" or Integer > 0 for SOL 400 only; Default = NO)
EPSU	Error tolerance for displacement (U) criterion. See Remarks 4., 16., 17. and 20. (Real > 0.0; Default = 1.0E-2)
EPSP	Error tolerance for load (P) criterion. See Remarks 4., 16. and 17. (Real > 0.0; Default = 1.0E-2)
EPSW	Error tolerance for work (W) criterion. See Remarks 4., 16., 17. and 20. (Real > 0.0; Default = 1.0E-2)

Describer	Meaning
MAXDIV	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. See Remark 9. (Integer $\neq$ 0; Default = 3)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 10. (Integer $\geq$ 0; Default = MAXITER for all methods except PFNT. For PFNT, Default = 0)
MAXLS	Maximum number of line searches allowed for each iteration. See Remark 11. (Integer $\geq$ 0; Default = 4 for all methods except PFNT. For PFNT, Default = 0)
FSTRESS	Fraction of effective stress ( $\bar{\sigma}$ ) used to limit the subincrement size in the material routines. See Remark 12. (0.0 < Real < 1.0; Default = 0.2)
LSTOL	Line search tolerance. See Remark 12. (0.01 < Real < 0.9; Default = 0.5)
MAXBIS	Maximum number of bisections allowed for each load increment. See Remark 13. (-10 $\leq$ MAXBIS $\leq$ 10; Default = 5 except for MAXITER < 0; Default = 0 if MAXITER < 0)
MAXR	Maximum ratio for the adjusted arc-length increment relative to the initial value. See Remark 14. (1.0 $\leq$ MAXR $\leq$ 40.0; Default = 20.0)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 15. (Real > 2.0; Default = 20.0)
MINITER	Minimum number of iterations for each increment, SOL 101 with contact and SOL 400 only. (Integer > 0; Default = 1; In contact analysis, Default = 2) When high accuracy is required, it is also recommended to set MINITER = 2.

Remarks:

1. The NLPARM entry is selected by the Case Control command NLPARM = ID. Each solution subcase requires an NLPARM command.
2. In cases of static analysis (DT = 0.0) using Newton methods, NINC is the number of equal subdivisions of the load change defined for the subcase. Applied loads, gravity loads, temperature sets, enforced displacements, etc., define the new loading conditions. The differences from the previous case are divided by NINC to define the incremental values. In cases of static analysis (DT = 0.0) using arc-length methods, NINC is used to determine the initial arc-length for the subcase, and the number of load subdivisions will not be equal to NINC. In cases of creep analysis (DT > 0.0), NINC is the number of time step increments.
3. For creep analysis, the unit of DT must be consistent with the unit used on the CREEP entry that defines the creep characteristics. Total creep time for the subcase or step is DT multiplied by the value in the field NINC; i.e., DT\*NINC. For SOL 400 with advanced nonlinear elements (elements with PSLDN1, PSHLN1, etc.) for creep analysis, if the total time does not equal 1.0, then the NLSTEP Bulk Data entry must be used. For SOL 600 DT is ignored except for creep analyses.

4. The stiffness update strategy is selected in the KMETHOD field.
  - If the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 9. and 13. for diverging solutions.
  - If the SEMI option is selected, the program for each load increment (i) performs a single iteration based upon the new load, (ii) updates the stiffness matrix, and (iii) resumes the normal AUTO option.
  - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if  $KSTEP \leq MAXITER$ . However, if  $KSTEP > MAXITER$ , stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and  $KSTEP = MAXITER$ . The "U" convergence test is not used if ITER-1 is selected.
  - If the FNT option is selected, the program will use the full Newton iteration method for which the stiffness matrix will be updated at every iteration. FNT option is available for SOL 400 only. For SOL 106, please use "KMETHOD=ITER and KSTEP=1" instead. In comparison with the PFNT method, the defaults for FNT are EPSU=0.01, EPSW=0.01 and MAXLS=4. See Remark 19.
  - If the PFNT option is selected, the program will use the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are internally set as EPSU=-0.01, EPSW=-0.01, and MAXLS=0. The PFNT method is available for SOL 400 only. See Remark 19.
5. For AUTO and SEMI options, the stiffness matrix is updated on convergence if KSTEP is less than the number of iterations that were required for convergence with the current stiffness.
6. The number of iterations for a load increment is limited to MAXITER. If the solution does not converge in MAXITER iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e., MAXBIS is attained or MAXBIS = 0) and MAXDIV is positive, the best attainable solution is computed and the analysis is continued to the next load increment. One best solution is computed for SOL 106 and 4 best solutions are computed for SOL 400. The analysis is terminated if the solution still diverges. If MAXDIV is negative, the analysis is terminated immediately.

For SOL 400 only, MAXITER can be negative. If MAXITER is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each load increment. The default for MAXBIS = 0, if MAXITER < 0. Also, for SOL 400, the value of MAXITER for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.

7. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W,



then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if CONV = 'blank' the code will use a default of "UPW" if no contact analysis and "PV" if a contact analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 21. For A, see Remark 22.

8. INTOUT controls the output requests for displacements, element forces and stresses, etc. YES or ALL must be specified in order to be able to perform a subsequent restart from the middle of a subcase.

INTOUT	Output Processed
YES	For every computed load increment.
NO	For the last load of the subcase or step.
ALL	For every computed and user-specified load increment.

- For the Newton family of iteration methods (i.e., when no NLPCI command is specified), the option ALL is equivalent to option YES since the computed load increment is always equal to the user-specified load increment.
  - For SOL 400 only, if the adaptive time stepping scheme is used (i.e., when a NLAUTO Bulk Data entry with the same ID is specified), INTOUT is allowed to be Integer > 0. In this case, the load step is divided into INTOUT increments for output. For example, if INTOUT=5, the output will be at load increments 0.2, 0.4, 0.6, 0.8, and 1.0. Please note that INTOUT defines only the output load increments, which are different from the analysis increments. The analysis load increment size is smaller than or equal to the output load increment size.
  - For arc-length methods (i.e., when the NLPCI command is specified) the computed load increment in general is not going to be equal to the user-specified load increment, and is not known in advance. The option ALL allows the user to obtain solutions at the desired intermediate load increments.
  - For SOL 600 only, the default is YES (see Remark 18.)
9. The ratio of energy errors before and after the iteration is defined as divergence rate ( $E^i$ ), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If  $E^i \geq 1$  or  $E^i < -10^{12}$ , then  $NDIV = NDIV + 2$

If  $-10^{12} < E^i < -1$ , then  $NDIV = NDIV + 1$

The solution is assumed to diverge when  $NDIV \geq |MAXDIV|$ . If the solution diverges and the load increment cannot be further bisected (i.e.,  $MAXBIS$  is attained or  $MAXBIS$  is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while  $MAXDIV$  is positive, the best solution is computed and the analysis is continued to the next load increment. If  $MAXDIV$  is negative, the analysis is terminated on the second divergence.

10. The BFGS update is performed if  $MAXQN > 0$ . As many as  $MAXQN$  quasi-Newton vectors can be accumulated. The BFGS update with these QN vectors provides a secant modulus in the search direction. If  $MAXQN$  is reached, no additional QN vectors will be accumulated. Accumulated QN vectors are purged when the stiffness is updated and the accumulation is resumed.
11. The line search is performed as required, if  $MAXLS > 0$ . In the line search, the displacement increment is scaled to minimize the energy error. The line search is not performed if the absolute value of the relative energy error is less than the value specified in  $LSTOL$ .
12. The number of subincrements in the material routines (elastoplastic and creep) is determined so that the subincrement size is approximately  $FSTRESS \cdot \bar{\sigma}$  (equivalent stress).  $FSTRESS$  is also used to establish a tolerance for error correction in the elastoplastic material; i.e.,

$$\text{error in yield function} < FSTRESS \cdot \bar{\sigma}$$

If the limit is exceeded at the converging state, the program will exit with a fatal message. Otherwise, the stress state is adjusted to the current yield surface.

13. The number of bisections for a load increment/arc-length is limited to the absolute value of  $MAXBIS$ . Different actions are taken when the solution diverges depending on the sign of  $MAXBIS$ . If  $MAXBIS$  is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If  $MAXBIS$  is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after  $|MAXBIS|$  bisections, the analysis is continued or terminated depending on the sign of  $MAXDIV$ . See Remark 9.
14.  $MAXR$  is used in the adaptive load increment/arc-length method to define the overall upper and lower bounds on the load increment/arc-length in the subcase; i.e.,

$$\frac{1}{MAXR} \leq \frac{\Delta l_n}{\Delta l_o} \leq MAXR$$

where  $\Delta l_n$  is the arc-length at step  $n$  and  $\Delta l_o$  is the original arc-length. The arc-length method for load increments is selected by an NLPCI Bulk Data entry. This entry must have the same ID as the NLPARM Bulk Data entry.

15. The bisection is activated if the incremental rotation for any degree-of-freedom ( $\Delta\theta_x$ ,  $\Delta\theta_y$ , or  $\Delta\theta_z$ ) exceeds the value specified by  $RTOLB$ . This bisection strategy is based on the incremental rotation and controlled by  $MAXBIS$ .

16. The default for NINC is 10, except if there is a GAP, Line Contact, Heat Transfer or PARAM,NLTOL,0, in which case the default is 1. Default tolerance sets are determined based on model type and desired accuracy. Accuracy is under user control and can be specified on the PARAM, NLTOL entry. For SOL 106, NLTOL's value is used only if the CONV, EPSU, EPSP and EPSW fields are blank, and if NINC is set to a value of 10 or larger. Otherwise, the NLTOL selection will be overridden. The overridden values are CONV=PW, EPSP=1.0E-3, and EPSW=1.0E-7. For SOL 400, NLTOL's value is used if the value of CONV, EPSU, EPSP, or EPSW fields are blank. The tables below list tolerances according to NLTOL selections:

**Table 21** Default Tolerances for Static Nonlinear SOL 106 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
1	High	PW	_____	1.0E-2	1.0E-3
2	Engineering	PW	_____	1.0E-2	1.0E-2
3	Prelim Design	PW	_____	1.0E-1	1.0E-1
None	Engineering	PW	_____	1.0E-2	1.0E-2

**Table 22** Default Tolerances for Static Nonlinear SOL 106 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
2	Engineering	PW	_____	1.0E-3	1.0E-5
None	Engineering	PW	_____	1.0E-3	1.0E-5

**Table 23** Default Tolerances for Static Nonlinear SOL 106 or 153 Models With Heat Transfer (Enter NLTOL Value of 0 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	_____	1.0E-3	1.0E-7
None	Very high	PW	_____	1.0E-3	1.0E-7

**Table 24** Default Tolerances for Static Nonlinear SOL 400 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3

**Table 24** Default Tolerances for Static Nonlinear SOL 400 Models Without Gaps, Contact or Heat Transfer

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
1	High	PW	-	1.0E-2	1.0E-3
2	Engineering	PW	-	1.0E-2	1.0E-2
3	Prelim Design	PW	-	1.0E-1	1.0E-1
None	Engineering	PW	-	1.0E-2	1.0E-2

**Table 25** Default Tolerances for Static Nonlinear SOL 400 Models With Gaps or Contact (Enter NLTOL Values of 0 or 2 Only or Omit the Parameter)

NLTOL	Designation	CONV	EPSU	EPSP	EPSW
0	Very high	PW	-	1.0E-3	1.0E-3
2	Engineering	PW	-	1.0E-3	1.0E-3
None	Engineering	PW	-	1.0E-3	1.0E-3

17. The method to compute the energy (work) error is different for SOL 106 and SOL 400. For SOL 106, the energy error is computed based on the residue forces. While, for SOL 400, the energy error computed is the total energy error, which is based on the nonlinear forces acting on the structure. At the start of the iteration, these two methods give approximately the same value. However, near convergence, the SOL 106 method will field a much smaller value than that provided by the SOL 400 method. The difference in these two methods is reflected in the default values shown in Remark 16. The reason for a new method used in SOL 400 is that it gives the true error of the physical energy. On the other hand, the error computed in SOL 106 has no counter part in the physical world.
18. For SOL 600, the only fields used are ID, NINC, DT (creep only), KMETHOD and INTOUT, however, PARAM,MARCOTIM is recommended instead of INTOUT. For other fields, advanced convergence controls are available using NLAUTO, NLSTRAT and PARAM,MARCDEF Bulk Data entries. For SOL 600, if INTOUT is specified all NLPARM's in the file must use the same values. The first INTOUT encountered is what is actually used. The default for INTOUT is YES. For SOL 600, the initial time step for each subcase is 1/NINC of the NLPARM applicable to that subcase. If TINIT or the NLAUTO entry is entered it overrides 1/NINC as the initial time step. For arc length methods NLPCI with the same ID as NLPARM must be entered and if AIFRACT or the NLSTRAT entry is entered it will override 1/NINC as the initial increment size. Beware that NLSTRAT entries, if used, must be entered for each subcase as well as for "subcase zero". The ID of NLSTRAT do not correspond to the NLPARM Id or to the subcase ID but are numbered sequentially starting with zero for Marc increment zero, one for the first subcase (regardless of its ID) etc. For KMETHOD only, strings AUTO and ITER are supported. If any other string is entered it will be assumed to be the same as AUTO in SOL 600.

19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed with respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.
21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.  
 By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.
22. For SOL 400, the convergence checking flag "A" is implemented. "A" means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force is small, i.e., PVA → UVA. The legal combinations for A and PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example, PVA → UVA, PVA → UNA, etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.
23. For SOL 101 contact, NLPARM can be used to control nonlinear solution process, such as the number of load increments. Since the only source of nonlinearity comes from contact, some fields, however, are neither relevant to the iterative solution nor considered as user-controllable. These fields include DT(=0.0), KMETHOD(=AUTO), KSTEP(=500), INTOUT(=NO) and MAXLS(=0).

NLPARM is not required for running SOL 101 contact jobs. If it is not provided by the user, Nastran will create one. All the values of its fields are printed in F06 file under `NON - LINEAR I T E R A T I O N S O L U T I O N C O N T R O L P A R A M E T E R S`. Listed below are the remaining defaults for SOL 101 contact analysis.

NINC	MAXITER	CONV	MAXDIV	MAXQN	FSTRESS	LSTOL	MAXBIS	MAXR	RTOLB	MINITER
10	25	"PV"	3	0	0.2	0.5	5	20.0	20.0	2

Note that the defaults for EPSU, EPSP and EPSW are different for  $NINC < 10$  and  $NINC \geq 10$ .

	EPSU	EPSP	EPSW
$NINC < 10$	1.e-3	1.e-3	1.e-7
$NINC \geq 10$	1.e-2	1.e-2	1.e-2

24. If Modules are present then this entry may only be specified in the main Bulk Data section.

## NLPCI

## Parameters for Arc-Length Methods in Nonlinear Static Analysis

Defines a set of parameters for the arc-length incremental solution strategies in nonlinear static analysis (SOL 106 and SOL 400). This entry will be used if a subcase contains an NLPARM command (NLPARM = ID).

Format:

1	2	3	4	5	6	7	8	9	10
NLPCI	ID	TYPE	MINALR	MAXALR	SCALE		DESITER	MXINC	

Example:

NLPCI	10	CRIS	1.0	1.0			12	10	
-------	----	------	-----	-----	--	--	----	----	--

Describer	Meaning
ID	Identification number of an associated NLPARM entry. (Integer > 0)
TYPE	Constraint type. See Remark 2. (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (0.0 < Real ≤ 1.0; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 3. and 4. (Real ≥ 1.0; Default = 4.0)
SCALE	Scale factor (w) for controlling the loading contribution in the arc-length constraint. SOL 106 only. (Real ≥ 0.0; Default = 0.0)
DESITER	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. See Remarks 3. and 4. (Integer > 0; Default = 12)
MXINC	Maximum number of controlled increment steps allowed within a subcase. See Remark 5. (Integer > 0; Default = 20)

Remarks:

- The NLPCI entry is selected by the Case Control command NLPARM = ID. There must also be an NLPARM entry with the same ID. However, for creep analysis (DT ≠ 0.0 in NLPARM entry), the arc-length methods cannot be activated, and the NLPCI entry is ignored if specified. The NLPCI entry is not recommended for heat transfer analysis in SOL 153. Arc-length method is not supported in contact analysis.
- The available constraint types are as follows:  
TYPE = “CRIS”:

$$\left\{ u_n^i - u_n^0 \right\}^T \left\{ u_n^i - u_n^0 \right\} + w^2 (\mu^i - \mu^0)^2 = \Delta l_n^2$$

TYPE = “RIKS”:

$$\left\{ u_n^i - u_n^{i-1} \right\}^T \left\{ u_n^1 - u_n^0 \right\} + w^2 \Delta \mu^i = 0$$

TYPE = “MRIKS”:

$$\left\{ u_n^i - u_n^{i-1} \right\}^T \left\{ u_n^{i-1} - u_n^0 \right\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^0) = 0$$

where:

$w$  = the user-specified scaling factor (SCALE)

$\mu$  = the load factor

$\Delta l$  = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor ( $w$ ) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor  $\mu$ . As the value of  $w$  is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite  $w$ , the arc-length method is degenerated to the conventional Newton’s method.

3. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one load increment to the next by:

$$\text{MINALR} \leq \frac{\Delta l_{\text{new}}}{\Delta l_{\text{old}}} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use MINALR = MAXALR = 1.

4. The arc-length  $\Delta l$  for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous load increment ( $I_{\text{max}}$ ) and the number of iterations desired for convergence in the current load increment (DESITER) as follows:

$$\Delta l_{\text{new}} = \Delta l_{\text{old}} \sqrt{\frac{\text{DESITER}}{I_{\text{max}}}}$$

5. The MXINC field is used to limit the number of controlled increment steps in case the solution never reaches the specified load. This field is useful in limiting the number of increments computed for a collapse analysis.
6. NLPCI does not support general contact in SOL 400.



## NLRGAP

## Nonlinear Load Proportional to Gap

Defines a nonlinear radial (circular) gap for transient response or nonlinear harmonic response.

Format:

1	2	3	4	5	6	7	8	9	10
NLRGAP	SID	GA	GB	PLANE	TABK	TABG	TABU	RADIUS	

Example:

NLRGAP	21	3	4	XY	3	10	6	1.6	
--------	----	---	---	----	---	----	---	-----	--

Describer	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GA	Inner (e.g., shaft) grid for radial gap. (Integer > 0)
GB	Outer (e.g., housing) grid for radial gap. (Integer > 0)
PLANE	Radial gap orientation plane: XY, YZ, or ZX. (Character; Default = XY.)
TABK	TABLED1 ID defining gap stiffness vs. time or frequency (Integer > 0) or the TABLED1 ID defining gap force vs. penetration (Integer < 0), or TABLED5 ID defining frequency and (gap force vs. penetration) for nonlinear harmonic response only. See Remark 11.
TABG	Table ID for radial gap clearance as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)
TABU	Table ID for radial coefficient of friction as function of time for transient response or frequency for nonlinear harmonic response. (Integer > 0)
RADIUS	Shaft radius. (Real $\geq$ 0.0; Default = 0.0)

Remarks:

1. NLRGAP must be selected with the Case Control command NONLINEAR = SID.
2. Multiple NLRGAP entries with the same SID are allowed.
3. The NLRGAP is not an element, but a nonlinear load similar to the NOLINi Bulk Data entries. It computes the relative displacements of GA and GB in the selected plane and applies appropriate nonlinear loads to simulate the radial contact.
4. The degrees-of-freedom in the XY, YZ, or ZX planes (depending on the PLANE) of GA and GB must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation. If RADIUS is > 0.0, then the in-plane rotation degree-of-freedom must also be in the solution set.

5. The NLRGAP is limited to use in direct transient or nonlinear harmonic response solution sequences. When enforced motion is used, the NLRGAP requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
6. The XY, YZ and ZX planes are relative to the displacement coordinates systems of GA and GB. In order to ensure correct results, points GA and GB must satisfy the following three conditions:
  - a. They must both be grid points
  - b. They must be coincident
  - c. They must have parallel displacement coordinate systemsThe program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message. The coincident check can be skipped by specifying `system(648)=1`. The default is 0.
7. The shaft radius is used only for the computation of friction induced torque.
8. In the underlying equations, a positive coefficient of friction is consistent with counter-clockwise shaft rotation from axis 1 towards axis 2 (anti-clockwise). A negative coefficient of friction is consistent with clockwise shaft rotation from axis 2 towards axis 1 (clockwise). See [Figure 9-103](#).
9. Nonlinear forces for the grids referenced on the NLRGAP can be output with the NLLOAD Case Control command. See [Figure 9-103](#) for the sign conventions.

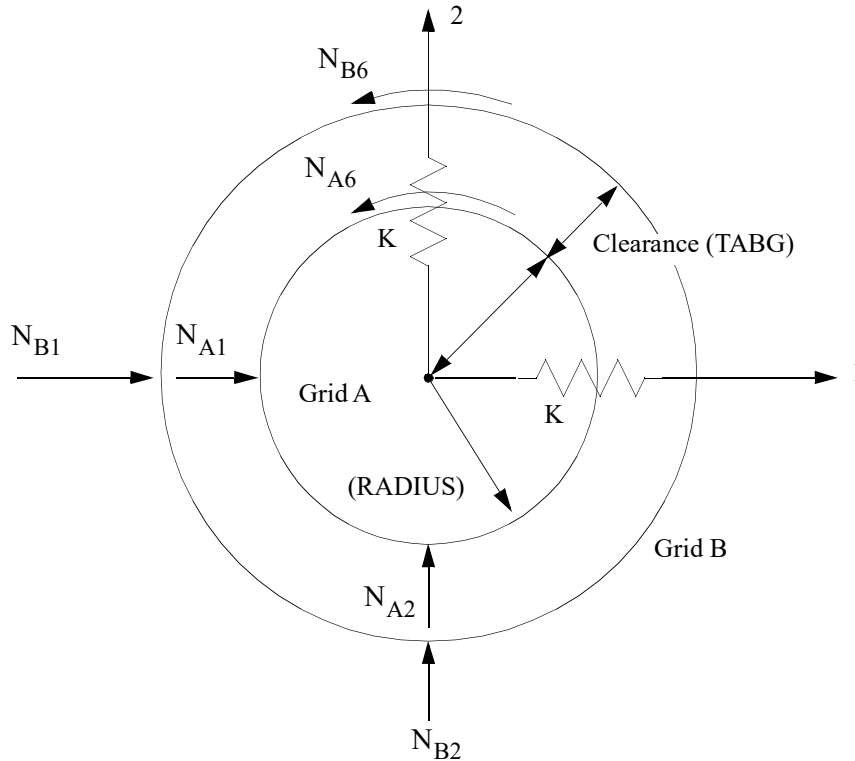


Figure 9-103 Radial Gap Orientation and Nonlinear Load Sign Conventions

10. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
11. TABK may reference either a TABLED1 ID or a TABLED5 ID.

For transient response, only a TABLED1 entry may be referenced. There are two possible scenarios:

- a. The number entered in the TABK field is a positive integer. In this case the integer references a TABLED1 entry defining pairs of time vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer. In this case the absolute value of the integer references a TABLED1 entry defining pairs of gap penetration vs. gap force.

For nonlinear harmonic response either a TABLED1 entry or a TABLED5 entry may be referenced leading to three possible scenarios:

- a. The number entered in the TABK field is a positive integer referencing a TABLED1 entry. In this case, the pairs of values defined on the TABLED1 entry are frequency vs. gap stiffness.
- b. The number entered in the TABK field is a negative integer, the absolute value of which references a TABLED1 entry. In this case, the pairs of values defined on the TABLED entry are gap penetration vs. gap force.

- c. The number entered in the TABK field is a positive integer referencing a TABLED5 entry. In this case, the pairs of values defined on the TABLED5 entry are frequency vs. a TABLED1 ID that defines pairs of gap penetration vs. gap force.
12. Forces due to TABK and TABU at GA and GB are only present when the gap is closed. A moment is applied only when the gap is closed and  $RADIUS > 0.0$ .

## NLRSD

## Nonlinear Transient Load Proportional to a Squeeze Film Damper

Defines a nonlinear transient radial squeeze film damper.

Format:

1	2	3	4	5	6	7	8	9	10
NLRSD	SID	GA	GB	PLANE	BDIA	BLEN	BCLR	SOLN	
	VISCO	PVAPCO	NPORT	PRES1	THETA1	PRES2	THETA2	NPNT	
	OFFSET1	OFFSET2	GROUP NAME	NAME2					
	RDATA1	RDATA2	RDATA3	RDATA4	RDATA5	RDATA6	RDATA7	RDATA8	

Example:

NLRSD	100	1001	1002	XY	1.0	2.0	0.05	LONG	
	2.1	300.0	1	100.0	30.0	120.0	90.0	51	
	0.01	0.0	GRPNAME						
	0.0				0.0	0.	0.0	0.0	

Alternate Example with UDS:

NLRSD	100	1001	1002	XY	1.0	1.0	1.0		
	1.0	1	1	1.0	1.0	1.0	1.0	1	
	1.0	1.0	MYSUB	EXT					
	1.0E-4								

Describer	Meaning
SID	Nonlinear load set identification number. (Integer > 0; Required)
GA	Inner (e.g., damper journal) grid for squeeze film damper. (Integer > 0; Required)
GB	Outer (e.g., housing) grid for squeeze film damper. (Integer > 0; Required)
PLANE	Radial gap orientation plane: XY, XZ, or ZX. See Remark 1. (Character; Default = XY)
BDIA	Inner journal diameter. (Real > 0.0; Required)
BLEN	Damper length. (Real > 0.0; Required)
BCLR	Damper radial clearance. (Real > 0.0; Required)
SOLN	Solution option: LONG or SHORT bearing. (Character; Default = LONG)
VISCO	Lubricant dynamic viscosity. (Real > 0.0; Required)
PVAPCO	Lubricant vapor pressure. (Real; Required)
NPORT	Number of lubrication ports: 1 or 2 (Integer; no Default)
PRES1	Boundary pressure for port 1. (Real ≥ 0.0; Required if NPORT = 1 or 2)

Descriptor	Meaning
THETA1	Angular position for port 1. ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT = 1 or 2). See Remark 2.
PRES2	Boundary pressure for port 2. ( $\text{Real} \geq 0.0$ ; Required if NPORT = 2).
THETA2	Angular position for port 2. See Remark 2. ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT = 2)
NPNT	Number of finite difference points for damper arc. (Odd Integer $\leq 201$ ; Default = 101)
OFFSET1	Offset in the SFD direction 1. (Real; Default = 0.0)
OFFSET2	Offset in the SFD direction 2. (Real; Default = 0.0)
GRPNAME	The GRPNAME is the name used for the group on the FMS CONNECT SERVICE statement; this is used with an external User Defined Service (UDS). See Remark 6. (Character or Blank)
EVALNAME	Value passed to the external User Defined Service as argument "evalname". (Character or Blank)
RDATAi	Parameters passed to the external User Defined Service as arguments "parmi". (Real; Default = 0.0). See Remark 7.

Remarks:

1. The XY, YZ, and ZX planes are relative to the displacement coordinates of GA and GB. The plane coordinates correspond to the NLRSD directions 1 and 2.

In order to ensure correct results, points GA and GB must satisfy the following three conditions:

- a. They must both be grid points
- b. They must be coincident
- c. They must have parallel displacement coordinate systems

The program checks for the above conditions. If they are not satisfied, the program terminates the execution with an appropriate fatal message.

2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the ZX plane.
3. OFFSET1 = Damper housing ID center offset displacement relative to OD center in the horizontal direction. Entered as a positive value for horizontally to the left (negative x-direction) displacement (inches).
4. OFFSET2 = Damper housing ID center offset displacement relative to OD center in the vertical direction. Entered as a positive value for downward (negative y-direction) displacement (inches). Positive entry typically used for -1 g compensation.
5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.

6. If GRPNAME is blank, the NLRSD entry used the values defined in the fields prior to GRPNAME. If GRPNAME contains a valid group (as defined on the FMS CONNECT SERVICE entry), the values in fields BDIA, BLEN, BCLR, SOLN, VISCO, PVAPCO, NPORT, PRES1, THETA1, PRES2, THETA2, NPNT, OFFSET1 and OFFSET2 are ignored and the NLRSD will obtain its characteristics from an external source via the CONNECT SERVICE architecture.
7. Parameters may be changed by the external service; the changed values will be stored and returned on a subsequent call.
8. The squeeze films damper defined via the CONNECT SERVICE architecture in NLRSD does not support rotors defined using ROTORSE.

**NLSTEP**

Describes the Control Parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Contact Analysis in SOL 101.

Specifies the convergence criteria, step size control and numerical procedure for time/load stepping in SOL 400. For multi-physics, it controls both structural and thermal analysis. Defines analysis preference and control parameters for contact analysis in SOL 101.

There are three groups of data that can be entered through this option:

1. General data which defined parameters that may be used for a variety of simulations. This data is provided by the GENERAL keyword.
2. Selecting the type of procedure used to control the time/load stepping procedure. These procedures are activated by the keywords: LCNT, FIXED, ADAPT, or ARCLN (arc length or continuation method). Only one of the keywords may be chosen in a loadcase.
3. Data associated with the physics type that are activated by the keywords: MECH, HEAT, COUP, and RCHEAT. One can enter as many as necessary.

The NLSTEP is selected by the Case Control Command NLSTEP=ID.

Format: (For SOL 400)

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME	CTRLDEF						
	"GENERAL"	MAXITER	MINITER	MAXBIS	CREEP				
	"FIXED"	NINC	NO						
	"ADAPT"	DTINITF	DTMINF	DTMAXF	NDESIR	SFACT	INTOUT	NSMAX	
		IDAMP	DAMP	CRITID	IPHYS	LIMTAR	RSMALL	RBIG	
		ADJUST	MSTEP	RB	UTOL				
	"ARCLN"	TYPE	DTINITFA	MINALR	MAXALR		NDESIRA	NSMAXA	
	"HEAT"	CONVH	EPSUH	EPSPH	EPSWH	KMETHODH	KSTEPH		
		MAXQNH	MAXLSH	LSTOLH					
	"MECH"	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCONV	
		MAXQN	MAXLS	LSTOL	FSTRESS				
	"COUP"	HGENPLAS	HGENFRIC						
	"RCHEAT"	SOLVER	DRLXCA	ARLXCA	BALENG	DAMPC	GRVCON	CSGFAC	
		NRLOOP	OUTINV	DTIME1					



Format: (For SOL 101)

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID		CTRLDEF						
	"LCNT"	NINCC	CONVC	EPSUC	EPSPC	EPSWC	MAXDIVC	MAXBIS	
		MAXITERC	MINITERC						

Example: (Fixed stepping, 30 increments, total time 4.3, max 5 bisections)

NLSTEP	10	4.3							
	GENERAL			5					
	FIXED	30							
	MECH	PV		0.01					

Example: (Automatic stepping, total time 4.3. Start out with 2%)

NLSTEP	20	4.3							
	GENERAL			5					
	ADAPT	0.02	1.-5		5		20		

Example: (Using SEVERELY default setting with automatic time stepping)

NLSTEP	10	4.3	SEVERELY						
	ADAPT								

Example: (Select the default control parameters for accuracy preference in SOL 101)

NLSTEP	10		LCACCU						
--------	----	--	--------	--	--	--	--	--	--

Example: (Four increments, P convergence criterion with error tolerance 1.e-4. The other parameters are defaults for performance preference in SOL 101)

NLSTEP	10		LCPERF						
	LCNT	4	P		1.e-4				

Describer	Meaning
ID	Identification number. (Integer > 0)
TOTTIM	Total time for the load case. (Real; Default = 1.0)

Describer	Meaning
CTRLDEF	<p>This Keyword is applicable in SOL101 static contact analysis or in SOL400 nonlinear static and transient analysis.</p> <p>For SOL400, the values "QLINEAR", "MILDLY", and "SEVERELY" are available along with the option of leaving blank. See Remark 14. for detail description of these three keyword options and the blank option.</p> <p>For SOL101, the values "LCPERF" or "LCACCU" are available along with the option of leaving blank. See Remark 15. for detail description of these two keyword options. If left blank the values as described under the "LCNT" keyword entry will be used.</p> <p>If SOL400 keyword values are present in a SOL101 static contact analysis run or SOL101 keyword values are present in a SOL400 nonlinear static run, the keywords are ignored.</p> <p>In SOL400 nonlinear transient analysis, the use of keywords "QLINEAR", "MILDLY", and "SEVERELY" or blank <i>cannot</i> properly set a "FIXED" NINC value or an "ADAPT" DTINITF value so the user <i>still must</i> set whichever one of these keyword entries value is relevant for a transient run.</p> <p>The keywords "QLINEAR", "MILDLY", and "SEVERELY" are ignored for a RC Network heat transfer simulation.</p>
"GENERAL"	Keyword for parameters used for overall analysis.
MAXITER	Maximum number of iterations allowed for each increment. See Remark 6. (Integer; Default = 10)
MINITER	Minimum number of iterations needed for each increment. (Integer >0; Default = 1) In contact analysis or CTRLDEF = SEVERELY, Default = 2). When high accuracy is required, it is also recommended to set MINITER = 2.
MAXBIS	Maximum number of bisections allowed in the current step. (Default = 10). See remark 17. for more information.
CREEP	<p>This flag allows the activation of creep for a particular step. Note: the permanent creep flag specified through the NLMOPTS, CREEP option should also be flagged for the activation. If set to 1, creep will be used; if set to 0, creep will not be used (even if creep data has been defined for a material using MATVP). (Integer 0 or 1; Default = 0).</p> <p>Adaptive load stepping based upon creep can be flagged using CRITTID below. For enhanced nonlinear elements only.</p>
"FIXED"	Keyword to indicate that fixed time stepping is to be used. See Remark 1.
NINC	Number of increments for fixed time stepping. (Integer > 0; Default = 50). The time step of each increment will be TOTTIME/NINC.
NO	Interval for output. Every NO-th increment will be saved for output. (Integer $\geq$ 0; Default = 1).

Descriptor	Meaning
"ADAPT"	Keyword to indicate that the adaptive load stepping procedure should be used. See Remark 1.
DTINITF	Initial time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.01). If DTINITF >= DTMAXF then, DTINITF is reset to DTMAXF. If CTRLDEF is set to QLNEAR, the user should set DTINITF equal to TOTTIM.
DTMINF	Minimum time step defined as fraction of total load step time (TOTTIM). (Real; Default 1e-5).
DTMAXF	Maximum time step defined as fraction of total load step time (TOTTIM). (Real; Default 0.5). For most nonlinear problems, this should be between 0.05 and 0.2 for dynamic simulations.
NDESIR	Desired number of iterations per increment. See Remark 2. (Integer; Default = 4)
SFACT	Factor for increasing time steps due to number of iterations. See Remark 4. (Real; Default = 1.2).
INTOUT	Output flag. Integer $\geq -1$ . (Default = 0) -1 Only the last increment of the step will be output. 0 Every computed load increment will be output. > 0 The output will be obtained at INTOUT equally spaced intervals. The time step will be temporarily adjusted if necessary in order to reach these points in time.
NSMAX	Maximum number of increments in the current load case. (Integer; Default = 99999). The job will stop if this limit is reached.
IDAMP	Flag for activating artificial damping for static analysis. (Integer, Default = 0). See Remarks 27. 0 No damping considered 4 Artificial damping is always turned on 5 Artificial damping is not turned on. But time step is adjusted based on damping energy as 4 6 When the time step reaches the minimum value and artificial damping is turned on
DAMP	Damping ratio. (Real; Default = 2.e-4).
CRITTID	ID of Bulk Data TABSCTL entry which defines the user criteria to use. See Remark 5. (Integer; Default 0)
IPHYS	Flag to determine if automatic physical criteria should be added and how analysis should proceed if a user criterion is not satisfied. (Integer; Default = 2) It is recommended to use 1.

Describer	Meaning
	2 Do not add automatic physical criteria; stop when any user criterion is not satisfied
	-2 Do not add automatic physical criteria; continue when user criteria are not satisfied
	1 Add automatic physical criteria; stop when any user criterion is not satisfied
	-1 Add automatic physical criteria; continue when any user criterion is not satisfied
LIMTAR	Enter 0 to treat user criteria as limits, 1 to treat user criteria as targets. (Integer; Default = 0). Only used if a user criterion is given through CRITTID. See Remark 5.
RSMALL	Smallest ratio between time step changes due to user criteria. (Real; Default = 0.1)
RBIG	Largest ratio between time step changes due to user criteria. (Real; Default = 10.0)
ADJUST	Time step skip factor for automatic time step adjustment. Only for dynamics. See Remark 16. (Integer; Default = 0).
MSTEP	Number of steps to obtain the dominant period response. See Remark 16. ( $10 \leq \text{Integer} \leq 200$ or $= -1$ ; Default = 10).
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 16. ( $0.1 \leq \text{Real} \leq 1.0$ ; Default = 0.6).
UTOL	Defines tolerance on displacement. ( $.0001 \leq \text{Real} \leq 1.0$ ; Default = 1.0)
"LCNT"	Keyword to indicate that the contact analysis in SOL101 should be used. See Remark 15.
NINCC	Number of increments. (Integer > 0; Default = 10 for CTRLDEF=""; Default=1 for CTRLDEF="LCPERF" or "LCACCU")
CONVC	Flags to select convergence criteria. (Character="U", "P", "W", "V", or any combination; Default = "PV" for CTRLDEF = "" or "LCPERF"; Default = "UPV" for CTRLDEF="LCACCU"). See Remark 19.
EPSUC	Error tolerance for displacement (U) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF=" " or "LCACCU")
EPSPC	Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF=" " or "LCACCU")
EPSWC	Error tolerance for work (W) criterion. (Real > 0.0; Default = 1.0E-7 for CTRLDEF = "LCPERF"; Default = 1.0E-2 for CTRLDEF=" " or "LCACCU")
MAXDIVC	Limit on probable divergence conditions per iteration before the solution is assumed to diverge. (Integer $\neq 0$ ; Default = 3 for CTRLDEF = "" or "LCPERF"; Default = 5 for CTRLDEF="LCACCU").
MAXBISC	Maximum number of bisections allowed for each load increment. ( $-10 \leq \text{Integer} \leq 10$ ; Default = 5)
MAXITERC	Limit on number of iterations for each load increment. (Integer $\geq 0$ , Default = 25)

Descriptor	Meaning
MINITERC	Minimum number of iterations of a load increment. (Integer > 0; Default = 1. In contact analysis, Default = 2).
“ARCLN”	Keyword to indicate that an arc length load stepping procedure is to be used. It does not support general contact in SOL 400. See Remark 1.
TYPE	Constraint type. (Character: “CRIS”, “RIKS”, or “MRIKS”; Default = “CRIS”) See Remark 7.
DTINITFA	Initial time step defined as a fraction of the load step time (TOTTIME) for the arc-length procedure. (Real; Default .01)
MINALR	Minimum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (0.0 < Real < 1.0; Default = 0.25)
MAXALR	Maximum allowable arc-length adjustment ratio between increments for the adaptive arc-length method. See Remarks 8. and 9. (Real > 1.0; Default = 4.0)
NDESIRA	Desired number of iterations for convergence to be used for the adaptive arc-length adjustment. (Integer > 0; Default = 4) See Remarks 8. and 9.
NSMAXA	Maximum number of increments in the current load case. (Integer; Default = 1000). The job will stop if this limit is reached
“HEAT”	Keyword for parameters for heat transfer analysis.
CONVH	Flags to select convergence criteria. (Character = “U”, “P”, “W”, “V”, “N”, “A” or any combination. (Default UPW)
EPSUH	Error tolerance for temperature (U) criterion. (Real; Default =1.0E-2)
EPSPH	Error tolerance for heat flux (P) criterion. (Real; Default =1.0E-2)
EPSWH	Error tolerance for work (W) criterion. (Real; Default =1.0E-2)
KMETHODH	Method for controlling stiffness updates. (Character = “PFNT”, “AUTO” or “ITER”). Default = “AUTO”. See Remark 22.
KSTEPH	Number of iterations before the stiffness update for the ITER method. (Integer; Default = 1).
MAXQNH	Maximum number of quasi-Newton correction vectors to be saved on database. (Integer; Default = MAXITER). Not used for PFNT.
MAXLSH	Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.
LSTOLH	Line Search tolerance. (0.01 < Real < 0.9; Default = 0.5)
“MECH”	Keyword for parameters for mechanical analysis
CONV	Flags to select convergence criteria. (Character = “U”, “P”, “W”, “V”, “N”, “A” or any combination; Default = PV). See Remark 19.
EPSU	Error tolerance for displacement (U) criterion. (Real; Default = -0.1)
EPSP	Error tolerance for residual load (P) criterion. (Real; Default = 0.1)

Describer	Meaning
EPSW	Error tolerance for work (W) criterion. (Real; Default = 0.1)
KMETHOD	Method for controlling stiffness updates. (Character = "PFNT" or "ITER"; Default = "PFNT"). See Remark 22.
KSTEP	Number of iterations before the stiffness update for the ITER method. (Integer; Default = 10).
MRCONV	Flag to specify if rotations and moments should be included in the convergence testing when CONV is set to UV, UN, PV, PN, UPV or UPN: (Integer; Default = 3) 0 check on forces, moments, displacements and rotations 1 check on forces, moments and displacements 2 check on forces, displacements and rotations 3 check on forces and displacements If there is no V and N in the CONV, MRCONV is ignored and all translational and rotational quantities are used to compute the convergence criteria.
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. (Integer; Default = MAXITER). Not used for PFNT.
MAXLS	Maximum number of line searches allowed for each iteration. (Integer; Default = 4). Not used for PFNT.
LSTOL	Line Search tolerance. (0.01 < Real < 0.9; Default = 0.5). Not used for PFNT.
FSTRESS	Fraction of effective stress used to limit the sub increment size in material routines (0.0 < Real < 1.0; Default = 0.2) For non material enhanced elements only.
"COUP"	Keyword for parameters for coupled analysis.
HGENPLAS	Conversion factor for heat generated due to plasticity. (Real; Default = 0.0)
HGENFRIC	Conversion factor for heat generated due to friction. (Real; Default = 0.0)
"RCHEAT"	Keyword to indicate that RC Heat Transfer Analysis is to be performed. See Remark 10.
SOLVER	The Relaxation scheme to be used. See Remark 12. (Character; Default = "SNSOR")
DRLXCA	Diffusion node convergence criterion. See Remark 11. (Real $\geq$ 0; Default = 1.e-3)
ARLXCA	Arithmetic node convergence criterion. See Remark 11. (Real $\geq$ 0.0; Default = 1.0e-3 degrees)
BALENG	Allowable system energy imbalance. See Remark 11. (Real $\geq$ 0.0; Default 0.0 energy/time)
DAMPC	Damping constant. (Real $\geq$ 0.0; Default 0.0 non dimensional)
GRVCON	Gravitation constant. (Real $\geq$ 0.0; Default 9.81 length/time <sup>2</sup> )
CSGFAC	Time step control factor. See Remark 13. (Real $\geq$ 0.0; Default 1.0 nondimensional)
NRLOOP	Number of relaxation loops allowed. (Integer $\geq$ 0; Default 5 loop)
OUTINV	Output interval. See Remark 13. (Real $\geq$ 0.0; Default 60.0 time)
DTIME1	Time step. See Remark 13. (Real $\geq$ 0.0; Default 0.0 time)

**Remarks:**

1. Note that the following applies:
  - a. Only one of LCNT, FIXED, ADAPT, ARCLN, or RCHEAT time/load stepping scheme can be used on a specific NLSTEP entry.  
Note that only LCNT may be used in SOL 101. If other selection of load stepping scheme is used, it will be ignored, and the default value of LCNT will be applied
  - b. FIXED or ADAPT may be used for a single physics STEP or for a coupled physics STEP/SUBSTEP.
  - c. If no LCNT, FIXED, ADAPT, or ARCLN appear on a NLSTEP entry, then the default is FIXED with 50 increments.
  - d. The ARCLN method is not supported in contact analysis.
  - e. The ARCLN method should not be used in transient dynamic analysis.
  - f. The ARCLN method should not be used in conjunction with HEAT or COUP.
  - g. The ARCLN method cannot be used with the Intel MKL PARDISO solver.
  - h. In one simulation, one cannot use both HEAT and RCHEAT.
  - i. One cannot use both COUP and RCHEAT.
  - j. The LCNT keyword can only be used in a SOL 101 static analysis.
2. The desired number of recycles (NDESIR) can be used in static mechanical, dynamic mechanical (see Remark 16) and heat transfer.
3. In coupled analysis, the time step change is calculated separately for heat and mechanical and the smallest of the two is used.
4. When the time step is increased due to desired number of recycles, the previous time step is multiplied with SFACT. When the time step is decreased the factor is calculated internally based upon the minimum time step.
5. User criteria can be given in the TABSCTL entry via CRITTID. These criteria include rotation, displacements, stresses, strains, creep strains. The time step is decreased if the current value of the value is larger than the user specified limit. If LIMTAR is equal to 1 (“target”) it also increases the time step for the next increment if the current value is smaller than the target value given.  
User criteria are generally useful for explicit creep analysis with adaptive time stepping. The two recommended criteria in this case would be:
  - (i) Normalized creep strain increment  $< 0.5$
  - (ii) Normalized stress change increment  $< 0.1$
6. If MAXITER is given a negative value and the MAXITER number of iterations are obtained, convergence is assumed and the analysis will continue with the next increment.
7. The “ARCLN” entry is applicable to “MECH” analysis only and is ignored for creep analysis. The available constraint types are as follows.

TYPE = “CRIS”:

$$\left\{ U_n^i - U_n^O \right\}^T \left\{ U_n^i - U_n^O \right\} + w^2 (\mu^i - \mu^O)^2 = \Delta I_n^2$$

TYPE = "RIKS":

$$\left\{ U_n^i - U_n^{i-1} \right\}^T \left\{ U_n^i - U_n^O \right\} + w^2 \Delta \mu^i = 0$$

TYPE = "MRIKS":

$$\left\{ U_n^i - U_n^{i-1} \right\}^T \left\{ U_n^{i-1} - U_n^O \right\} + w^2 \Delta \mu^i (\mu^{i-1} - \mu^O) = 0$$

where:

$w$  = user specified scaling factor (SCALEA)

$\mu$  = load factor

$l$  = the arc-length

The constraint equation has a disparity in the dimension by mixing the displacements with the load factor. The scaling factor ( $w$ ) is introduced as user input so that the user can make constraint equation unit-dependent by a proper scaling of the load factor ( $\mu$ ). As the value of  $\mu$  is increased, the constraint equation is gradually dominated by the load term. In the limiting case of infinite, the arc-length method is degenerated to the conventional Newton's method

8. The MINALR and MAXALR fields are used to limit the adjustment of the arc-length from one increment to the next by:

$$\text{MINALR} \leq \Delta l_{new} / \Delta l_{old} \leq \text{MAXALR}$$

The arc-length adjustment is based on the convergence rate (i.e., number of iterations required for convergence) and the change in stiffness. For constant arc-length during analysis, use:

$$\text{MINALR} = \text{MAXALR} = 1$$

9. The arc-length  $l$  for the variable arc-length strategy is adjusted based on the number of iterations that were required for convergence in the previous increment ( $I_{max}$ ) and the number of iterations desired for convergence in the current increment (NDESIRA) as follows:

$$\Delta l_{new} = \Delta l_{old} (\text{NDESIRA} / I_{max})^{1/2}$$

10. This entry is required for a non finite element, Resistance-Capacitor network method of analysis for heat transfer.



11. Convergence is determined by the combination of DRLXCA, ARLXCA, and BALENG. DRLXCA and ARLXCA determine if relaxation is met on a node by node basis, rather than a residual vector length.
12. If in Case Control the ANALYSIS=RCNS, then valid values of SOLVER are:

SNSOR (Default)	Successive over-relaxation method
STDSTL	An iterative solver aimed at the fourth root of a quartic for the network equations (good for strong radiation dependence).

If in Case Control the ANALYSIS=RCNT, then valid values are

SNDUFR (Recommended)	An unconditionally stable, explicit method based on a modified Dufort-Frankel scheme
SNFRDL	Fast, accurate explicit forward differencing transient method
FWDBKL	Implicit forward/backward differencing Crank Nicolson method
SNADE	Alternating direction explicit method
ATSDUF	SNFUFR with automatic time step based on ERRMIN/ERRMAX
ATSF BK	FWDBKL with automatic time step based on ERRMIN/ERRMAX
SNTSM	Weighted implicit forward/backward differencing method
SNTSM3	Weighted implicit forward/backward differencing method
SNTSM1	Weighted implicit forward/backward differencing method
SNTSM4	Weighted implicit forward/backward differencing method

If SOLVER is left blank or set to SNSOR and ANALYSIS=RCNT then internally the RC code will select SNDUFR.

13. About the time step:
  - a. The default computed time step (DTIMEU) = CSGMIN\*CSGFAC. CSGMIN is based on the conductance in the model and can be checked in the .sot file. If CSGFAC is not specified, it is internally set to 1.0.
  - b. In a normal sized model, CSGMIN is usually small enough for the time step which will assure a convergent transient run.
  - c. CSGFAC is used to adjust the time step. It is recommended to determine the best CSGFAC to the model while maintaining acceptable temperature errors.
  - d. If OUTPUT < CSGFAC\*CSGMIN or OUTPUT < DTIMEI, then OUTPUT becomes the time step. All the OUTPUT points are automatically required to be calculated.
  - e. DTIMEI is the forced time step which will ignore any other factors. Sometimes it may lead to inaccurate answers if it is too large. DTIMEI does not affect the automatic time step solvers.

- f. If the model size is very small, CSGMIN may be too big for the time step. A small CSGFAC or DTIMEI should be used to adjust the time step.
- g. CSGFAC\*CSGMIN or DTIMEI should be small enough to “catch” any details in time fields, temperature fields or flux arrays.
14. In SOL 400, CTRLDEF entry is only valid for "MECH", "HEAT" and "COUP". The description and default of each option is listed below.

CTRLDEF	<p>Keyword to select the default setting of the control parameters. (Character = "QLINEAR", "MILDLY", "SEVERELY". No default.</p> <p>Stress analysis (NLSTAT, NLTRAN):</p> <pre> QLINEAR : FIXED = 1 inc, ADAPT : initial time ratio = 1.0 (maximum time ratio = 1.0) MILDLY  : FIXED = 10 inc, ADAPT : initial time ratio = 0.1 SEVERELY : FIXED = 50 inc, ADAPT : initial time ratio = 0.01 Blank   : FIXED = 50 inc, ADAPT : initial time ratio = 0.01  QLINEAR   CONV      EPSU*      EPSP      EPSW      KMETHOD PV         0.001     0.001     0.001     ---      PFNT MILDLY    PV         0.01      0.01      ---      PFNT SEVERELY  PV         0.1       0.01      ---      PFNT Blank     PV         0.1       0.1       ---      PFNT </pre> <p>[Note] EPSU* is a suggestion when user wants to set CONV=UPV</p> <p>Thermal analysis (HSTAT, HTRAN) :</p> <pre> QLINEAR : FIXED = 1 MILDLY  : FIXED = 10 SEVERELY : FIXED = 50 Blank   : FIXED = 50  QLINEAR   CON VH      EPSUH      EPSPH      EPSWH      KMETHOD UPW       0.001     0.001     0.001     0.001     AUTO MILDLY    UPW       0.01      0.01      0.01      AUTO SEVERELY  UP       0.01      0.01      ---      AUTO Blank     UPW       0.01      0.01      0.01      AUTO </pre> <p>For stress analysis, both (1) CTRLDEF and (2) "FIXED" or "ADAPT" have to be set. For heat analysis, both CTRLDEF and "FIXED" have to be set. Otherwise the standard default setting for "FIXED" or "ADAPT" will be set.</p>
---------	---

15. "LCPERF" specifies the performance preference during analysis, while "LCACCU" prefers accuracy for analysis. These keywords must be defined if the smart contact in SOL 101 default is required. Specification of LCNT keyword is optional. Listed below are default control parameters of contact in SOL 101 if LCNT keyword or some of its fields are not defined. If the CTRLDEF field is blank, default control parameters will be same as those of NLPARM Bulk Data entry.

CTRLDEF	NINCC	CONVC	EPSUC	EPSPC	EPSWC	MAXDIVC	MAXBISC	MAXITERC	MINITERC
LCPERF	1	PV	1.e-3	1.e-3	1.e-7	3	5	25	2
LCACCU	1	UPV	1.e-2	1.e-2	1.e-2	5	5	25	2
	10	PV	1.e-2	1.e-2	1.e-2	3	5	25	2

The default error tolerances EPSPC and EPSWC are relaxed for the accuracy preference LCACCU. These relaxations are imperative to avoid convergence problems with an additional displacement convergence criterion. The default control parameters are selected based on the test results of typical contact in SOL 101 models. Therefore, these empirical data are only served for the purpose of general convenience. The users can customize any control parameters while performing their analysis.

For some contact models, specifying "LCPERF" or "LCACCU" preference may result in poor convergence. In these cases, the user may increase the number of increments (NINCC) to achieve optimal results.

16. If  $10 \leq \text{MSTEP} \leq 200$  : activating frequency based adaptive time step adjustment with bounds (RB) when specified increment (controlled by ADJUST) converges. It cannot be combined with iteration based adaptive time step adjustment, controlled by NDESIR, and NDESIR is always ignored in this case. If  $\text{MSTEP} = -1$ : activating frequency based adaptive time step adjustment without bounds (RB). It can be combined with iteration based adaptive time step adjustment controlled by NDESIR. If  $\text{NDESIR}=0$ , time step is only adjusted after the increment converges.
17. If MAXBIS is positive and the solution does not converge after MAXBIS bisections, the best solution is computed and the analysis is continued to the next time step. If MAXBIS is negative and the solution does not converge in  $|\text{MAXBIS}|$  bisection, the analysis is terminated. It is recommended to enter 0.
18. SOL400 only support the displacement error computation with respect to the delta displacement of a load increment for EPSU. Although a positive EPSU is assigned, internally it will be changed into a negative value. If  $\text{EPSW} > 0.0$ , the energy error is computed with respect to the total energy. If  $\text{EPSW} < 0.0$ , the energy error is computed with respect to the delta energy of a load increment.
19. The test flags (U = displacement error, P = load equilibrium error, W = work error, V = vector component method, N = length method, and A = auto switch) and the tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All the requested criteria (combination of U, P, W, V and/or N) are satisfied upon convergence. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For SOL 400 if  $\text{CONV} = \text{'blank'}$  the code will use a default of "UPW" if heat analysis and "PV" if a structural stress analysis is performed. See the *MSC Nastran Handbook for Nonlinear Analysis* for more details on convergence criteria. For V and N, see Remark 20. For A, see Remark 21.
20. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if  $\text{CONV}=\text{UV}$ , then V checking method will be performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users.  $\text{CONV}=\text{V}$  is the same as  $\text{CONV}=\text{UPV}$ . If both V and N are specified; V will take precedence over N. For example,  $\text{CONV}=\text{VN}$  is the same as  $\text{CONV}=\text{V}$ .

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

21. For SOL 400, the convergence checking flag “A” is implemented. “A” means automatically switching to an appropriate convergence checking flag if an unappropriated one is selected for a particular problem. For example, for the problem of stress-free contact analysis, the convergence checking flag PV is inappropriate because this may result of zero divided by zero in convergence checking computation. In this case, PV is switched to UV automatically if A is selected and the residual force is small, i.e.,  $PVA \rightarrow UVA$ . The legal combinations for A are PA, UA, WA, PVA, UVA, PNA, and UNA. The rules for auto-switching are that P is switched to U, U is switched to P, and W is switched to UP. For example,  $PVA \rightarrow UVA$ ,  $PVA \rightarrow UNA$ , etc. For all other combinations, the A selection is ignored, for example, UPA is the same as UP.
22. The stiffness update strategy is selected in the KMETHOD and KMETHODH field for mechanical and thermal analysis, respectively.
- AUTO may be used for thermal analysis only. PFNT and ITER may be used in both mechanical and thermal analysis.
  - In the thermal analysis, if the AUTO option is selected, the program automatically selects the most efficient strategy based on convergence rates. At each step the number of iterations required to converge is estimated. Stiffness is updated, if (i) estimated number of iterations to converge exceeds MAXITER, (ii) estimated time required for convergence with current stiffness exceeds the estimated time required for convergence with updated stiffness, and (iii) solution diverges. See Remarks 17. and 23. for diverging solutions. Available for the heat transfer pass only.
  - If the ITER option is selected, the program updates the stiffness matrix at every KSTEP iterations and on convergence if  $KSTEP < MAXITER$ . However, if  $KSTEP > MAXITER$ , stiffness matrix is never updated. Note that the modified Newton-Raphson iteration method is obtained by selecting the ITER option and  $KSTEP = MAXITER$ .
  - If the PFNT option is selected, the program will use the Pure Full Newton iteration method.
23. The ratio of energy errors before and after the iteration is defined as divergence rate ( $E^i$ ), i.e.,

$$E^i = \frac{\{\Delta u^i\}^T \{R^i\}}{\{\Delta u^i\}^T \{R^{i-1}\}}$$

Depending on the divergence rate, the number of diverging iteration (NDIV) is incremented as follows:

If  $E^i \geq 1$  or  $E^i < -10^{12}$ , then  $NDIV = NDIV + 2$

If  $-10^{12} < E^i < -1$ , then  $NDIV = NDIV + 1$

The solution is assumed to diverge when  $NDIV \geq |MAXDIVC|$ . If the solution diverges and the load increment cannot be further bisected (i.e., MAXBIS is attained or MAXBIS is zero), the stiffness is updated based on the previous iteration and the analysis is continued. If the solution diverges again in the same load increment while MAXDIVC is positive, the best solution is computed and the analysis is continued to the next load increment. If MAXDIVC is negative, the analysis is terminated on the second divergence.

24. Please note that in NLTRAN analysis, the output is also influenced by NLPACK. For example, one NLTRAN analysis has NLSTEP as:

```
NLSTEP, 900, 0.2
, fixed, 2000, 20
, mech, u
```

It has total 2000 increments, and asks output every 20 increments. Therefore, total output is 100 time step. In NASTRAN, default value of NLPACK for NLTRAN is 100, in this model, therefore, NASTRAN will write the results to OP2 after collecting 100 output, i.e., this model will write results to OP2 only one time. With intermediate output request, only one OP2 file will be created.

If NLPACK=1, NASTRAN will write results to OP2 for every output request. In this model, NASTRAN will write results to OP2 for every output request, i.e., writing 100 times in this model. With "intermediate output request, 100 OP2 files will be created.

If NLPACK=2, NASTRAN will write results to OP2 for every two output requests. In this model, NASTRAN will write results to OP2 for every two output request, i.e., writing 50 times in this model. With "intermediate output request, 50 OP2 files will be created.

For NO of FIXED in NLSTEP, it is similar to INTOUT of ADAPT.

25. If Modules are present then this entry may only be specified in the main Bulk Data section.
26. NLSTEP entry does not work for creep analysis when using the traditional elements. In SOL 400, NLPARM entry must be used to perform creep analysis with the traditional elements model.
27. Artificial damping takes effect in the following way with different options.

IDAMP=4: Artificial damping is always turned on. Both time stepping and added damping are controlled through the step.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.
- The damping factor to be used is computed based on the estimated damping energy and the estimated total energy for the step.

IDAMP=5: Artificial damping is not turned on. Similar algorithm as IDAMP = 4 except that only the time stepping is controlled based on the damping.

- A comparison of the incremental damping energy to the predicted incremental total strain energy is used as a criterion for time step control.

IDAMP=6: When the time step reaches the minimum value and the analysis is going to stop prematurely, two attempts are made to avoid a premature exit:

- The increment is repeated with a new time step = 10 tmin and quasi-static damping is added in a manner similar to the IDAMP = 4 scheme.

- If this does not work, the increment is repeated with a new time step = 100 tmin and the process is repeated. Note that once damping is turned on to avoid the premature stop, it remains on for the rest of the step.

**NLSTRAT****Strategy Parameters for SOL 600 Nonlinear Structural Analysis**

Defines strategy parameters for nonlinear structural analysis used in SOL 600 only. For SOL 600 Heat Transfer, see NLHEATC.

Format:

1	2	3	4	5	6	7	8	9	10
NLSTRAT	ID	Param1	Value1	Param2	Value2	Param3	Value3		
	Param4	Value4	Param5	Value5	etc				

Example:

NLSTRAT	501	CONVTYP	4	RESPF	.015	ALPHA	.05		
	KNONPOS	1							

Descriptor	Meaning
ID	Identification number referenced by a Case Control command with regard to time steps or load steps (such as SUBCASE). If ID = 0, the values entered will be used for Marc increment zero. For the first subcase ID = 1; for the second subcase ID = 2, etc. If there are no subcases in the model, enter ID = 1. If NLSTRAT with ID > 0 is entered and if the ISOLVER option is entered, another NLSTRAT entry with ID = 0 and the same ISOLVER option must be entered. If NLSTRAT is used, there must be an NLSTRAT entry for each subcase. Multiple NLSTRAT entries with the same ID are not allowed. (Integer ≥ 0)
PARAMi	Name of the NLSTRAT parameter. Allowable names are given in <a href="#">Table 9-30</a> . (Character).
VALi	Value of the parameter. See <a href="#">Table 9-30</a> . (Real or Integer)

Table 9-30 Parameters

Name	Description, Type and Value Convergence Criteria
CONVTYP	Convergence Criteria -- (Integer) - If not set, value will be determined by NLPARM or entry - CONTROL(2,4) The possibilities are: 0 -- Convergence based on residuals 1 -- Convergence based on displacements 2 -- Convergence based on energy 4 -- Convergence based on residuals or displacements 5 -- Convergence based on residuals and displacements
IRELABS	Flag for relative or absolute convergence criteria (Integer) CONTROL(2,5) 0 -- Testing is done on relative error 1 -- Testing is done on absolute value 2 -- Testing is done on relative error testing unless reactions or incremental displacements are below minimum value, in which case absolute tolerance testing is used.
RCK1	Used for Relative Checking - Maximum residual force ratio (maximum allowable value of maximum residual force divided by maximum reaction force) or displacement ratio (maximum allowable value of the change in displacement increment divided by displacement increment) depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,1); Default = 0.1)
RCK2	Used for Relative Checking - Maximum residual moment ratio or if autoswitch is on, the rotation ratio depending on CONVTYP (Real $\geq 0$ ; CONTROL(3,2); Default = no checking)
RCK3	Used for Relative Checking - Minimum reaction force ratio or minimum displacement ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,3); no Default; if 0.0, checking is bypassed or absolute testing is performed)
RCK4	Used for Relative Checking - Minimum moment ratio or rotation ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,4))
ABCK1	Used for Absolute Checking - Maximum residual force ratio or displacement ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,5); Default = no checking)
ABCK2	Used for Absolute Checking - Maximum residual moment ratio or rotation ratio depending on CONVTYP. (Real $\geq 0$ ; CONTROL(3,6) - Default = no checking)
MAXDI	Maximum change in displacement increment divided by displacement increment (Real) Default is no checking (real) - CONTROL (3a-1) - Enter only if CONVTYP is 4 or 5.
MAXRI	Maximum change in rotational increment divided by rotational increment (Real) Default is no checking (Real) - CONTROL (3a-2) - Enter only if CONVTYP is 4 or 5.
MINDI	Minimum change in displacement increment divided by displacement increment (real) Default is no checking (Real) - CONTROL (3a-3) - Enter only if CONVTYP is 4 or 5.
MINRI	Minimum change in rotational increment divided by rotational increment (real) Default is no checking (Real) - CONTROL (3a-4) - Enter only if CONVTYP is 4 or 5.



Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MAXD	Maximum value of displacement increment (Real) Default is no checking (real) - CONTROL (3a-5) - Enter only if CONVTYPE is 4 or 5.
MAXR	Minimum value of rotational increment (real) Default is no checking (Real) - CONTROL (3a-6) - Enter only if CONVTYPE is 4 or 5.
IPRCONV	Flag controls printing of convergence. CONTROL(2,9) (Integer; Default = 0 = no print)
AUTOSW	Flag to turn on or off Marc's Auto Switch (Integer) Controls switching between convergence testing of residuals and displacements when residuals are small - CONTROL (2,11). 0 -- Off (Default unless NLAUTO entry is entered) 1 -- On (Default only if NLAUTO entry is entered)
Newton Iterations	
MAXSTEP	Maximum number of load steps. CONTROL(2,1) (Integer; Default = 9999)
MAXREC	Maximum number of recycle steps per load step (Integer; Default = 3) If set to a negative value, if convergence is not obtained after maxrec recycles, a warning is issued and the analysis proceeds to the next step (not recommended) CONTROL(2,2)
MINREC	Minimum number of recycle steps per load step. CONTROL(2,3) (Integer; Default = 0)
IKMETH	Newton method (Integer; Default = 1) 1 = Full Newton, 2 = Modified Newton, 3 = Newton-Raphson with strain correction, 8 = Secant stiffness CONTROL(2,6)
IKUPD	Reassembly interval of stiffness and mass. AUTO LOAD (2,2) DYNAMIC CHANGE (2,5) (Integer)
IKNONPOS	Solve a non positive definite stiffness (1) or not (0). CONTROL(2,7) For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not. (Integer)
IKINIT	Initial stiffness control (Integer) CONTROL (2,10) 0 - Normal full contribution 1 - For Mooney material, reduced contribution of hydrostatic pressure based on initial stress 2 - No initial stiffness 3 - Use stress at beginning of increment but not in the last iteration 4 - Use only positive stresses in initial stress stiffness (faster than option 0 and is always stable for thin shell structures).
General Parameter	
STRAINS	Scale factor for strain increments. PARAMETERS(2,1) (Real $\geq 0$ ; Default = 1.0)
PENBOUN	Penalty value to enforce certain boundary conditions. PARAMETERS(2,2) (Real)

Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
FSTRESS	Fraction of the hydrostatic pressure subtracted from the stress tensor in initial stress calculation. PARAMETERS(3,5) (Real)
<b>Load Step or Time Step Control</b>	
MAXTSC	Maximum number of allowable time step cuts. AUTO LOAD (2,3) (Integer $\geq 0$ ) 0 - No automatic restart from the previously converged step >1 - Maximum number of time step cutbacks allowed.
<b>Transient Analysis Damping Parameters</b>	
BETA	Beta parameter used by Newmark-beta procedure. PARAMETERS (2,5) (Real; Default = 0.25)
GAMMA	Gamma parameter used by Newmark-beta procedure. PARAMETERS(2,6) (Real; Default = 0.5)
GAMMA1	Gamma1 parameter used by Single Step Houbolt procedure. PARAMETERS (2,7) (Real; Default = 1.5)
GAMMA2	Gamma parameter used by Single Step Houbolt procedure. PARAMETERS (2,8) (Real; Default = -0.5)
<b>Solver-Related Parameters</b>	
ISOLVER	Type of solver (Integer $\geq 0$ ) SOLVER(2,1) 0 - Profile Direct Solver 2 - Sparse Iterative 4 - Sparse Direct 6 - Hardware provided direct sparse 8 - Multifrontal direct sparse (Default) 9 - The CASI iterative solver will be used 10 - The mixed direct/iterative solver will be used. The other NLSTRAT entries that apply to solvers 2 and 9 may also be used for solver 10 but are not required. 11 - Pardiso direct solver 12 - MUMPS parallel direct solver (Note: Solvers that do not require other NLSTRAT entries can be specified by PARAM,MARCSOLV. For solvers 11 or 12 please also see parameters <a href="#">MUMPSOLV</a> and <a href="#">MRTHREAD</a> )
ISYMM	Nonsymmetric solver option (Integer $\geq 0$ ; Default = 0) SOLVER (2,2) 0 - Symmetric solver 1 - Non symmetric solver
NONPOS	Nonpositive Definite solver option (Integer $\geq 0$ ; Default = 0) SOLVER (2,3) 0 - Error if system is nonpositive-definite 1 - Solve nonpositive definite systems if possible For jobs with multiple subcases, use IKNONPOS instead of NONPOS if some subcases should handle nonpositive-definite systems and other should not.

Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
MBYTE	Solver type 6 or 8 memory option (Integer $\geq 0$ ) SOLVER (2,8). Enter the number of 4-byte words in millions to be used if solver type 6 (SGI only) or solver type 8 (all other systems) is to be used. For example, if 96MB is needed, enter 96.
MAXITER	Maximum number of iterations (Iterative solver only). (Integer; Default = 1000) Enter a negative value if program is to continue even though iterations have not fully converged. SOLVER (3,1)
PREVITER	Enter 1 if the previous solution is to be used as the initial trial value (Iterative solver only) (Integer $> 0$ ; Default = 0) SOLVER (3,2)
PRECOND	Preconditioner Option (Iterative solver only) (Integer $> 0$ ) SOLVER (3,3) For the CASI solver (solver 9): 1- CASI solver with standard preconditioner 0- CASI solver with primal preconditioner For the standard iterative solver (solver 2): 3 - Use diagonal preconditioner 4 - Use scaled-diagonal preconditioner 5 - Use incomplete Cholesky preconditioner
CJTOL	Enter Conjugate Gradient Convergence Tolerance (Iterative solver only) (Real; Default = 0.001) SOLVER (4,1)
<p>Note: Arc Length and Other Parameters for Marc's AUTO INCREMENT Option</p> <p>An NLPCI entry is needed in addition to the options below to trigger the AUTO INCREMENT option. It is usually used for post-buckling problems.)</p>	
AITYPE	Arc Length Method. AUTO INCREMENT (2,8) (Integer $\geq 0$ ; Default = 3) 0 standard load control 1 - Crisfield quadratic constraint method 2 - Riks/Ramm linear constraint method 3 - Modified Riks/Ramm (linear constrain method) 4 - Crisfield, switch to modified Riks/Ramm if no real root found
AIMAXCUT	Maximum number or time step cutbacks. AUTO INCREMENT (2,9) (Integer $\geq 0$ ; Default if not entered = 5) 0 - No automatic restart (cutbacks) from previous converged step are allowed >0 - Maximum number of load step cutbacks Note for shell buckling problems it is best to set this value to 5 or 10.
AIFRACT	Fraction of total load increment that is applied in the first cycle of the first increment. AUTO INCREMENT (2,1) (Real) Note this value is not set from NLPARM. It needs to be entered if AUTO INCREMENT is to be used for buckling problems.
AIMAXINC	Maximum number of increments. For most problems, this value should be entered and set to a large value such as 99999. AUTO INCREMENT (2,2) (Integer $> 0$ )

Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
AINRECYC	Desired number of recycles per increment used to increase or decrease load steps. AUTO INCREMENT (2,3) (Integer > 0; Default = 3)
AIMAXF	Maximum fraction of the total load that can be applied in any increment. It is recommended that for most nonlinear problems, this value be 0.1 or smaller. AUTO INCREMENT (2,4) (Real; Default=0.05 if the model does not have contact and 0.01 if the model has contact). Note for shell buckling problems it is best not to set this value (i.e., leave it as default). If AIMAXF is set to a small value, the problem will probably diverge and/or get bad results.
AIARCM	Maximum arc length multiplier (norm of displacement vector to initial arc length). AUTO INCREMENT (2,5) (Real; Default is fraction of load divided by initial fraction of load)
AITOTT	Total Time period to be covered, used in conjunction with contact analysis. AUTO INCREMENT (2,6) (Real; Default = 1.0)
AIARC0	Fraction of the initial arc length to define a minimal arc length. AUTO INCREMENT (2,7) (Real; Default = 0.01)
<b>Fully Coupled Heat-Structural Analysis Controls</b>	
TCHANGE	Maximum nodal temperature change allowed. CONTROL(4,1) (Real; Default = 20.0)
TEVAL	Maximum nodal temperature change allowed before properties are re-evaluated and matrices reassembled. CONTROL(4,2) (Real; Default = 100.0)
TERROR	Maximum error in temperature estimates used for property evaluation. CONTROL(4,3) (Real; Default = 0.0; which bypasses the test)
<b>Contact-Related Parameters</b>	
ANG2D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 2D contact (Real; Default = 8.625 degrees) PARAMETERS (3,1)
ANG3D	Angle at which a node separates from a convex corner or becomes stuck in a concave corner in 3D contact (Real; Default = 20.0 degrees) PARAMETERS (3,2)
<b>Other Parameters</b>	
DRILLF	The factor used to calculate the drilling mode for shell elements (types 22, 75, 138, 139, 140) DRILLF=1.0E-6*K6ROT. The default for DRILLF (0.0001) and K6ROT (100.0) produce the same results. If K6ROT is entered, it will be used for DRILLF unless DRILLF is also entered. DRILLF has precedence over K6ROT. (Real; Default = 0.0001) PARAMETERS (3,6)
REZONEF	Incremental displacement scale factor after a rezoning increment (Real; Default=1.0). Note that a value of 1.0 improves friction convergence but may result in an inside-out element. PARAMETERS (3,7)
UGAS	Universal Gas Constant (Real; Default = 8.314 J mol <sup>-1</sup> K <sup>-1</sup> ) PARAMETERS (4,1)

Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
TOFSET	Offset temperature between user units and absolute zero temperature (Real; Default=273.15 which is correct for Centigrade). If temperature units are Kelvin (K) or Rankine (R), enter a negative value and the temperature offset is set to zero PARAMETERS (4,2)
TWEIGHT	Thermal properties evaluation weight (Real; Default = 0.5) PARAMETERS (4,3)
SPFACT	Surface projection factor for single step Houbolt method (Real; Default = 0.0) PARAMETERS (4,4)
STEFAN	Stefan Boltzman Constant (Real; Default = 5.67051E-8 W/m <sup>2</sup> K <sup>4</sup> PARAMETERS (4,5)
PLANKS	Planks second constant. (Real; Default = 14387.69 micro MK PARAMETERS (4,6)
CLIGHT	Speed of light in a vacuum. (Real; Default = 2.9979E14 micro M/s PARAMETERS (4,7)
RAPMAX	Maximum change in the incremental displacement in a Newton-Raphson iteration. (Real; Default = 1.0E30) PARAMETERS (4,8)
FISTIF	Initial friction stiffness for model 6 used in first cycle of an increment to define the friction stiffness matrix in cases where a touching node has a zero normal force and the amount of sliding does not exceed the elastic sticking limit. (Real; Default = 0.0 in which case the program calculates it.) PARAMETERS (5,1)
SNGMIN	Minimum value that indicates a singularity if a direct solver is used. (Real; Default = 0.0 in which case the value is set internally by the program.) PARAMETERS (5, 2)
RTMAX	Maximum change in temperature per iteration in radiation simulations. (Real; Default = 10 times the maximum error in temperature estimate or 100.0) PARAMETERS (5,3)
<b>New Items for Version 2005 r3 and Subsequent</b>	
IASMBL	Assembly flag. If set to 1, the stiffness matrix is assembled each iteration. Note that this switches off the modified Newton-Raphson procedure if chosen. (Integer)

Table 9-30 Parameters (continued)

Name	Description, Type and Value Convergence Criteria
INNER	For some material models, such as damage, cracking, and Chaboche, there is an inner iteration loop to insure accuracy. The maximum number of iterations allowed can be set here. (Integer; Default = 50)
RIGLNK	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)
RLROTT	Rigid Link Rotation Tolerance: Maximum allowable value of the change in rotation increment at the retained nodes of RBE2, rigid link 80 or beam-shell offset nodes. Default is 0.0, in which case, no checking on rigid link rotations takes place. (Real)  <b>Note:</b> If CONVTYPE is 4 or 5, the rigid link rotation tolerance entered here circumvents the corresponding value RIGLNK above. For all versions with PARAM, MARCVER less than 12, the rigid link rotation tolerance if left at 0, is reset to 0.001 radians to ensure backward compatibility for RBE2. In this case, the rigid link rotation tolerance should be set to a negative number to by-pass the check.
ENRGCH	Maximum allowable value of the change in energy increment. Enter only if CONVTYPE=2. (Real; Default = 0.1)

## Remarks:

1. This entry matches Marc's CONTROL, AUTO LOAD, DYNAMIC CHANGE, PARAMETERS, and SOLVER definitions.
2. NLSTRAT is recognized only when Marc is executed from SOL 600.
3. Correlation between NLSTRAT names and Marc CONTROL entry fields

2-1 MAXSTEP	3-1 RCK1	4-1 MAXDI	5-1 MAXENRG	6-1 TCHANGE
2-2 MAXREC	3-2 RCK2	4-2 MAXRI		6-2 TEVAL
2-3 MINREC	3-3 RCK3	4-3 MINDI		6-3 TERROR
2-4 CONVTYPE	3-4 RCK4	4-4 MINRI		6-4 VOLTMAX
2-5 IRELABS	3-5 ABCK1	4-5 MAXD		
2-6 IKMETH	3-6 ABCK2	4-6 MAXR		7-1 ESRELER
2-7 IKNONPOS	4-7 RIGLNK	4-7 RLROTT		7-2 ESABSER
2-8 Not Used				
2-9 IPRCONVS			5-1 ENRGCH	
2-10 IKINT				
2-11 AUTOSW				

2-12 IASMBL

2-13 INERR

- Items 6-4,7-1,7-2 are not presently available using SOL 600).
- 3-1 to 3-6 is entered only if CONVTYPE=0, 4 or 5.
- 4-1 to 4-6 is entered only if CONVTYPE=1, 4 or 5.
- 5-1 is entered only if CONVTYPE=2.

4. Correlation between NLSTRAT names and Marc PARAMETERS entry fields

2-1 STRAINS	3-1 ANG2D	4-1 UGAS	5-1 FISTIF
2-2 PENBOUN	3-2 ANG3D	4-2 TOFSET	5-2 SNGMIN
2-3 PFPLAS	3-3 RATE0	4-3 TWEIGHT	5-3 RTMAX
2-4 PFFLUID	3-4 RATEC	4-4 SPFACT	
2-5 BETA	3-5 FSTRESS	4-5 STEFAN	
2-6 GAMMA	3-6 DRILLF	4-6 PLANKS	
2-7 GAMMA1	3-7 REZONEF	4-7 CLIGHT	
2-8 GAMMA2		4-8 RAPMAX	

5. Correlation between NLSTRAT names and Marc SOLVER entry fields

2-1 ISOLVER	3-1 MAXITER	4-1 CJTOL
2-2 ISYMM	3-2 PREVITER	
2-3 NONPOS	3-3 PRECOND	
2-8 MBYTE		

6. Correlation between NLSTRAT names and Marc AUTO INCREMENT entry fields

2-1 AIFRACT	2-6 AITOTT
2-2 AIMAXINC	2-7 AIARC0
2-3 AINRECYC	2-8 AITYPE
2-4 AIMAXF	2-9 AIMAXCUT
2-5 AIARCM	

7. Correlation between NLSTRAT names and Marc AUTO LOAD entry fields

2-2 IKUPD

2-3 MAXTSC

8. Correlation between NLSTRAT names and Marc DYNAMIC CHANGE entry fields

2-5 IKUPD

9. The ISOLVER must be the same for all load cases (and Phase 0) or Marc will abort. It is recommended that all other SOLVER items also be the same for the entire run. If ISOLVER is a value other than 8, NLSTART with ID=0 must be entered with the desired ISOLVER type. Multiply NLSTRAT entries with ID=1, 2, etc. may be used to change other values if desired.
10. ISYMM must be the same for all load cases. All discussions from note 10 apply to ISYMM.



**NOLIN1**

**Nonlinear Load as a Tabular Function**

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$\text{Function of displacement: } P_i(t) = S \cdot T(u_j(t)) \text{ or } P_i(f) = S \cdot T(u_j(f)) \tag{9-26}$$

$$\text{Function of velocity: } P_i(t) = S \cdot T(\dot{u}_j(t)) \text{ or } P_i(f) = S \cdot T(\dot{u}_j(f)) \tag{9-27}$$

where  $t$  is time,  $f$  is frequency and  $u_j(t/f)$  and  $\dot{u}_j(t/f)$  are the displacement and velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN1	SID	GI	CI	S	GJ	CJ	TID		

Example:

NOLIN1	21	3	4	2.1	3	10	6		
--------	----	---	---	-----	---	----	---	--	--

Describer	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

TID Identification number of a TABLEDi entry. (Integer > 0)

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR = SID.



2. Nonlinear loads may not be referenced on DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN1 entries must be members of the solution set. This means the e-set (EPOINT entry) for modal formulation and the d-set for direct formulation.
4. Nonlinear loads as a function of velocity (9-27) are denoted by components ten greater than the actual component number; i.e., a component of 11 is component 1 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
6. RC network solver does not support NOLIN1 for thermal analysis.
7. When enforced motion is used, the NOLIN1 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
8. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NOLIN2**

**Nonlinear Load as the Product of Two Variables**

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = S \cdot X_j(t) \cdot X_k(t) \text{ or } P_i(f) = S \cdot X_j(f) \cdot X_k(f)$$

where  $t$  is time,  $f$  is frequency and  $X_j(t/f)$  and  $X_k(t/f)$  can be either displacement or velocity at points GJ and GK in the directions of CJ and CK.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN2	SID	GI	CI	S	GJ	CJ	GK	CK	

Example:

NOLIN2	14	2	1	2.9	2	1	2		
--------	----	---	---	-----	---	---	---	--	--

Describer	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ, GK	Grid, scalar, or extra point identification number. (Integer > 0)
CJ, CK	Component number for GJ, GK according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

Remarks:

1. Nonlinear loads must be selected with the Case Control command NONLINEAR=SID.
2. Nonlinear loads may not be referenced on a DLOAD entry.
3. All degrees-of-freedom referenced on NOLIN2 entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. GI-CI, GJ-CJ, and G  
K-CK may be the same point.



5. Nonlinear loads may be a function of displacement ( $X = u$ ) or velocity ( $X = \dot{u}$ ). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_t = \frac{u_t - u_{t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{t-1}$  is the displacement of GJ-CJ or GK-CK for the previous time step.

6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support NOLIN2 for thermal analysis.
8. When enforced motion is used, the NOLIN2 requires that PARAM,ENFMETH,ABS be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with DIAG 10, although it is *not* recommended.

**NOLIN3**

**Nonlinear Load as a Positive Variable Raised to a Power**

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} S \cdot [X_j(t)]^A, & X_j(t) > 0 \\ 0, & X_j(t) \leq 0 \end{cases} \quad \text{or} \quad P_i(f) = \begin{cases} S \cdot [X_j(f)]^A, & X_j(f) > 0 \\ 0, & X_j(f) \leq 0 \end{cases}$$

where  $t$  is time,  $f$  is frequency and  $X_j(t/f)$  may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN3	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN3	4	102		-6.1	2	15	-3.5		
--------	---	-----	--	------	---	----	------	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which the nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 ≤ Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A Exponent of the forcing function. (Real)

## Remarks:

1. Nonlinear loads must be selected with the Case Control command `NONLINEAR = SID`.
2. Nonlinear loads may not be referenced on a `DLOAD` entry.
3. All degrees-of-freedom referenced on `NOLIN3` entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ( $X_j = u_j$ ) or velocity ( $X_j = \dot{u}_j$ ). Velocities are denoted by a component number ten greater than the actual component number; e.g., a component of 16 is component 6 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. Use a `NOLIN4` entry for the negative range of  $X_j(t)$ .
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support `NOLIN3` for thermal analysis.
8. When enforced motion is used, the `NOLIN3` requires that `PARAM,ENFMETH,ABS` be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with `DIAG 10`, although it is *not* recommended.

**NOLIN4**

**Nonlinear Load as a Negative Variable Raised to a Power**

Defines a forcing function for transient response or nonlinear harmonic response of the form

$$P_i(t) = \begin{cases} -S \cdot [-X_j(t)]^A, & X_j(t) < 0 \\ 0, & X_j(t) \geq 0 \end{cases} \text{ or } P_i(f) = \begin{cases} -S \cdot [-X_j(f)]^A, & X_j(f) < 0 \\ 0, & X_j(f) \geq 0 \end{cases}$$

where *t* is time, *f* is frequency and *X<sub>j</sub>(t/f)* may be a displacement or a velocity at point GJ in the direction of CJ.

Format:

1	2	3	4	5	6	7	8	9	10
NOLIN4	SID	GI	CI	S	GJ	CJ	A		

Example:

NOLIN4	2	4	6	2.0	101		16.3		
--------	---	---	---	-----	-----	--	------	--	--

Descriptor	Meaning
SID	Nonlinear load set identification number. (Integer > 0)
GI	Grid, scalar, or extra point identification number at which nonlinear load is to be applied. (Integer > 0)
CI	Component number for GI. (0 < Integer ≤ 6; blank or zero if GI is a scalar or extra point.)
S	Scale factor. (Real)
GJ	Grid, scalar, or extra point identification number. (Integer > 0)
CJ	Component number for GJ according to the following table:

Type of Point	Displacement	Velocity
Grid	1 ≤ Integer ≤ 6	11 ≤ Integer ≤ 16
Scalar	Blank or zero	Integer = 10
Extra	Blank or zero	Integer = 10

A Exponent of forcing function. (Real)

## Remarks:

1. Nonlinear loads must be selected with the Case Control command `NONLINEAR = SID`.
2. Nonlinear loads may not be referenced on a `DLOAD` entry.
3. All degrees-of-freedom referenced on `NOLIN4` entries must be members of the solution set. This means the e-set for modal formulation and the d-set for direct formulation.
4. Nonlinear loads may be a function of displacement ( $X_j = u_j$ ) or velocity ( $X_j = \dot{u}_j$ ). Velocities are denoted by a component number ten greater than the actual component number; i.e., a component of 10 is component 0 (velocity). The velocity is determined by

$$\dot{u}_{j,t} = \frac{u_{j,t} - u_{j,t-1}}{\Delta t}$$

where  $\Delta t$  is the time step interval and  $u_{j,t-1}$  is the displacement of GJ-CJ for the previous time step.

5. Use a `NOLIN3` entry for the positive range of  $X_j(t)$ .
6. The time step algorithm in transient solution sequences may lose unconditional stability when this load entry is used. In most practical cases, the time step size chosen to reach a certain accuracy is below the stability limit. It is recommended to decrease the time step if results diverge. Additional recommendations are outlined in the user guides.
7. RC network solver does not support `NOLIN4` for thermal analysis.
8. When enforced motion is used, the `NOLIN4` requires that `PARAM,ENFMETH,ABS` be specified because the generated loads are a function of total displacement and not relative displacement.
9. An alternate option for averaging nonlinear loads is available with `DIAG 10`, although it is *not* recommended.



## NSM Non Structural Mass Entry by ID

Defines a set of non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSM	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	-etc.-						

Example:

NSM	3	PSHELL	15	.022					
-----	---	--------	----	------	--	--	--	--	--

Describer	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	NSM value (Real)

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is  $1000 \cdot ID + i$ , where  $i = 1$  to number of harmonics.
3. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

## NSM1

### Alternate Form for NSM Entry

Defines non structural mass entries by VALUE,ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSM1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSM1	3	ELEMENT	.044	1240	1500	THRU	1600	BY	
	2	2440	THRU	2560					

Alternate Form and Example(s), applicable when NASTRAN SYSTEM(444)=0 (IFPSTAR=NO):

(All must be in FIELD 5 and non continuation is allowed)

NSM1	SID	TYPE	VALUE	ID	THRU	ID			
NSM1	SID	TYPE	VALUE	ALL					
NSM1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	

Describer	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
VALUE	NSM value (Real)
ID	Property or Element ID. (Integer > 0 or "ALL" or "THRU" or "BY" or N (the BY increment))

Remarks:

1. Non structural mass sets must be selected with Case Control command NSM = SID.
2. For CCONEAX the element ID is  $1000 \cdot ID + i$ , where  $i = 1$  to number of harmonics.
3. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:

NSM1,12,PCOMP,0.045,ALL

would for example get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLs would have .045 added to their nonstructural mass.

4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

## NSMADD Non Structural Mass Set Combination

Defines non structural mass as the sum of the sets listed.

Format:

1	2	3	4	5	6	7	8	9	10
NSMADD	SID	S1	S2	S3	S4	S5	S6	S7	
	S8	S9	S10	etc.	-				

Example(s):

NSMADD	3	17	18	19	20	22	23	24	
	25	26	27	28					
NSMADD	3	29	40	50	55				

Describer	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
Si	Identification numbers of non structural mass sets defined via NSM, NSML, NSM1, and NSML1 entries. (Integer > 0; SID ≠ Si)

Remarks:

1. The non structural mass sets must be selected with the Case Control command NSM = SID.
2. No Si may be the identification number of a non structural mass set defined by another NSMADD entry.
3. NSMADD entries take precedence over NSM, NSML, NSM1 or NSML1 entries. If both have the same set ID, only the NSMADD entry will be used.
4. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.

## NSML

## Lumped Non Structural Mass Entry by ID

Defines a set of lumped non structural mass.

Format:

1	2	3	4	5	6	7	8	9	10
NSML	SID	TYPE	ID	VALUE	ID	VALUE	ID	VALUE	
	ID	VALUE	-etc.-						

Example:

NSML	3	PSHELL	15	.66					
------	---	--------	----	-----	--	--	--	--	--

Describer	Meaning
SID	Identification number of non structural mass set. (Integer > 0)
TYPE	Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)
ID	Property or Element ID. (Integer > 0)
VALUE	A lumped mass value to be distributed. (Real)

Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is  $NSM = VALUE / \sum_{elements} Area$  and for Line elements the calculation is  $NSM = VALUE / \sum_{elements} Length$ .
3. Non structural mass sets must be selected with Case Control command NSM = SID.
4. This entry is not allowed for the CCONEAX element.
5. This entry will cause an equivalent NSM entry to be generated using the computed value for NSM.
6. The ELSUM Case Control command will give a summary of both structural and nonstructural mass by element or property type.
7. Undefined property/element IDs are ignored.

## NSML1

## Alternate Form for NSML Entry

Defines lumped non structural mass entries by VALUE,ID list.

Format:

1	2	3	4	5	6	7	8	9	10
NSML1	SID	TYPE	VALUE	ID	ID	ID	ID	ID	
	ID	ID	ID	etc.	-				

Example:

NSML1	3	ELEMENT	.044	1240	1500				
-------	---	---------	------	------	------	--	--	--	--

Alternate Form and Example(s):

NSML1	SID	TYPE	VALUE	ID	THRU	ID	ID	THRU	
	ID	ID	THRU	ID	ID	THRU	ID	ID	
	THRU	ID	...						

NSML1	15	PSHELL	.067	1240	THRU	1760			
	2567	THRU	2568	35689	THRU	40998			
	76	THRU	300						

NSML1	SID	TYPE	VALUE	ID	THRU	ID	BY	N	
	ID	THRU	ID	BY	N	...			

NSML1	3	PSHELL	.067	1240	THRU	1760	1763	1764	
	2567	THRU	2568	35689	TO	40999	BY	2	
	76666	76668	79834						

(ALL must be in FIELD 5 and no continuation is allowed)

NSML1	SID	TYPE	VALUE	ALL					
-------	-----	------	-------	-----	--	--	--	--	--

NSML1	59	PTUBE	.0123	ALL					
-------	----	-------	-------	-----	--	--	--	--	--

Descriptor	Meaning
------------	---------

SID Identification number of non structural mass set. (Integer > 0)

TYPE Set points to either Property entries or Element entries. Properties are: PSHELL, PCOMP, PCOMPG, PBAR, PBARL, PBEAM, PBEAML, PBCOMP, PROD, CONROD, PBEND, PSHEAR, PTUBE, PCONEAX, and PRAC2D. ELEMENT list of individual element IDs of element that can have NSM. (Character)

Describer	Meaning
VALUE	A lumped mass value to be distributed (Real)
ID	Property or Element ID. (Integer > 0 or "ALL" or "THRU" or "TO" or "BY" or N (the BY increment))

#### Remarks:

1. If TYPE = ELEMENT is used, line element (CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD) IDs cannot be mixed with Area element (CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR, CSHEAR, and CRAC2D) IDs.
2. For Area elements the calculation is  $NSM = VALUE / \sum_{elements} Area$  and for Line elements the calculation is  $NSM = VALUE / \sum_{elements} Length$ .
3. For NSML1 entries with multiple "THRU" and "THRU,BY" and "ID" lists or any such combination of entries, the  $NSM = VALUE / \sum_{elements} Area$  and for Line elements the calculation is  $NSM = VALUE / \sum_{elements} Length$  is based on the individual parent card plus all continuation entries. If an element appears more than once in these multiple combinations, its area or length will be used multiple times in the sum.
4. Nonstructural mass sets must be selected with Case Control command NSM=SID.
5. This entry is not allowed for the CCONEAX element.
6. PBEAML and PBCOMP are treated as PBEAM, PBARL is treated as PBAR, and PCOMP or PCOMPG is treated as PSHELL; therefore a command such as:  
`NSML1,12,PCOMP,1.35,ALL`  
would, for example, get all PSHELLs in the file. The converted PCOMPs or PCOMPGs plus any existing PSHELLs would have a mass of 1.35 added to their nonstructural mass.
7. The ELSUM Case Control command will give a summary of both structural and non structural mass by element or property type.
8. With the "THRU" and "THRU", "BY" forms, blanks fields are allowed for readability. Any combination of a list of IDs and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing IDs. That is the list of IDs in a THRU range need not be continuous.
9. Undefined property/element IDs are ignored.

## NTHICK

## Nodal Thickness Values - SOL 600

Defines nodal thickness values for beams, plates and/or shells. This is the Marc's nodal thickness option used in SOL 600 only.

Format:

1	2	3	4	5	6	7	8	9	10
NTHICK	ID1	ID2	THICK						

Example:

NTHICK	151	180	0.255						
--------	-----	-----	-------	--	--	--	--	--	--

Describer	Meaning
ID1	First Nodal ID to which the thickness applies. (Integer > 0)
ID2	Last Nodal ID to which the thickness applies. (Integer; Default = EID1)
THICK	Thickness for all beam, plate or shell elements connecting the nodes specified. (Real > 0.0)

Remarks:

1. The option allows specification of beam, plate and/or shell thickness on a nodal basis. Thickness values specified on property entries overrides values specified by this entry.
2. For all elements including composite elements, nodal thickness is the total thickness.
3. Discontinuities must be modeled using property entries.



## OMIT Omitted Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

Example:

OMIT	16	2	23	3516			1	4	
------	----	---	----	------	--	--	---	---	--

Describer	Meaning
ID <sub>i</sub>	Grid or scalar point identification number. (Integer > 0)
C <sub>i</sub>	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)

Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See [“Degree-of-Freedom Sets”](#) for a list of these entries.
2. Up to 24 degrees-of-freedom may be specified on a single entry.
3. In many cases it may be more convenient to use OMIT1, ASET, or ASET1 entries.
4. In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.
5. SOL 400 does not support ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, SUPORT<sub>i</sub>, and QSET<sub>i</sub> except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:
    - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, and QSET<sub>i</sub> entries.

## OMIT1 Omitted Degrees-of-Freedom, Alternate Form 1

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMIT1	C	G1	G2	G3	G4	G5	G6	G7	
	G8	G9	G10	-etc.-					

Example:

OMIT1	3	2	1	3	10	9	6	5	
	7	8							

Alternate Format and Example:

OMIT1	C	G1	“THRU”	G2					
OMIT1	0	17	THRU	109					

Describer	Meaning
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; zero or blank for scalar points.)
Gi	Grid or scalar point identification number. (Integer > 0; for “THRU” option, G1 < G2.)

Remarks:

- The degrees-of-freedom specified on this entry form members of the mutually exclusive o-set. They may not be specified on other entries that define mutually exclusive sets. See “[Degree-of-Freedom Sets](#)” for a list of these entries.
- If the alternate format is used, not all points in the range G1 through G2 have to be defined. Undefined points will collectively produce a warning message but will otherwise be ignored.
- In nonlinear analysis, degrees-of-freedom attached to nonlinear elements must be placed in the a-set. In other words, nonlinear degrees-of-freedom must not be specified on OMIT or OMIT1 entries.
- SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - Linear perturbation with:
    - EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.

- ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark [21](#). under the ADAMSMNF Case Control command.
- c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## OMITAX

## Omitted Conical Shell Degrees-of-Freedom

Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).

Format:

1	2	3	4	5	6	7	8	9	10
OMITAX	RID1	HID1	C1	RID2	HID2	C2			

Example:

OMITAX	2	6	3	4	7	1			
--------	---	---	---	---	---	---	--	--	--

Describer	Meaning
RIDi	Ring identification number. (Integer > 0)
HIDi	Harmonic identification number. (Integer ≥ 0)
Ci	Component number(s). (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

Remarks:

1. OMITAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may be specified on this entry.
3. Degrees-of-freedom appearing on OMITAX entries may not appear on MPCAX, SUPAX, or SPCAX entries.
4. SOL 400 does not support OMITAX as well ASETi, OMITi, BSETi, CSETi, SUPORTi.

## Entries P

### PAABSF

### Frequency-Dependent Absorbers Element Property

#### Format:

1	2	3	4	5	6	7	8	9	10
PAABSF	PID	TZREID	TZIMID	S	A	B	K	RHOC	

#### Example:

PAABSF	44	38	47						
--------	----	----	----	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number that matches the identification number of the corresponding CAABSF entry. (Integer > 0)
TZREID	Identification number of a TABLEDi entry that defines the resistance as a function of frequency. The real part of the impedance. See Remark 1. (Integer ≥ 0 or blank)
TZIMID	Identification number of a TABLEDi entry that defines the reactance as a function of frequency. The imaginary part of the impedance. See Remark 1. (Integer ≥ 0 or blank)
S	Impedance scale factor. (Real; Default = 1.0)
A	Area factor when 1 or 2 grid points are specified on the CAABSF entry. (Real > 0.0; Default = 1.0)
B	Equivalent structural damping coefficient. See Remark 1. (Real ≥ 0.0; Default = 0.0)
K	Equivalent structural stiffness coefficient. See Remark 1. (Real ≥ 0.0; Default = 0.0)
RHOC	Constant used in data recovery for calculating an absorption coefficient. RHO is the media density, and C is the speed of sound in the media. (Real; Default = 1.0)

#### Remarks:

- At least one of the four fields TZREID, TZIMID, B, or K must be specified.
- If only one grid point is specified on the CAABSF entry, then the impedance  $Z(f) = Z_R + iZ_I$  is the total impedance at the point. If two grids are specified, then the impedance is the impedance per unit length. If three or four points are specified, then the impedance is the impedance per unit area.  $Z_R(f) = \text{TZREID}(f) + B$  and  $Z_I(f) = \text{TZIMID}(f) - K/(2\pi f)$ .
- The resistance represents a damper quantity B. The reactance represents a quantity of the type  $(\omega M - K/\omega)$ . The impedance is defined as  $Z = p/\dot{u}$  where  $p$  is the pressure and  $\dot{u}$  is the velocity.
- The impedance scale factor S is used in computing element stiffness and damping terms as:

$$k = \frac{A}{S} \cdot \frac{2\pi f Z_I(f)}{Z_R^2 + Z_I^2} \int (\text{of shape functions})$$

$$b = \frac{A}{S} \cdot \frac{Z_R(f)}{Z_R^2 + Z_I^2} \int (\text{of shape functions})$$

The value of  $(Z_R^2 + Z_I^2)$  must be greater than machine epsilon--a machine dependent constant in the neighborhood of 1.E-15. The scale factor S can be used to ensure this constraint while retaining the same units.

- The output for the element is specified by the STRESS Case Control command and consists of the resistance, reactance, and absorption coefficient. The absorption coefficient is defined as:

$$a = \frac{4(Z_R/\rho c)}{(Z_R/\rho c + 1)^2 + (Z_I/\rho c)^2}$$

## PACABS Acoustic Absorber Property

Defines the properties of the acoustic absorber element.

### Format:

1	2	3	4	5	6	7	8	9	10
PACABS	PID	SYNTH	TID1	TID2	TID3		CUTFR	B	
	K	M							

### Example:

PACABS	12		1	2	3	3.5	500.0		
--------	----	--	---	---	---	-----	-------	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
SYNTH	Request the calculation of B, K, and M from the tables TIDi below. (Character = “YES” or “NO”; Default = “YES”)
TID1	Identification of the TABLEDi entry that defines the resistance. See Remark 2. (Integer > 0 or blank)
TID2	Identification of the TABLEDi entry that defines the reactance. See Remark 2. (Integer > 0 or blank)
TID3	Identification of the TABLEDi entry that defines the weighting function. See Remark 2. (Integer > 0 or blank)
CUTFR	Cutoff frequency for tables referenced above. (Real > 0.0)
B, K, M	Equivalent damping, stiffness and mass values per unit area. (Real ≥ 0.0)

### Remarks:

1. PACABS is referenced by a CHACAB entry only.
2. If SYNTH = “YES”, then TID1 and TID2 must be supplied (TID3 is optional) and the equivalent structural model will be derived from tables TIDi. If TID3 is blank, then the weighting function defaults to 1.0.
3. If SYNTH = “NO”, then the equivalent structural model will be derived from one of B, K, or M.
4. The continuation entry is optional.
5. All data defined in tables TIDi must be a function of frequency in cycles/unit time.

## PACBAR

## Acoustic Barrier Property

Defines the properties of the acoustic barrier element.

### Format:

1	2	3	4	5	6	7	8	9	10
PACBAR	PID	MBACK	MSEPTM	FRESON	KRESON				

### Example:

PACBAR	12	1.0	0.01	400.0					
--------	----	-----	------	-------	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MBACK	Mass per unit area of the backing material. (Real > 0.0)
MSEPTM	Mass per unit area of the septum material. (Real > 0.0)
FRESON	Resonant frequency of the sandwich construction in hertz. (Real > 0.0 or blank)
KRESON	Resonant stiffness of the sandwich construction. (Real > 0.0 or blank)

### Remarks:

1. PACBAR is referenced by a CHACBR entry only.
2. Either FRESON or KRESON must be specified, but not both.



## PACINF Acoustic Conjugate Infinite Element Property

Defines the properties of acoustic conjugate infinite elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PACINF	PID	MID	RIO	XP	YP	ZP			

### Example:

PACINF	100	10	5	0.	1.	2.			
--------	-----	----	---	----	----	----	--	--	--

Describer	Meaning
PID	Property Identification Number of PACINF entry. (Integer > 0)
MID	Material Identification Number of a MAT10 entry. (Integer > 0)
RIO	Radial Interpolation Order. (Integer > 0; no Default)
XP, YP, ZP	Coordinates of the Pole of the Infinite Elements (in the Basic Coordinate System).

### Remark:

1. The location of the pole together with the connecting grid points of the element, define the geometry of the element, see [Figure 9-104](#).

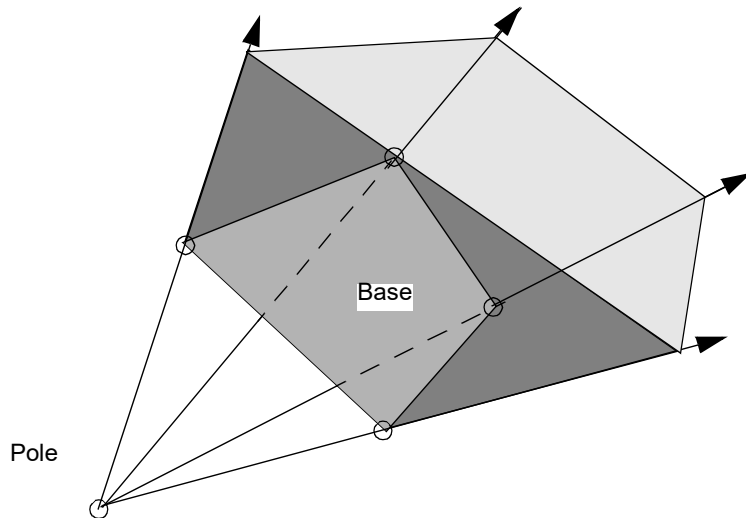


Figure 9-104 Geometry of Infinite Element

2. The radial interpolation order required depends on the directivity of the pressure field.

## PAERO1 Aerodynamic Panel Property

Defines associated bodies for the panels in the Doublet-Lattice method.

### Format:

1	2	3	4	5	6	7	8	9	10
PAERO1	PID	B1	B2	B3	B4	B5	B6		

### Example:

PAERO1	1	3							
--------	---	---	--	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number referenced by a CAERO1 entry. (Integer > 0)
Bi	Identification number of CAERO2 entries for associated bodies. Embedded blanks are not allowed. (Integer ≥ 0 or blank)

### Remarks:

1. The associated bodies must be in the same aerodynamic group, as specified in the IGID field on CAERO2 entry.
2. If there are no bodies, the entry is still required (with Bi fields blank).
3. The Bi numbers above must appear on a CAERO2 entry to define these bodies completely.

**PAERO2****Aerodynamic Body Properties**

Defines the cross-sectional properties of aerodynamic bodies.

**Format:**

1	2	3	4	5	6	7	8	9	10
PAERO2	PID	ORIENT	WIDTH	AR	LRSB	LRIB	LTH1	LTH2	
	THI1	THN1	THI2	THN2	THI3	THN3			

**Example:**

PAERO2	2	Z	6.0	1.0	22	91	100		
	1	3							

Describer	Meaning
PID	Property identification number. (Integer > 0)
ORIENT	Orientation flag. Type of motion allowed for bodies. Refers to the aerodynamic coordinate system of ACSID. See AERO entry. (Character = "Z", "Y", or "ZY")
WIDTH	Reference half-width of body and the half-width of the constant width interference tube. (Real > 0.0)
AR	Aspect ratio of the interference tube (height/width). (Real > 0.0)
LRSB	Identification number of an AEFAC T entry containing a list of slender body half-widths at the end points of the slender body elements. If blank or zero, the value of WIDTH will be used. (Integer ≥ 0 or blank)
LRIB	Identification number of an AEFAC T entry containing a list of interference body half-widths at the end points of the interference elements. If blank or zero, the value of WIDTH will be used. (Integer ≥ 0 or blank)
LTH1, LTH2	Identification number of AEFAC T entries for defining $\theta$ arrays for interference calculations. (Integer ≥ 0)
THI <sub>i</sub> , THN <sub>i</sub>	The first and last interference element of a body to use the $\theta_1$ array; the others use the $\theta_2$ array. (Integer ≥ 0)

**Remarks:**

1. The half-widths (given on AEFAC T entries referenced in fields 6 and 7) are specified at division points. The number of entries on an AEFAC T entry used to specify half-widths must be one greater than the number of elements.
2. The half-width at the first point (i.e., the nose) on a slender body is usually 0.0; thus, it is recommended (but not required) that the LRSB data is supplied with a zero first value.
3. THI<sub>i</sub> and THN<sub>i</sub> are interference element numbers on a body. The first element is one for each body.

- A body is represented by a slender body surrounded by an interference tube. The slender body creates the downwash due to the motion of the body, while the interference tube represents the effects upon panels and other bodies.

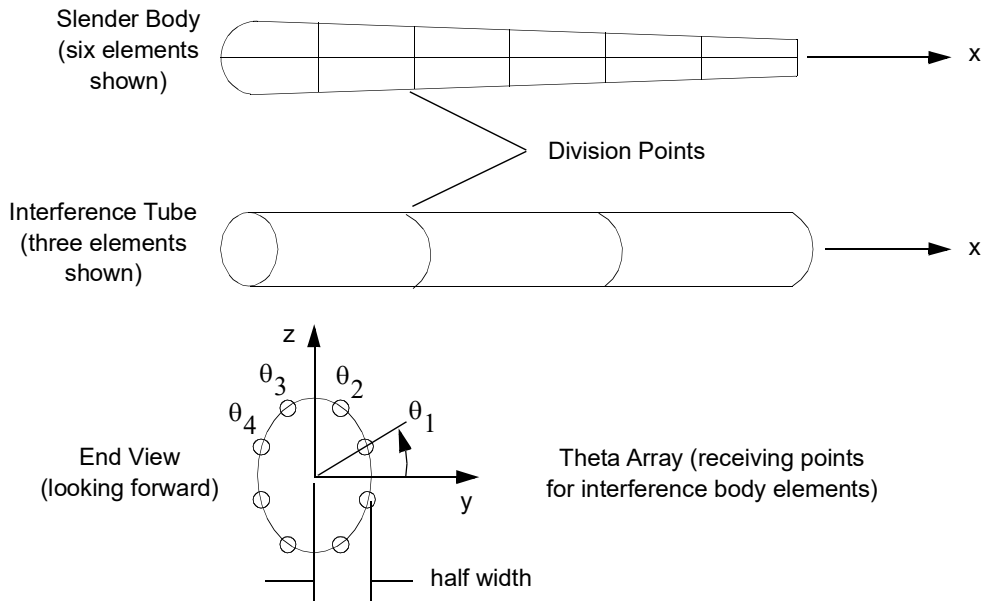


Figure 9-105 Idealization of Aerodynamic Body

- The angles  $\theta_1$  and  $\theta_2$  are input in degrees using the aerodynamic element coordinate system as the reference coordinate system for defining the theta points.
- Distribution of the theta points need not be uniform. A theta point must be placed a finite distance from any aerodynamic box edge; preferably the box edge would be equidistant from any two theta points. This aerodynamic coordinate system is defined on the AERO Bulk Data entry.
- For half models, the theta arrays LTH1 and LTH2 should encompass a full 360 degree range.

**PAERO3****Aerodynamic Panel Property**

Defines the number of Mach boxes in the flow direction and the location of cranks and control surfaces of a Mach box lifting surface.

**Format:**

1	2	3	4	5	6	7	8	9	10
PAERO3	PID	NBOX	NCTRL		X5	Y5	X6	Y6	
	X7	Y7	X8	Y8	X9	Y9	X10	Y10	
	X11	Y11	X12	Y12					

**Example:**

PAERO3	2001	15	1		0.	65.			
	78.	65.	108.	65.	82.	97.5	112.	97.5	
	86.	130.	116.	130.					

Describer	Meaning
PID	Property identification number. (Integer > 0)
NBOX	Number of Mach boxes in the flow direction. (0 < Integer < 50)
NCTRL	Number of control surfaces. (Integer 0, 1, or 2)
X5 through Y12	Locations of points 5 through 12, which are in the aerodynamic coordinate system, to define the cranks and control surface geometry. (Real)

**Remarks:**

1. For an illustration of the geometry, see the CAERO3 entry description.
2. If  $Y5 \leq 0.0$ , there is no leading edge crank. Also, if  $Y6 \leq 0.0$ , there is no trailing edge crank.
3. If NCTRL = 0, no continuations are required. If NCTRL = 1 or 2, then NCTRL continuations are required.
4.  $Y7 \geq Y8$ ,  $Y9 \geq Y10$ , and  $Y11 \geq Y12$ .
5. The number of Mach boxes in the spanwise direction (NSB) may be found from the following formula:

$$NSB = \text{INT} \left[ \frac{\beta \cdot y_{max}}{\left( \frac{x_{max}}{NBOX + 0.5} \right)} + 0.5 \right]$$

where:

$$\beta = \sqrt{M^2 - 1}$$

$x_{max}$  = maximum chordwise direction

$y_{max}$  = maximum spanwise direction

NBOX = initial number of boxes specified in field 3

The number of Mach boxes in the streamwise direction may then be computed from:

$$NBOX = \text{INT} \left[ \frac{x_{max}}{\left( \frac{\beta \cdot y_{max}}{NSB - 0.5} \right)} + 0.999 \right]$$

The number of chordwise boxes specified by the user ( $NBOX \geq 50$ ) will be replaced by a floating point number (usually slightly higher than NBOX). The method contracts the mesh equally in both dimensions until a box edge lies on the surface tip. This mesh size is then used to compute the number of chordwise boxes.

**Note:** A minimum of seven Mach boxes in the flow direction (NBOX) is recommended.

**PAERO4**

**Aerodynamic Strip Properties**

Defines properties of each strip element for Strip theory.

**Format:**

1	2	3	4	5	6	7	8	9	10
PAERO4	PID	CLA	LCLA	CIRC	LCIRC	DOC1	CAOC1	GAPOC1	
	DOC2	CAOC2	GAPOC2	DOC3	CAOC3	GAPOC3	-etc.-		

**Example:**

PAERO4	6001	1	501	0	0	0.0	0.0	0.0	
	0.50	0.25	0.02	0.53	0.24	0.0			

Describer	Meaning
PID	Property identification number. (Integer > 0)
CLA	Select Prandtl-Glauert correction. (Integer = -1, 0, 1; Default = 0) -1 Compressibility correction made to lift curve slope data for a reference Mach number. 0 No correction and no list needed. (Default) +1 No correction and lift curve slope provided by a list as a function of strip location and Mach number.
LCLA	Identification number of the AEFACCT entry that lists the lift curve slope on all strips for each Mach number on the MKAEROi entry. See Remark 7(b.) below. (Integer = 0 if CLA = 0, > 0 if CLA ≠ 0)
CIRC	Select Theodorsen’s function $C(k)$ or the number of exponential coefficients used to approximate $C(k)$ . (Integer = 0, 1, 2, 3; Default = 0; Must be zero if CLA ≠ 0.) 0 Theodorsen function. 1, 2, 3 Approximate function with $b_0, b_1, \beta_1, \dots, b_n, \beta_n$ $n = 1, 2, 3$ .
LCIRC	Identification number of the AEFACCT entry that lists the $b, \beta$ values for each Mach number. See Remark 7c., 7d., and 7e. below; variable $b$ ’s and $\beta$ ’s for each mi on the MKAEROi entry. (Integer = 0 if CIRC = 0, > 0 if CIRC ≠ 0)
DOCi	$d/c$ distance of the control surface hinge aft of the quarter-chord divided by the strip chord. (Real ≥ 0.0)
CAOCi	$c_a/c$ control surface chord divided by the strip chord. (Real ≥ 0.0)
GAPOCi	$g/c$ control surface gap divided by the strip chord. (Real ≥ 0.0)

**Remarks:**

1. PAERO4 is required for Strip theory with three fields (DOCi, CAOCi, GAPOCi) specified per strip.



2. If CLA = -1, lift curve slope data at one Mach number are needed on the AEFACCT entry.
3. If CAOCi = 0.0, there is no control surface.
4. If GAPOCivb = 0.0, there is no slot flow.
5. If GAPOCi < 0.01, then 0.01 is used.
6. Embedded blank fields are not permitted.
7. Table 9-31 lists the lift curve slope or lag function selection and the AEFACCT entry formats used for Strip theory:

Table 9-31 Strip Theory Function Selections and AEFACCT Entry Formats

Theodorsen Function	Data Type Input	Parameter Combinations				Number of Words	Entry Format Index
		CLA	LCLA	CIRC	LCIRC		
Exact	Lift Curve Slope $c_{l_{ai}} = 2\pi$	0	0	0	0	No AEFACCT entry required	
	$c_{l_{ai}}$ Input, Uses Prandtl-Glauert Correction	-1	ID	0	0	(NSTRIPI+1)	a.
	$c_{l_{ai}}$ Input, for All m's on MKAERO Entry	1	ID	0	0	(NSTRIPI+1)*NMACH	b.
Approximate Coefficients	$b_{0i}, b_{1i}, \beta_{1i}$ , etc.	0	0	1	ID	4*NMACH	c.
		0	0	2	ID	6*NMACH	d.
		0	0	3	ID	8*NMACH	e.

Entry Format

- a. AEFACCT, ID,  $m_1, c_{l\alpha_1}, c_{l\alpha_2}, \dots, c_{l\alpha_{NSTRIPI}}$
  - b. AEFACCT, ID,  $m_1, c_{l\alpha_{11}}, c_{l\alpha_{21}}, \dots, c_{l\alpha_{NSTRIPI1}}, m_2, c_{l\alpha_{12}}, c_{l\alpha_{22}}, \dots, c_{l\alpha_{NSTRIPI1}}, c_{l\alpha_{NSTRIPI2}}$ , for all m on MKAEROi data entry
  - c. AEFACCT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, m_2, b_{02}, b_{12}, P_{12}, m_3$ , etc.
  - d. AEFACCT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, m_2$ , etc.
  - e. AEFACCT, ID,  $m_1, b_{01}, b_{11}, \beta_{11}, b_{21}, \beta_{21}, b_{31}, \beta_{31}, m_2$  etc.
8. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.



## PAERO5 Aerodynamic Panel Property

Defines properties of each strip element for Piston theory.

### Format:

1	2	3	4	5	6	7	8	9	10
PAERO5	PID	NALPHA	LALPHA	NXIS	LXIS	NTAUS	LTAUS		
	CAOC1	CAOC2	CAOC3	CAOC4	CAOC5				

### Example:

PAERO5	7001	1	702	1	701	1	700		
	0.0	0.0	5.25	3.99375	0.0				

Describer	Meaning
-----------	---------

PID	Property identification number. (Unique Integer > 0)
NALPHA	Number of angle of attack ( $\alpha$ ) values to be input for each Mach number ( $m_i$ ) on the MKAERO1 or MKAERO2 entry. (Integer > 0)

NALPHA	Meaning
1	$\alpha$ is the same value for all strips; enter one value, in units of degrees, on the AEFACCT entry for each Mach number.
Number of Strips	$\alpha$ is different for each strip; enter $\alpha$ 's, in units of degrees, in the following order: $m_1, \alpha_1, \alpha_2, \dots, m_2, \alpha_1, \alpha_2, \dots$ , etc.

LALPHA	ID number of the AEFACCT entry that lists the $\alpha$ 's for the strips at each Mach number in the MKAERO1 or MKAERO2 entry. (Integer > 0)
NXIS	Number of dimensionless chord coordinates ( $\xi$ ) to be input. (Integer $\geq 0$ , Default = 0)

NXIS	Meaning
0	No $\xi$ 's are required. (Default)
1	$\xi$ 's are the same for all strips; enter values for one strip on the AEFACCT entry ( $\xi_h$ if NTHICK > 0, or $\xi_m$ and $\xi_h$ if NTHICK = 0)
Number of Strips	$\xi$ 's have to be input for each strip ( $\xi_{h1}, \xi_{h2}, \dots, \xi_{hNSPAN}$ , if NTHICK > 0, or $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSPAN}, \xi_{hNSPAN}, \beta_{hNSPAN}$ if NTHICK = 0)

- LXIS Identification number of AEFAC<sub>T</sub> entry that lists the  $\xi$  values for the strip in order indicated by values of NXIS and NTHICK. (Integer = 0 if  $c_a = 0$  and NTHICK > 0 or LXIS > 0 if  $c_a = 0$  and/or NTHICK = 0)
- NTAUS Parameter used to select the number of thickness ratio ( $\tau$ ) values to be input. (Integer  $\geq 0$ , Default = 0)

NTAUS	Meaning
0	No $\tau$ 's are required. (Default)
1	$\tau$ 's are the same for all strips; enter ( $\tau_1, \tau_{h1}, \tau_{t1}$ ) values for one strip on AEFAC <sub>T</sub> entry.
Number of Strips	$\tau$ 's must be input for each strip on an AEFAC <sub>T</sub> entry in the following order:  ( $\tau_1, \tau_{h1}, \tau_{t1}, \tau_2, \tau_{h2}, \tau_{t2}, \dots, \tau_{NSPAN}, \tau_{hNSPAN}, \tau_{tNSPAN}$ )

- LTAUS Identification number of AEFAC<sub>T</sub> entry that lists the  $\tau$  values for the strips. (Integer = 0 or blank if NTAUS = 0, LTAUS > 0 if NTAUS > 0)
- CAOCi  $c_a/c$  = control surface chord divided by the strip chord. (Real  $\geq 0.0$ )

**Remarks:**

1. The continuation entry is required for Piston theory with one entry (CAOCi) per strip.
2. Embedded blank fields are not allowed on the continuation entry.
3. If CAOCi = 0.0, there is no control surface.
4. [Table 9-32](#) lists the thickness data input and AEFAC<sub>T</sub> entry format used for Piston theory.

Table 9-32 Thickness Data Input and AEFACT Entry Format for Piston Theory

Type of Input	Parameter Combinations						Number of Words	Entry Format Index
	CAOG	NGHICK	NXIS	LXIS	NTAUS	LTAUS		
No control surfaces, Integrals input are same for all strips	0.	ID <sup>(a)</sup>	0	0	0	0	6	a.
With control surfaces, Integrals input, same hinge on all strips	≠ 0	ID <sup>(b)</sup>	1	ID <sup>(c)</sup>	0	0	12 1	b. c.
With control surfaces, Integrals input, variable hinge	≠ 0	ID <sup>(b)</sup>	NSTRIP	ID <sup>(d)</sup>	0	0	12 NSTRIP	b. d.
No control surfaces, thickness inputs are same for all strips	0.0	0	1	ID <sup>(f)</sup>	1	ID <sup>(e)</sup>	3 2	e. f.
With control surfaces, thickness inputs are same for all strips	≠ 0.0	0	1	ID <sup>(f)</sup>	1	ID <sup>(e)</sup>	3 2	e. f.
With control surfaces, thickness inputs vary for strips	≠ 0.0	0	NSTRIP	ID <sup>(h)</sup>	NSTRIP	ID <sup>(g)</sup>	3*NSTRIP 2*NSTRIP	g. h.

Entry Format

- a. AEFACT, ID,  $I_1, I_2, I_3, I_4, I_5, I_6$
- b. AEFACT, ID,  $I_1, \dots, I_6, J_1, \dots, J_6, I_1, I_2, I_3, I_4, I_5, I_6$
- c. AEFACT, ID,  $\xi_h$
- d. AEFACT, ID,  $\xi_{h1}, \xi_{h2}, \xi_{h3}, \dots, \xi_{hNSTRIP}$
- e. AEFACT, ID,  $\tau_m, \tau_h, \tau_t$
- f. AEFACT, ID,  $\xi_m, \xi_h$
- g. AEFACT, ID,  $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$   
 $\tau_{m1}, \tau_{h1}, \tau_{t1}, \tau_{m2}, \tau_{h2}, \tau_{t2}, \dots, \tau_{mNSTRIP}, \tau_{hNSTRIP}, \tau_{tNSTRIP}$
- h. AEFACT, ID,  $\xi_{m1}, \xi_{h1}, \xi_{m2}, \xi_{h2}, \dots, \xi_{mNSTRIP}, \xi_{hNSTRIP}$

5. The following table lists the angle-of-attack distribution and AEFACT entry formats used for Piston theory.

Type of Distribution	Parameter Combinations		Number of Words	Entry Format Index
	NALPHA	LALPHA		
Equal angle of attack on all strips	1	ID	2*NMACH	a.
Unequal angle of attack	NSTRIP	ID	(1 + NSTRIP) * NMACH	b.

Entry Format

- a. AEFACT, ID,  $m_1$ ,  $\alpha_1$ ,  $m_2$ ,  $\alpha_2$ , ... ,
- b. AEFACT, ID,  $m_1$ ,  $\alpha_{11}$ ,  $\alpha_{21}$ ,  $\alpha_{31}$ , ...,  $\alpha_{NSTRIP1}$ ,  $m_2$ ,  $\alpha_{12}$ ,  $\alpha_{22}$ , ...,  $\alpha_{NSTRIP2}$ ,  $m_2$ , etc., for all m on MKAEROi entry.
- c. A control surface rotation is positive when the trailing edge moves in the negative z-direction of the aerodynamic element coordinate system; see the *MSC Nastran: Aeroelastic Analysis User's Guide*.

## PANEL Panel Definition for Coupled Fluid-Structural Analysis

Defines one or more panels by referencing sets of grid points, elements or properties.

### Format:

1	2	3	4	5	6	7	8	9	10
PANEL	NAME1	SETID1	NAME2	SETID2	NAME3	SETID3	NAME4	SETID4	

### Example:

PANEL	BKDOOR	103							
-------	--------	-----	--	--	--	--	--	--	--

Descriptor	Meaning
NAMEi	Panel label. (Character)
SETIDi	Identification number of a SET1 or SET3 entry that lists the grid points, elements or properties of the panel. On this entry, the SET1 should only be used for grid definition. (Integer > 0)

### Remarks:

1. If a set of grid points is referenced, the set must list only structural grid points.
2. If an element is assigned to a panel, it is recommended that all of its connections points belong to the same panel.
3. If a set of elements is referenced, the set must list only structural elements. The panel will consist of all grid points that are connection points of these elements.
4. If a set of property identifiers is referenced, the properties must be referenced by structural elements. The panel will consist of all grid points that are connection points of elements referencing one of the properties contained in the set.
5. NAMEi is used only for labeling the output of the panel participation factors (cf. the description of the PFMODE and PFPANEL Case Control commands).

**PARAM**      Parameter

Specifies values for parameters used in solution sequences or user-written DMAP programs.

**Format:**

1	2	3	4	5	6	7	8	9	10
PARAM	N	V1	V2						

**Example:**

PARAM	IRES	1							
-------	------	---	--	--	--	--	--	--	--

Describer	Meaning
N	Parameter name (one to eight alphanumeric characters, the first of which is alphabetic).
V1, V2	Parameter value based on parameter type, as follows:

Type	V1	V2
Integer	Integer	Blank
Real, single-precision	Real	Blank
Character	Character	Blank
Real, double-precision	Double-precision real	Blank
Complex, single-precision	Real or blank	Real or blank
Complex, double-precision	Double-precision real	Double-precision real

**Remarks:**

1. See [Parameters](#) for a list of parameters used in solution sequences that may be set by the user on PARAM entries.
2. If the large field entry format is used, the second physical entry line must be present, even though fields 6 through 9 are blank except for SOL 700.
3. The first 8 characters of N must be unique. N with more than 8 characters is normally only used by SOL 700.
4. If the Bulk Data involves the use of part superelements or external superelements, the following points should be noted regarding the use of the PARAM Bulk Data entry:
  - a. PARAM entries specified in the Main Bulk Data portion of the input data apply *only to the residual and not to the part superelements or external superelements*.
  - b. PARAM entries specified in the BEGIN SUPER portion of the Bulk Data for a part superelement or an external superelement apply *only to that superelement*.

- c. The most convenient way of ensuring that PARAM entries apply not only to the residual, but also to all part superelements and external superelements is to specify such PARAM entries in Case Control, not in the Main Bulk Data. This is particularly relevant for such PARAMs as POST.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

## PARAMARC

## Parallel Domain Decomposition in Marc from SOL 600

Specifies parallel regions for domain decomposition in nonlinear analysis when Marc is executed from MSC Nastran. Used in SOL 600 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PARAMARC	ID	KIND	NPROC	IDSOLVE	ACCUR				
	IDP	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	IDP	ID8	ID9	etc.					

### Example: To create 4 parallel processes using Marc's single file input

PARAMARC	51		4	0	1.0E-4				
----------	----	--	---	---	--------	--	--	--	--

### Example: To create 2 parallel processes by specifying element numbers

PARAMARC	201	2	2						
	1	1	THRU	5000	BY	1			
	2	5001	5002	5005	5010	5012	5013	5015	
	2	5020	5030	5040	5050				

Describer	Meaning
ID	Identification number of the PARAMARC entry -- Not presently used. (Integer)
KIND	Designates how parallel domains are created. (Integer = 0 or Blank; Default = 0) 0 Parallel processing is accomplished using Marc's single file input. (Marc Version 2005 or subsequent must be used. The command line to execute Marc is changed from -np N (or -nproc N) to -nps N where N is the number of processors. The maximum number of processors for Marc is 256. Continuation lines may not be entered for KIND=0. 2 Parallel domains will be specified by the user and the continuation lines are required.
NPROC	Number of parallel processes requested. See Remark 11. (Integer; Default = 0)
IDSOLV	Inter-domain solution type. See Remark 11. (Integer; Default = 0) 0 Iterative Solver 1 Direct Solver
ACCUR	Inter-domain solve accuracy. See Remark 11. (Real; Default = 1.0E-2) For an accurate solution of difficult nonlinear models, ACCUR should be set to 1.0E-4.



Describer	Meaning
IDP	Parallel process ID for following group of elements or grid ID's. See Remark 5. (Integer > 0; Required)
ID(i)	Element number (if KIND=2). (Integer > 0; Required if KIND=2)
THRU	Enter THRU in field 4 (may be omitted if I2 and INC are not required. The word TO instead of THRU may also be used.)
ID2	Ending element number. (Integer > 0)
BY	Enter BY in field 6 (may be omitted in INC=1)
INC	Increment ID's. (Integer; Default = 1, may be negative but not zero)

**Remarks:**

1. The PARAMARC entry is recognized only when Marc is executed from within SOL 600.
2. Use of KIND=2 provides more control than does the automatic (KIND=0) option.
3. If parallel jobs are run on different computers across a network, as opposed to using multiple processors in the same box, a host file is normally needed. Consult MSC technical support to determine how to setup a host file for your computer system. Use of the host file is triggered by Bulk Data PARAM,MARHOST,Name.
4. Continuation lines should not be entered unless KIND=2.
5. The continuation entries should be entered as many times as necessary to completely define each parallel region for KIND=2.
6. The string TO may be substituted for THRU if so desired.
7. For PC Windows systems, the default type of MPI for SOL 600 is Intel MPI. The first time a parallel job is run, the user may be prompted for domain\user\_name and then for password. Since MSC Nastran is a batch process, the user will not normally see the prompts and the job may appear to hang. If the job appears to hang, carefully enter the following information in exactly the same way you enter it to login into your PC:
 

```
domain\user_name
password
```
8. If multiple computers are used across a network, all computers must normally be the same type of computer, run the same operating system, be in the same domain, have the same user name and passwords. Also, a host file is required to describe the machines to be used. Further details are provided in the *SOL 600 Parallel Processing User's Guide*.
9. See PARAM,MRPARALL for additional notes concerning SOL 600 parallel processing.
10. If running on a PC, see PARAM,MARCMPII for options to keep the small MPI service (for example, ismpd.exe for Intel MPI) running or not after the Marc portion of the job has completed.
11. Parallel processing for SOL 600 is accomplished using domain decomposition. A solution of each domain is performed using its own processor. When each of the individual processor solutions converge, the overall solution at each increment is put together from the domains and iterated until convergence occurs. Variables IDSOLV and ACCUR control how this final iteration is performed



and its accuracy. `IDSOLV=0` (iterative solver) will usually be faster than `IDSOLV=1` (direct solver), however it is sometimes necessary that `ACCUR` be reduced to `1.0E-3` or `1.0E-4` to obtain sufficient accuracy for difficult nonlinear models, particularly those with contact, post-buckling, or large strain plasticity.

**PAXISYM**

## Properties of Axisymmetric Line Element - SOL 400

Defines the properties of axisymmetric line elements.

**Format:**

1	2	3	4	5	6	7	8	9	10
PAXISYM	PID	MID		T1	T2	ANAL			

**Example:**

PAXISYM	98	17		0.1					
---------	----	----	--	-----	--	--	--	--	--

Describer	Meaning
PID	Property identification number of a CAXISYM entry. (Integer > 0)
MID	Identification number of a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entry. Identification number of a MAT4 or MAT5 for heat transfer. See Remark 3. (Integer > 0)
T1	Thickness at G1. (Real > 0)
T2	Thickness at G2. (Real ≥ 0)
ANAL	Analysis type, IS=structural, IH=heat, ISH=structural-heat (CHAR Default=ISH)

**Remarks:**

1. PAXISYM identification entries should be unique with respect to all other property entries.
2. For elements with only two grids, only T1 is applicable. For elements with three grids, if T2 is blank or 0.0 then constant thickness is assumed.
3. The MID entry may point to MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE entries for structures and MAT4 or MAT5 entries for heat transfer. The tables below show associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials					
MAT1	MAT2	MAT3	MAT8	MATORT	MATHE
MATVE	<MATVE>	<MATVE>	<MATVE>	<MATVE>	MATVE
MATVP	MATVP	MATVP		MATVP	
MATEP	MATEP	MATEP	MATEP	MATEP	
MATF	MATF	MATF	MATF	MATF	
MATS1		MATS3	MATS8	MATSORT	

<MATVE> refers to the ALTERNATE format for type ORTHO

### Heat Materials

MAT4

MAT5

MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries. MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The CAXISYM element uses PLOADX1 for loading.
5. The element does not support composite materials, via PCOMP or PCOMPG.

## PAXSYMH

## Linear Axisymmetric Harmonic Element Properties

Defines the properties of a linear axisymmetric harmonic element.

### Format:

1	2	3	4	5	6	7	8	9	10
PAXSYMH	PID	MID	CID	NHARM	INT				

### Example:

PAXSYMH	100	10	5						
---------	-----	----	---	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0). See Remarks 1. and 5.
MID	Identification number of MAT1 or MAT9 entry. (Integer > 0).
CID	Identification number of element coordinate system. (Integer ≥ 0; Default = 0). See Remark 2..
NHARM	Harmonic index. (Integer ≥ 0; Default = 1). See Remark 3..
INT	Integration scheme. (Integer 0, 1, 2 or 3; Default = 0). See Remark 4.

### Remarks:

1. This entry is referenced by the CQUADX and CTRIAX entries.
2. The Y-axis of CID defines the axis of symmetry while its X-Y plane defines the two-dimensional plane of the axisymmetric harmonic element. The X-axis of CID thus defines the radial direction.
3. The gyroscopic matrix is generated for the element only for the case of NHARM = 1 (the default).
4. The integration scheme INT selects the number of Gauss points used for matrix generation as indicated below:

INT	Order	Gauss Points CQUADX	Gauss Points CTRIAX
1	1x1	1	1
2	2x2	4	3
3	3x3	9	7

INT=0 uses INT=2 (no mid-side nodes) or INT=3 (mid-side nodes). The default INT=0 is recommended. The use of INT=1 will produce a singular stiffness and is meant for experimental use only.

5. PAXSYMH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PAXSYMH property entries have unique identification numbers with respect to all other property entries, else unexpected grouping results may occur. There must be uniqueness among PAXSYMH, PLPLANE and PSHELL entries.

## PBAR Simple Beam Property

Defines the properties of a simple beam element (CBAR entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PBAR	PID	MID	A	I1	I2	J	NSM		
	C1	C2	D1	D2	E1	E2	F1	F2	
	K1	K2	I12						

### Example:

PBAR	39	6	2.9		5.97				
			2.0	4.0					

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2. and 3. (Integer > 0)
A	Area of bar cross section. (Real; Default = 0.0)
I1, I2, I12	Area moments of inertia. See Figure 9-106. (Real; $I1 \geq 0.0$ , $I2 \geq 0.0$ , $I1 * I2 \geq I12^2$ ; Default = 0.0)
J	Torsional constant. See Figure 9-106. (Real; Default = $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences)
NSM	Nonstructural mass per unit length. (Real)
Ci, Di, Ei, Fi	Stress recovery coefficients. (Real; Default = 0.0)
K1, K2	Area factor for shear. See Remark 5. (Real or blank)

### Remarks:

- Both continuation entries may be omitted.
- For structural problems, MID must reference a MAT1 material entry.
- For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
- See the CBAR entry description for a discussion of bar element geometry.
- The transverse shear stiffnesses times unit length in planes 1 and 2 are  $K1 * A * G$  and  $K2 * A * G$ , respectively, where G is the shear modulus. The default values for K1 and K2 are infinite; in other words, the transverse shear flexibilities are set equal to zero. K1 and K2 are ignored if  $I12 \neq 0$ . K1 and K2 must be blank if  $A = 0.0$ .

6. The stress recovery coefficients C1 and C2, etc., are the y and z coordinates in the bar element coordinate system of a point at which stresses are computed. Stresses are computed at both ends of the bar. For conventional element, only bending components of strain and stress are outputted at points C1 and C2, etc. Membrane components of strain and stress are outputted as axial value. For advanced nonlinear element, the bending and membrane components of strain and stress are superposed and outputted together at the points C1 and C2, etc., there is no axial value individually.
7. For response spectra analysis on stress recovery coefficients, the CBEAM element entry should be used because bar element results will be inaccurate.
8. [Figure 9-106](#) describes the PBAR element coordinate system.

where:

$$\begin{aligned}
 I1 &= I_{zz_{elem}} \\
 I2 &= I_{yy_{elem}} \\
 I12 &= I_{zy_{elem}} \\
 J &= I_{xx_{elem}}
 \end{aligned}$$

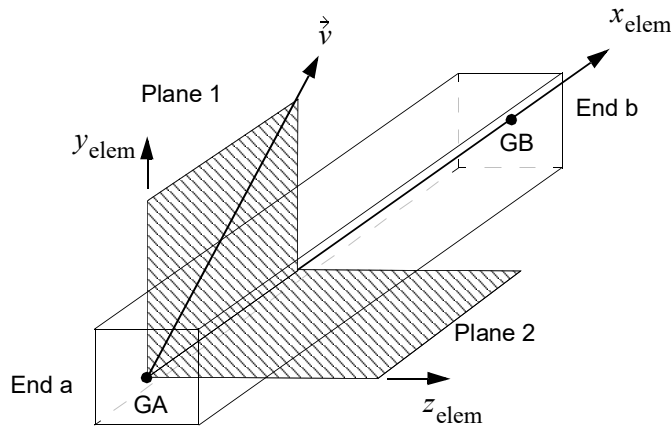


Figure 9-106 PBAR Element Coordinate System

9. For cross-sections that are not doubly symmetric, when a beam is loaded through the centroid, it may in addition to bending undergo rotation. The CBAR element, by default, does not represent this behavior because the shear center is not explicitly accounted for on a CBAR entry.

By definition, for the CBAR element, the load is applied at the centroid and not at the shear center. If load application at the shear center is desired, appropriately applied offsets can be used on the CBAR entry or the CBAR/PBAR element replaced by the CBEAM/PBEAM element which by default explicitly applies the load through the shear center.

If warping considerations and bi-moment calculations are important, the CBEAM/PBEAM element should be used.



10. Mass moment of inertial formulation has changed in Version 2003. System (398) may be used to select the formulation in pre-Version 2004 systems.
11. PBAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

## PBARL

## Simple Beam Cross-Section Property

Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.

### Format:

1	2	3	4	5	6	7	8	9	10
PBARL	PID	MID	GROUP	TYPE					
	DIM1	DIM2	DIM3	DIM4	DIM5	DIM6	DIM7	DIM8	
	DIM9	-etc.-	NSM						

### Example:

PBARL	39	6		I					
	14.	6.	.5	.5	.5	.5	.2		

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. See Remarks 6. and 8. (Character; Default = "MSCBML0")
TYPE	Cross-section type. See Remarks 6. and 8. and Figure 9-107. (Character: "ROD", "TUBE", "TUBE2", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP="MSCBML0")
DIMi	Cross-sectional dimensions. (Real > 0.0 for GROUP = "MSCBML0")
NSM	Nonstructural mass per unit length. NSM is specified after the last DIMi. (Default = 0.0)

### Remarks:

1. For structural problems, PBARL entries must reference a MAT1 material entry.
2. PBARL is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBARL property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.
3. See CBAR entry for a discussion of bar element geometry.
4. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
5. For response spectra analysis on stress recovery coefficients, the CBEAM element should be used because results for the CBAR element will not be accurate.

6. The GROUP is associated with an FMS CONNECT statement that specifies the evaluator. A reserved GROUP name is "MSCBML0". Users may create their own cross-section types. Each of the types will require one or more subroutines to convert DIMi information to geometric property information contained on a PBAR entry and optimization information. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion on how to include a user-defined beam library.
7. A function of this entry is to derive an equivalent PBAR entry. Any sorted echo request will also cause printout and/or punch of the derived PBAR.
8. For GROUP = "MSCBML0", the cross-sectional properties, shear flexibility factors, and stress recovery points (C, D, E, and F) are computed using the TYPE and DIMi as shown in [Figure 9-107](#) through [Figure 9-110](#). The figures show the origin of the cross section, but the PBARL does not account for differences between the shear center and the neutral axis and the properties are computed relative to the neutral axis. This results in approximations for section types 'I', 'CHAN', 'T', 'CHAN1', 'T1', 'CHAN2', 'T2', 'L' and 'BOX1'. The PBEAML provides a better representation of section properties in these cases and is recommended.

The PBARL does not account for offsets between the neutral axis and the shear center. Therefore, the CHAN, CHAN1 and CHAN2 cross-sections may produce incorrect results. The PBEAML is recommended.

9. For DBOX section, the default value for DIM5 to DIM10 are based on the following rules:
  - a. DIM5, DIM6, DIM7 and DIM8 have a default value of DIM4 if not provided.
  - b. DIM9 and DIM10 have a default value of DIM6 if not provided.

**Note:** The above default value rules for DIM5 to DIM10 are not applicable to design optimization property value update.

10. The finite element formulation (FEF) utilized for arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBARL when GROUP=MSCBML0. The original beam equations which are based on thin-walled assumptions can be accessed via Bulk Data entry 'MDLPRM,TWBRBML,1'. Thus the shear stiffness factors K1 and K2 are calculated as in Remark 11. of the PBMSECT entry and the element will behave according to the Timoshenko beam theory by default. Thus the resulting CBAR will have non-infinite K1/K2 shear flexibility factors, and therefore will not behave as a Bernoulli-Euler.
11. For optimization, individual DIMx of PBARL can be selected as designed properties even with the finite element formulation.

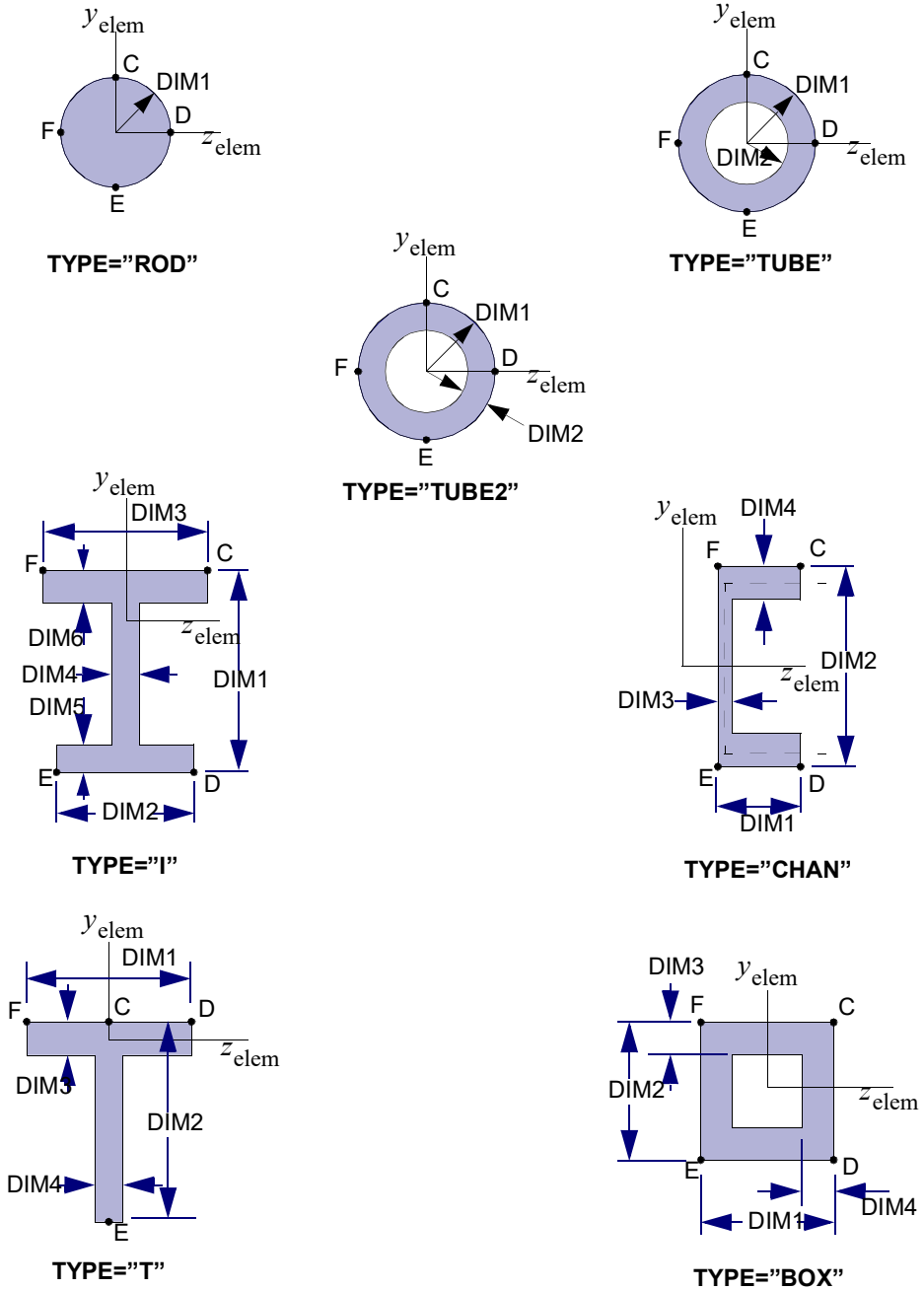
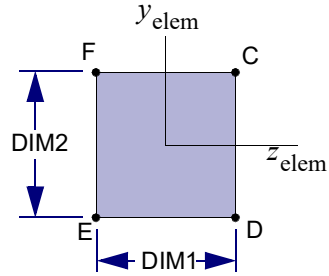
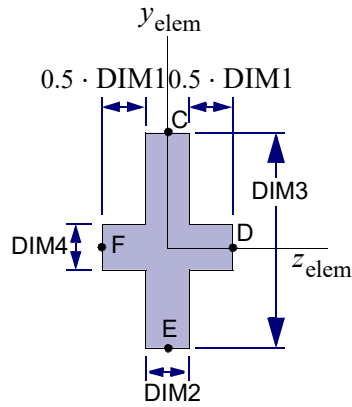


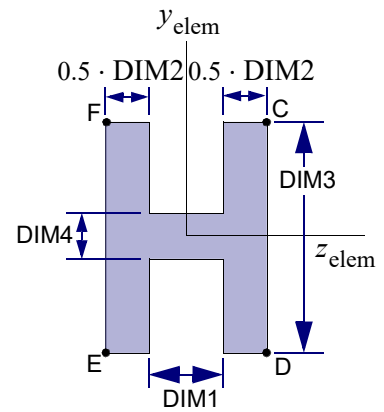
Figure 9-107 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"



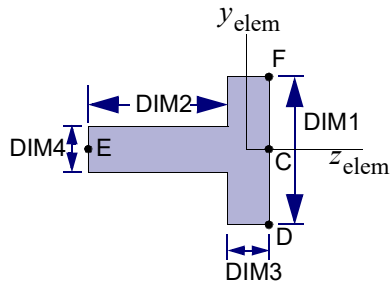
TYPE="BAR"



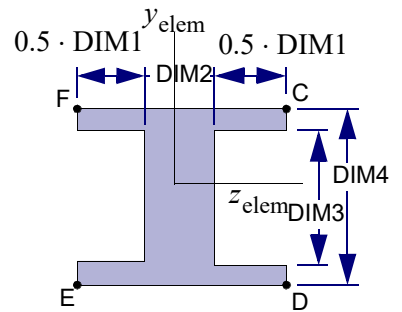
TYPE="CROSS"



TYPE="H"

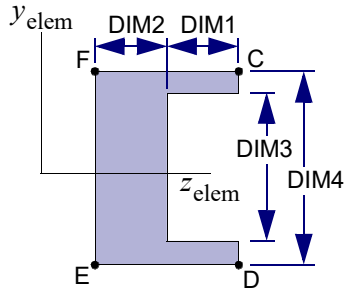


TYPE="T1"

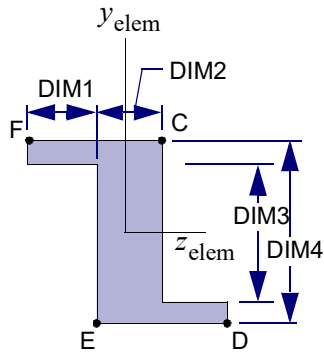


TYPE="I1"

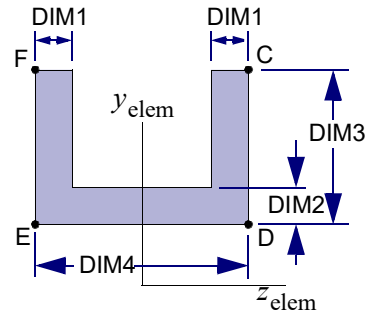
Figure 9-108 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)



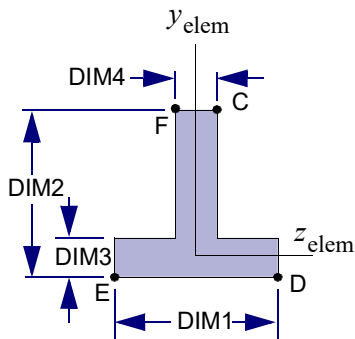
TYPE="CHAN1"



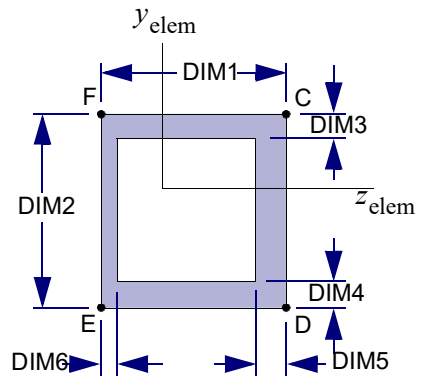
TYPE="Z"



TYPE="CHAN2"



TYPE="T2"



TYPE="BOX1"

Figure 9-109 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

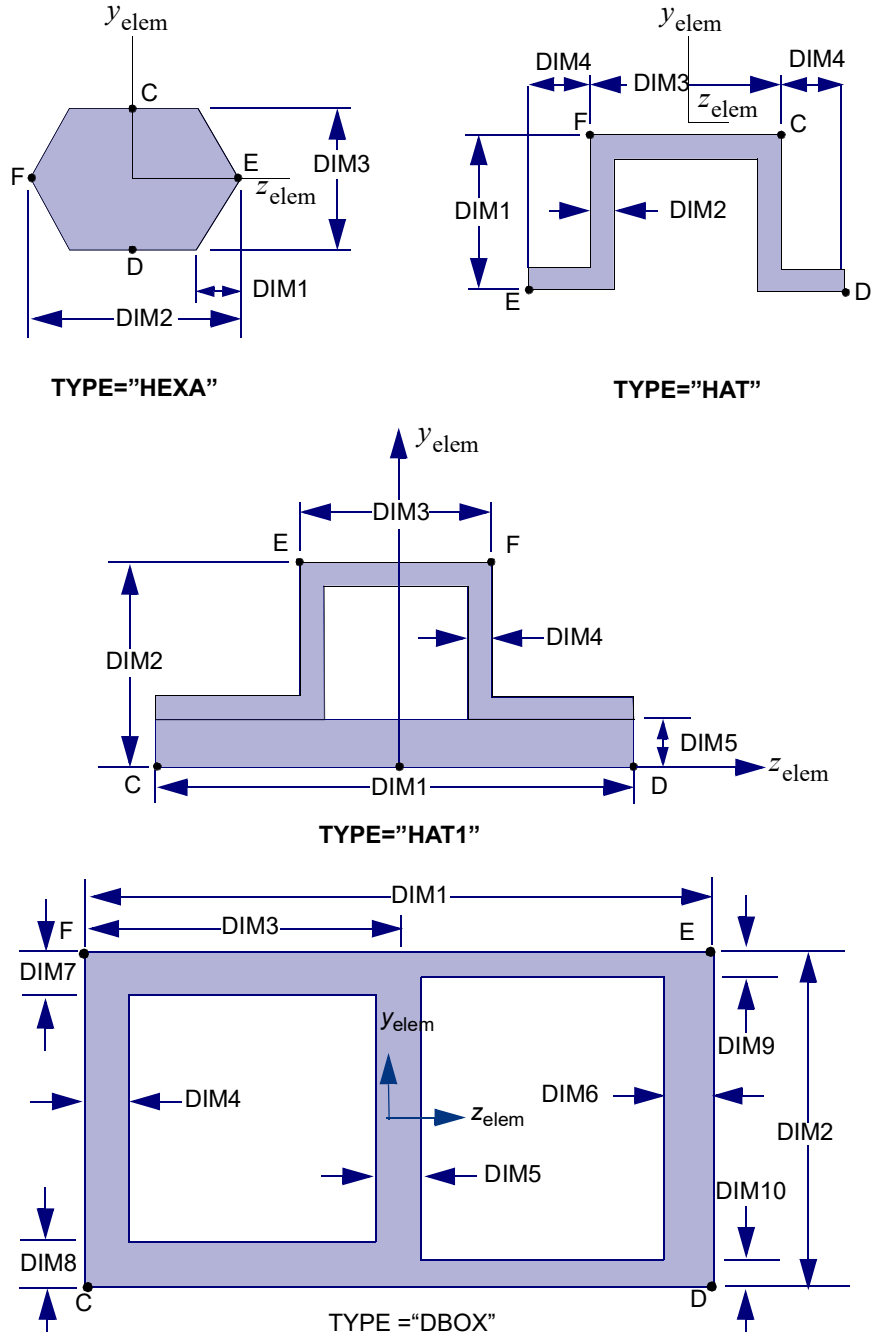


Figure 9-110 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0" (continued)

## PBARN1

## Nonlinear Property Extensions for a PBAR or PBARL Entry

Specifies additional nonlinear properties for elements that point to a PBAR or PBARL entry in SOL 400.

### Format:

1	2	3	4	5	6	7	8	9	10
PBARN1	PID	MID		SECT					
	"C2"	BEH2	INT2						

### Example:

PBARN1	29	73							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PBAR entry. (Integer > 0,)
MID	Material ID. Remark 10.(Integer ≥ 0)
SECT	Section integration. SECT = "S" a smeared cross section is used for integration. SECT = "N" a numerically integrated cross section is used. See Remark 7. (Character Default S or blank)
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 4. (Character Default BAR)
INT2	Integration scheme. See Remarks 4. and 5. (Character Default LC)

### Remarks:

1. The PID above must point to an existing PBAR or PBARL Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PBAR or PBARL entry. If > 0 it will override the MID value on the PBAR entry.
3. The MID entry may point to the MAT1 entry. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials
MAT1
MATVE
MATVP
MATEP



Implicit Structural Materials
MATF
MATS1
MATSMA

4. BEH2 refers to the nonlinear structural behavior of the BAR element. An underlined item delineates a default.

Structural Classification of Elements		
Element Structural Type	BEHAV CODE	Integration Code
Bar	BAR	<u>LC</u> LCC LS

5. Integration codes in Remark 4. are:

INT CODE	Integration Type
LC	Linear/Cubic
LCC	Linear/Cubic Closed section
LS	Linear-shear

6. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.

7. Smearred section support.

Primary BAR Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
PBAR	LC	S	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam.	Smearred section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LS	S	Linear Interpolation for axial, transverse displacements and rotations	Both Thin-Walled and Solid Sections with elastic materials. Captures transverse shear effects - useful for deep beams	Smearred section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LC or LS	N			Not Supported
	LCC	S			Not Supported
	LCC	N			Not Supported
PBARL	LC or LS	S			Not Supported
	LC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Solid Sections. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials are supported. MATHE/MATHP are not supported.

Primary BAR Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
	LS	N	Linear Interpolation for axial displacements, transverse displacements and rotations	Solid Sections. Captures transverse shear effects - useful for deep beams	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported.
	LCC	S			Not Supported
	LCC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Sections. Euler-Bernoulli Beam.	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported.

8. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
9. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
10. The structural element damping coefficient, GE, is not supported on elements which reference PBARN1.

## PBCOMP

## Beam Property (Alternate Form of PBEAM)

Alternate form of the PBEAM entry to define properties of a uniform cross-sectional beam referenced by a CBEAM entry. This entry is also used to specify lumped areas of the beam cross section for nonlinear analysis and/or composite analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
PBCOMP	PID	MID	A	I1	I2	I12	J	NSM	
	K1	K2	M1	M2	N1	N2	SYMOPT		
	Y1	Z1	C1	MID1					
	Y2	Z2	C2	MID2					
	-etc.-								

### Example:

PBCOMP	39	6	2.9						
							1		
	-0.5	1.2	0.1	18					
	0.2	0.9	0.15						

Describer	Meaning
PID	Property identification number. See Remark 1. (Integer > 0)
MID	Material identification number. See Remarks 3. and 6. (Integer > 0)
A	Area of beam cross section. (Real > 0.0)
I1	Area moment of inertia in plane 1 about the neutral axis. See Remark 7. (Real > 0.0)
I2	Area moment of inertia in plane 2 about the neutral axis. See Remark 7. (Real > 0.0)
I12	Area product of inertia. See Remark 7. (Real; Default = 0.0, but $I1 \cdot I2 - (I12)^2 > 0.0$ )
J	Torsional stiffness parameter. See Remark 7. (Real > 0.0; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real > 0.0; Default = 0.0)
K1, K2	Shear stiffness factor K in $K \cdot A \cdot G$ for plane 1 and plane 2. See Remark 5. (Real > 0.0; Default = 1.0)
M1, M2	The (y,z) coordinates of center of gravity of nonstructural mass. See the figure in the CBEAM entry description. (Real; Default = 0.0)
N1, N2	The (y,z) coordinates of neutral axis. See the figure in the CBEAM entry description. (Real; Default = 0.0)
SYMOPT	Symmetry option to input lumped areas for the beam cross section. See Figure 9-112 and Remark 8. ( $0 \leq \text{Integer} \leq 5$ ; Default = 0)

Describer	Meaning
Yi, Zi	The (y,z) coordinates of the lumped areas in the element coordinate system. See Remark 1. (Real)
Ci	Fraction of the total area for the i-th lumped area. (Real > 0.0; Default = 0.0)
MIDi	Material identification number for the i-th integration point. See Remark 6. (Integer > 0)

**Remarks:**

1. PBCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
2. The second continuation entry may be repeated 18 more times. If SYMOPT = 5 a maximum of 21 continuation entries is allowed; i.e., a maximum of 20 lumped areas may be input. If SYMOPT = 1 through 4, the total number of areas input plus the total number generated by symmetry must not exceed 20. If these are not specified, the program defaults, as usual, to the elliptically distributed eight nonlinear rods. See [Figure 9-111](#).

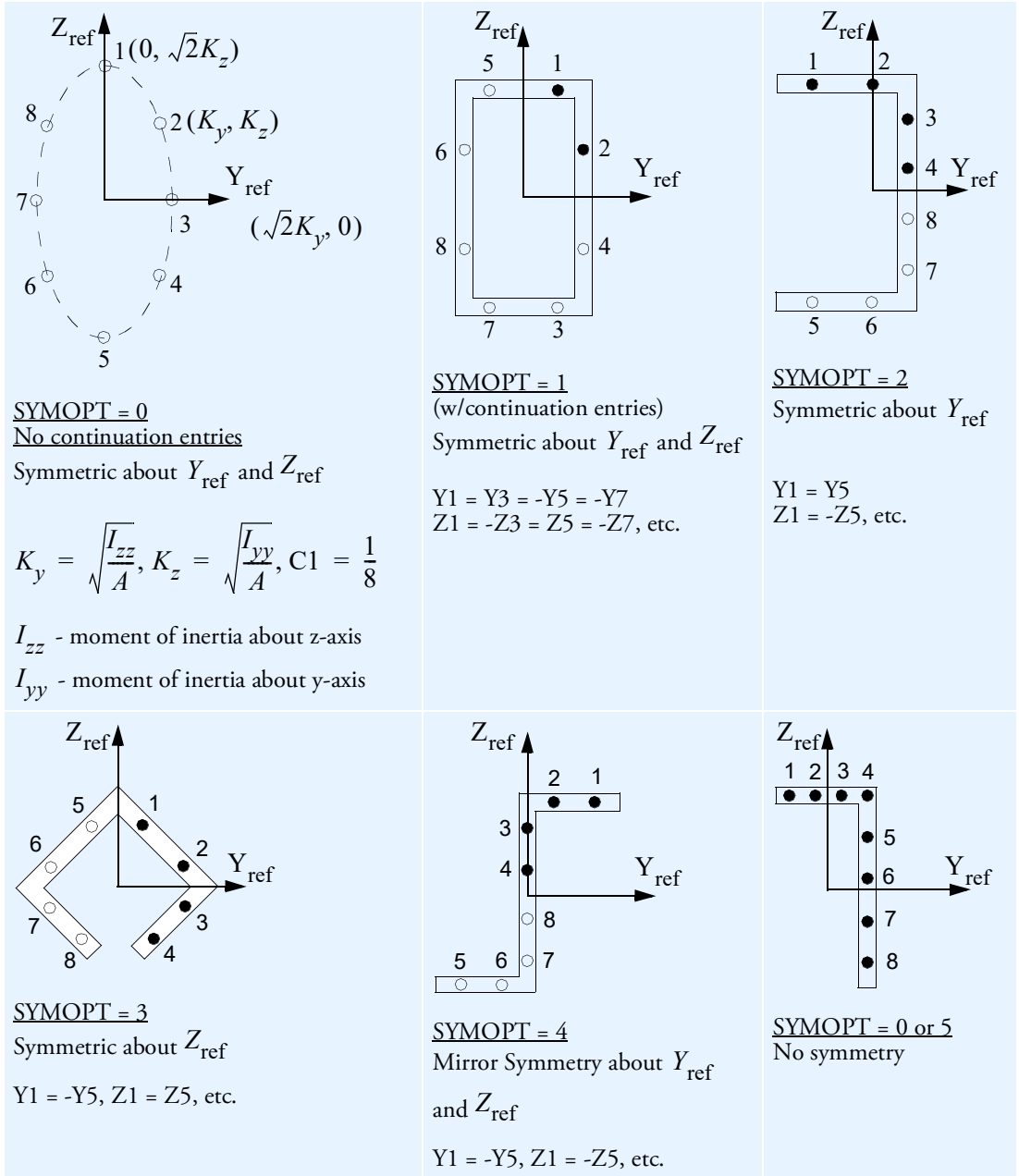


Figure 9-111 PBCOMP Entry SYMOPT Type Examples with 8 Lumped Areas

**Notes:** Integration points (lumped area) are numbered 1 through 8.

User-specified points are denoted by ● and the mirrored points are denoted by ○.

3. For structural problems, MID and MIDi must reference a MAT1 material entry. For material nonlinear analysis, the material should be perfectly plastic since the plastic hinge formulation is not valid for strain hardening. For heat transfer problems, MID and MIDi must reference a MAT4 or MAT5 material entry.
4. For the case where the user specifies I1, I2 and I12 on the parent entry, the stress-output location may also be specified on continuation entries. The (y,z) coordinates specified on these entries will serve as stress output locations with the corresponding Ci's set to 0. Stress output is provided at the first four lumped area locations only. If one of the symmetry options is used and fewer than four lumped areas are input explicitly, the sequence of output locations in the imaged quadrants is shown in [Figure 9-111](#). For one specific example in the model shown in Remark 8. ([Figure 9-112](#)), output can be obtained at points 1 and 2 and in the image points 3 and 4.
5. Blank fields for K1 and K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear stiffness becomes rigid and the transverse shear flexibilities are set to 0.0.
6. The values  $E_0$  and  $G_0$  are computed based on the value of MID on the parent entry. MID will follow the same symmetry rules as Ci depending on the value of SYMOPT. If the MIDi field on a continuation entry is blank, the value will be that of MID on the parent entry. MIDi values may be input on continuations without the corresponding Yi, Zi, and Ci values to allow different stress-strain laws.
7. If the lumped cross-sectional areas are specified, fields I1, I2, and I12 will be ignored. These and other modified values will be calculated based on the input data (Yi, Zi, Ci, MIDi) as follows:

$$y_{NA} = \frac{\sum_{i=1}^n Y_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$z_{NA} = \frac{\sum_{i=1}^n Z_i C_i E_i}{\sum_{i=1}^n C_i E_i}$$

$$\bar{A} = A \sum_{i=1}^n \frac{C_i E_i}{E_o}$$

$$\bar{I}_1 = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})^2}{E_o}$$

$$\bar{I}_2 = A \sum_{i=1}^n \frac{C_i E_i (Z_i - z_{NA})^2}{E_o}$$

$$\bar{I}_{12} = A \sum_{i=1}^n \frac{C_i E_i (Y_i - y_{NA})(Z_i - z_{NA})}{E_o}$$

$$J = J \sum_{i=1}^n \frac{C_i G_i}{G_o}$$

where n is the number of lumped cross-sectional areas specified.

- For a doubly symmetric section (SYMOPT = 1), if the lumped areas are specified on either axis, the symmetry option will double the areas. For example, for the section shown in [Figure 9-112](#), points 2 and 4 are coincident and so are points 6 and 8. In such cases, it is recommended that users input the value of area as half of the actual value at point 2 to obtain the desired effect.

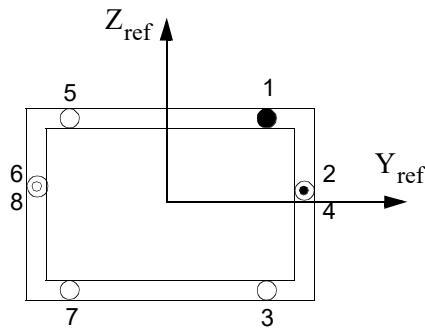


Figure 9-112 Doubly Symmetric PBCOMP Section



9. For SYMOPT =0, 5, or blank at least three  $Y_i$  and one  $Z_i$  coordinates must be nonzero. All of the points (input plus mirrored) should not lie on a straight line. All of the points (input plus mirrored) should not lie on a straight line or a FATAL message will be issued.

## PBEAM

## Beam Property

Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams.

### Format:

1	2	3	4	5	6	7	8	9	10
PBEAM	PID	MID	A(A)	I1(A)	I2(A)	I12(A)	J(A)	NSM(A)	
	C1 (A)	C2 (A)	D1 (A)	D2 (A)	E1 (A)	E2 (A)	F1 (A)	F2 (A)	

The next two continuations are repeated for each intermediate station as described in Remark 6. and SO and X/XB must be specified.

	SO	X/XB	A	I1	I2	I12	J	NSM	
	C1	C2	D1	D2	E1	E2	F1	F2	

The last two continuations are:

	K1	K2	S1	S2	NSI(A)	NSI(B)	CW(A)	CW(B)	
	M1(A)	M2(A)	M1(B)	M2(B)	N1(A)	N2(A)	N1(B)	N2(B)	

### Example:

Tapered beam with A=2.9 at end A and A=5.3 at end B.

PBEAM	39	6	2.9	3.5	5.97				
			2.0	-4.0					
	YES	1.0	5.3	56.2	78.6				
			2.5	-5.0					
			1.1		2.1		0.21		
					0.5		0.0		

Describer	Meaning	Default Values
PID	Property identification number. (Integer > 0)	Required
MID	Material identification number. See Remarks 1. and 3. (Integer > 0)	Required
A(A)	Area of the beam cross section at end A. (Real > 0.0)	Required
I1(A)	Area moment of inertia at end A for bending in plane 1 about the neutral axis. See Remark 10. (Real > 0.0)	Required
I2(A)	Area moment of inertia at end A for bending in plane 2 about the neutral axis. See Remark 10. (Real > 0.0)	Required

Describer	Meaning	Default Values
I12(A)	Area product of inertia at end A. See Remark 10. (Real, but $I1 \cdot I2 - (I12)^2 > 0.0$ )	0.0
J(A)	Torsional stiffness parameter at end A. See Remark 10. (Real $\geq 0.0$ but $> 0.0$ if warping is present)	Default = $\frac{1}{2}(I_1 + I_2)$ for SOL 600 and 0.0 for all other solution sequences
NSM(A)	Nonstructural mass per unit length at end A. (Real)	0.0
Ci(A), Di(A)	The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see the diagram following the remarks) at end A for stress data recovery. (Real)	y = z = 0.0
Ei(A), Fi(A)		
SO	Stress output request option. See Remark 9. (Character)  "YES" Stresses recovered at points Ci, Di, Ei, and Fi on the next continuation.  "YESA" Stresses recovered at points with the same y and z location as end A.  "NO" No stresses or forces are recovered.	Required*
X/XB	Distance from end A in the element coordinate system divided by the length of the element See Figure 9-113 in Remark 10. (Real, $0.0 < x/xb \leq 1.0$ )	Required* See Remark 6.
A, I1, I2, I12, J, NSM Ci, Di, Ei, Fi	Area, moments of inertia, torsional stiffness parameter, and nonstructural mass for the cross section located at x. (Real; J > 0.0 if warping is present.)  The y and z locations (i = 1 corresponds to y and i = 2 corresponds to z) in element coordinates relative to the shear center (see Figure 9-113 in Remark 10.) for the cross section located at X/XB. The values are fiber locations for stress data recovery. Ignored for beam p-elements. (Real)	See Remark 7.
K1, K2	Shear stiffness factor K in $K \cdot A \cdot G$ for plane 1 and plane 2. See Remark 12. (Real)	1.0, 1.0
S1, S2	Shear relief coefficient due to taper for plane 1 and plane 2.	0.0, 0.0
NSI(A), NSI(B)	Nonstructural mass moment of inertia per unit length about nonstructural mass center of gravity at end A and end B. See Figure 9-113. (Real)	0.0, same as end A

Describer	Meaning	Default Values
CW(A), CW(B)	Warping coefficient for end A and end B. See Remark 11. (Real)	0.0, same as end A
M1(A), M2(A), M1(B), M2(B)	(y,z) coordinates of center of gravity of nonstructural mass for end A and end B. See Figure 9-113. (Real)	0.0 (no offset from shear center), same values as end A
N1(A), N2(A), N1(B), N2(B)	(y,z) coordinates of neutral axis for end A and end B. See Figure 9-113. (Real)	0.0 (no offset from shear center), same values as end A

### Remarks:

- For structural analysis, MID must reference a MAT1 material entry (SOL 600). The beam may be described by any valid stress-strain law. A plastic hinge is not used for SOL 600 and SOL 700; instead, a standard nonlinear analysis is performed
- For material nonlinear analysis, MID may also reference a MATS1 entry, but the material properties must be defined as elastic-perfectly plastic; for example, H = 0.0 on the MATS1 entry. Also, only one-eighth of the length at each end of the element abides by material nonlinear law; i.e., the element is modeled as a plastic hinge. Any other type of material property specification may yield inaccurate results.
- For heat transfer analysis, MID must reference a MAT4 or MAT5 material entry.
- If no stress data at end A is to be recovered and a continuation with the SO field is specified, then the first continuation entry, which contains the fields C1(A) through F2(A), may be omitted.
- If SO is “YESA” or “NO”, the third continuation entry, which contains the fields C1 through F2, must be omitted. If SO is “YES”, the continuation for Ci, Di, Ei, and Fi must be the next entry. The blank fields are defaulted to 0.0 on these continuations.
- The rules for the continuations entries are:
  - The second and third continuation entries, which contain fields SO through F2, may be repeated nine more times for intermediate X/XB values for linear beam elements. The order of these continuation pairs is independent of the X/XB value; however, one value of X/XB must be 1.0, corresponding to end B. The intermediate stress output requests will be ignored in the nonlinear solution sequences (SOLs 106 and 129).
  - The value of X/XB must be unique among the stations of a PBEAM. Duplication of X/XB is not permitted.
  - The fourth and fifth continuation entries, which contain fields K1 through N2(B), are optional and may be omitted if the default values are appropriate.
- If any fields 4 through 9 are blank on the continuation with the value of X/XB = 1.0, then the values for A, I1, I2, I12, J and NSM are set to the values given for end A. For the continuations that have intermediate values of X/XB between 0.0 and 1.0 and use the default option (any of the fields 4 through 9 are blank), a linear interpolation between the values at ends A and B is performed to obtain the missing section properties.

8. Blank fields for K1, K2 are defaulted to 1.0. If a value of 0.0 is used for K1 and K2, the transverse shear flexibilities are set to 0.0 and field G on the MAT1 entry selected by MID must be nonzero.
9. If end B forces are desired and station data are input, use “YES” or “YESA” in the SO field of the record with X/XB=1. If station data are not input you will get end B forces if forces are requested.
10. Figure 9-113 describes the PBEAM element coordinate system.

$I1 = I_{(zz)_{na}}$	$N1(A) = y_{na}$	$N1(B) = y_{nb}$
$I2 = I_{(yy)_{na}}$	$N2(A) = z_{na}$	$N2(B) = z_{nb}$
$I12 = I_{(zy)_{na}}$	$M1(A) = y_{ma}$	$M1(B) = y_{mb}$
$J = I_{(xx)_{na}}$	$M2(A) = z_{ma}$	$M2(B) = z_{mb}$

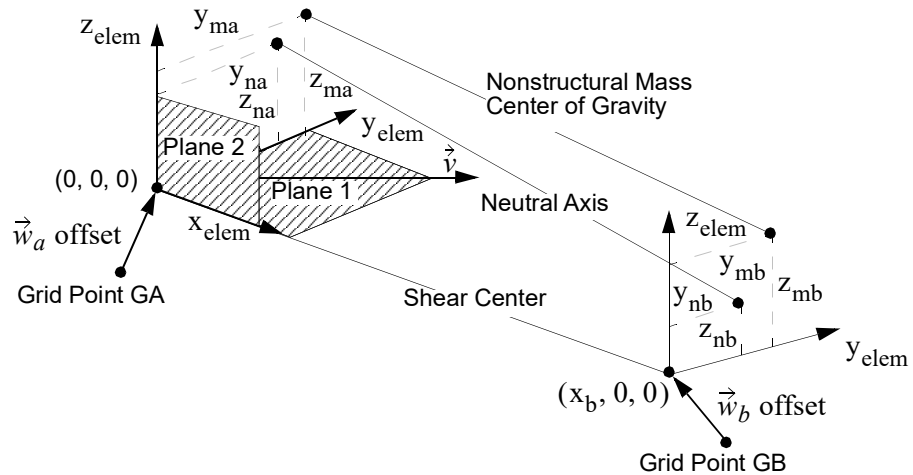


Figure 9-113 PBEAM Element Coordinate System

11. The warping coefficient  $CW$  is represented in the following differential equation for the torsion of a beam about the axis of the shear centers:

$$G \frac{d}{dx} \left( J \frac{d\theta}{dx} \right) - E \frac{d^2}{dx^2} \left( CW \frac{d^2\theta}{dx^2} \right) = m$$

where:

G = shear modulus  
 J = torsional stiffness  
 E = Young's modulus

$\theta$  = angle of rotation at any cross-section

$m$  = applied torsional moment per unit length

**Note:** CW has units of (length)<sup>6</sup>.

12. The shear stiffness factors  $K_1$  and  $K_2$  adjust the effective transverse shear cross-section area according to the Timoshenko beam theory. Their default values of 1.0 approximate the effects of shear deformation. To neglect shear deformation (i.e., to obtain the Bernoulli-Euler beam theory), the values of  $K_1$  and  $K_2$  should be set to 0.0.
13. In nonlinear analysis the location of the 8 plastic rods is the same on the PBEAM entry as it is on the SYMOPT field on the PBCOMP entry when SYMOPT = 0. Please see the [Figure 9-111](#) for more information on the SYMOPT field.
14. For SOL 600, for structural analysis, MID must reference a MAT1 material entry.
15. RC network solver only supports constant cross section beam for thermal analysis.
16. PBEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

**PBEAM3****Three-node Beam Property**

Defines the properties of a three-node beam element (CBEAM3 entry).

**Format:**

1	2	3	4	5	6	7	8	9	10
PBEAM3	PID	MID	A(A)	IZ(A)	IY(A)	IYZ(A)	J(A)	NSM(A)	
	CY(A)	CZ(A)	DY(A)	DZ(A)	EY(A)	EZ(A)	FY(A)	FZ(A)	
	SO(B)		A(B)	IZ(B)	IY(B)	IYZ(B)	J(B)	NSM(B)	
	CY(B)	CZ(B)	DY(B)	DZ(B)	EY(B)	EZ(B)	FY(B)	FZ(B)	
	SO(C)		A(C)	IZ(C)	IY(C)	IYZ(C)	J(C)	NSM(C)	
	CY(C)	CZ(C)	DY(C)	DZ(C)	EY(C)	EZ(C)	FY(C)	FZ(C)	
	KY	KZ	NY(A)	NZ(A)	NY(B)	NZ(B)	NY(C)	NZ(C)	
	MY(A)	MZ(A)	MY(B)	MZ(B)	MY(C)	MZ(C)	NSIY(A)	NSIZ(A)	
	NSIYZ(A)	NSIY(B)	NSIZ(B)	NSIYZ(B)	NSIY(C)	NSIZ(C)	NSIYZ(C)	CW(A)	
	CW(B)	CW(C)	STRESS						
	WC(A)	WYC(A)	WZC(A)	WD(A)	WYD(A)	WZD(A)	WE(A)	WYE(A)	
	WZE(A)	WF(A)	WYF(A)	WZF(A)	WC(B)	WYC(B)	WZC(B)	WD(B)	
	WYD(B)	WZD(B)	WE(B)	WYE(B)	WZE(B)	WF(B)	WYF(B)	WZF(B)	
	WC(C)	WYC(C)	WZC(C)	WD(C)	WYD(C)	WZD(C)	WE(C)	WYE(C)	
	WZE(C)	WF(C)	WYF(C)	WZF(C)					

**Example:**

PBEAM3	1010	2	2.9	3.5	5.97			1.0	
	0.2	3.0	-1.2	2.6	2.0	0.5			
	YES		1.0	23.6	34.7				
	1.1	3.2							
	YESA		3.2	2.1	3.2			1.0	
	0.8		0.5						
	0.9	1.0		1.5					
		1.0							

Descriptor	Meaning
PID	Property identification number. (Integer > 0; Required)
MID	Material identification number. See Remark 1. (Integer > 0; Required)
A(A)	Area of the beam cross-section at end A. (Real > 0.0; Required)
IZ(A)	Area moment of inertia at end A about local z-axis and the neutral axis. (Real > 0.0; Required)

Describer	Meaning
IY(A)	Area moment of inertia at end A about local y-axis and the neutral axis. (Real > 0.0; Required)
IYZ(A)	Area product of inertia at end A about local y- and z-axes and the neutral axis. If y- and z- axes are principal axes, then IYZ(A)=0.0. (Real, but $I_y \cdot I_z - I_{yz}^2 > 0.0$ ; Default = 0.0)
J(A)	Torsional stiffness parameter at end A. (Real > 0.0; Default = IZ+IY)
NSM(A)	Nonstructural mass per unit length at end A. (Real; Default = 0.0)
Ci(j), Di(j) Ei(j), Fi(j)	The local y and z coordinates ( $i=Y, Z$ ) at point $j$ ( $j=A, B, C$ ), used for stress output. (Real; Default = 0.0)
A(j), IZ(j), IY(j) IYZ(j), J(j), NSM(j)	Area, moments of inertia, torsional stiffness parameter and nonstructural mass for the cross-section at $j$ ( $j=B, C$ ). (Real; See Remark 2.)
SO(j)	Stress output request option at $j$ ( $j=B, C$ ). (Character; Default = "YESA") "YES"      Stresses are recovered at $C_i, D_i, E_i,$ and $F_i$ on the next continuation. "YESA"     Stresses are recovered at points with the same ( $y, z$ ) location at end A
KY, KZ	Shear effectiveness factors for local y- and z-directions. (Real > 0.0, Default = 1.0)
NY(j), NZ(j)	Local ( $y, z$ ) coordinates of neutral axis for $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j = B, C$ )
MY(j), MZ(j)	Local ( $y, z$ ) coordinates of nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B,C$ )
NSIY(j), NSIZ(j)	Nonstructural mass moments of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$ )
NSIYZ(j)	Nonstructural mass product of inertia per unit length about local y and z-axes, respectively, with regard to the nonstructural mass center of gravity at $j$ ( $j=A, B, C$ ). (Real, Default = 0.0 at end A and same values as end A for $j=B, C$ )
CW(j)	Warping coefficient at $j$ ( $j=A, B, C$ ). (Real $\geq 0.0$ ; Default = 0.0 at end A; same values as end A for $j = B, C$ )
STRESS	Location selection for stress, strain and force output. (Character; Default = "GRID"; See Remark 3.)
Wi(j)	Values of warping function at stress recovery points $i = C, D, E$ and $F$ , at location $j=A, B,$ and $C$ . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$ )
WYi(j), WZi(j)	Gradients of warping function in the local ( $y, z$ ) coordinate system at stress recovery points $i=C, D, E,$ and $F$ , at location $j=A, B,$ and $C$ . (Real; Default = 0.0 at end A and same values as end A for $j=B, C$ .)



**Remarks:**

1. For structural analysis, MID must reference a MAT1, MAT2 or MAT8 material entry.
2. If any fields 4 through 9, for values of A, IZ, IY, IYZ, J and NSM at end B or C, are blank, then those values for end B or C are set to the values given for end A.
3. If STRESS="GRID", then the stresses, strains and forces are recovered at A, B and C. If STRESS="GAUSS", then the stresses, strains and forces are recovered at Gauss integration points,  $\xi = \{ 1/\sqrt{3}, 1/\sqrt{3}, 0 \}$ . The beam cross-section properties at these points are interpolated from those at A, B and C.
4. If all fields of  $W_i(j)$ ,  $WY_i(j)$  and  $WZ_i(j)$  ( $i=C, D, E, F$  and  $j=A, B, C$ ), are left blank, both stresses and strains due to the warping effect will not be recovered at the stress recovery points.
5. PBEAM3 is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAM3 property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
6. When PBMSECT is used the resulting PBEAM3 image printed after the:

\*\*\* USER INFORMATION MESSAGE 4379 (IFP9B)

THE USER SUPPLIED PBMSECT BULK DATA ENTRIES ARE REPLACED BY THE FOLLOWING PBEAM3 ENTRIES.

contains special composite related data and will look different from the above entry description.

## PBEAMD

PBEAMD usage is no longer recommended and will be removed in a future version. Use PBEAM, PBDISCR, PBSPOT, PBEAM71 instead.

## PBEAML Beam Cross-Section Property

Defines the properties of a beam element by cross-sectional dimensions.

### Format:

(Note: n = number of dimensions and m = number of intermediate stations)

1	2	3	4	5	6	7	8	9	10
PBEAML	PID	MID	GROUP	TYPE					
	DIM1(A)	DIM2(A)	-etc.-	DIMn(A)	NSM(A)	SO(1)	X(1)/XB	DIM1(1)	
	DIM2(1)	-etc.-	DIMn(1)	NSM(1)	SO(2)	X(2)/XB	DIM1(2)	DIM2(2)	
	-etc.-	DIMn(2)	NSM(m)	-etc.-	SO(m)	X(m)/XB	DIM1(m)	-etc.-	
	DIMn(m)	NSM(m)	SO(B)	1.0	DIM1(B)	DIM2(B)	-etc.-	DIMn(B)	
	NSM(B)								

### Example:

PBEAML	99	21		T					
	12.	14.8	2.5	2.6		NO	0.4	6.	
	7.	1.2	2.6		YES	0.6	6.	7.8	
	5.6	2.3		YES					

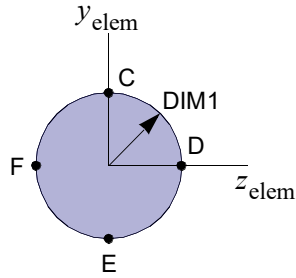
Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GROUP	Cross-section group. (Character; Default = "MSCBML0")
TYPE	Cross-section shape. See Remark 4. (Character: "ROD", "TUBE", "TUBE2", "L", "I", "CHAN", "T", "BOX", "BAR", "CROSS", "H", "T1", "I1", "CHAN1", "Z", "CHAN2", "T2", "BOX1", "HEXA", "HAT", "HAT1", "DBOX" for GROUP = "MSCBML0")
DIMi(j)	Cross-section dimensions at end A, intermediate station j and end B. (Real > 0.0 for GROUP = "MSCBML0")
NSM(j)	Nonstructural mass per unit length. (Default = 0.0)
SO(j),SO(B)	Stress output request option for intermediate station j and end B. (Character; Default = "YES")
YES	Stresses and forces are recovered at all points shown under TYPE figures below at points labeled C, D, E, F

Describer	Meaning
	NO No stresses or forces are recovered.
X(j)/XB	Distance from end A to intermediate station j in the element coordinate system divided by the length of the element. (REAL, 0.0<x(j)/xb ≤1.0, default=1.0)

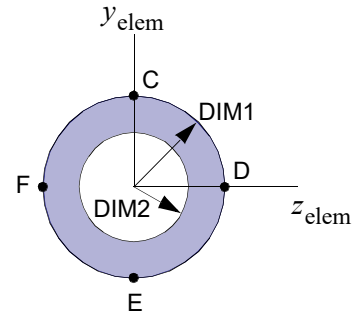
**Remarks:**

1. For structural problems, PBEAML entries must reference a MAT1 material entry.
2. PBEAML is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEAML property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.
3. For heat-transfer problems, the MID must reference a MAT4 or MAT5 material entry.
4. See the PBEAM entry description for a discussion of beam-element geometry.
5. If any of the fields NSM(B), DIMi(B) are blank on the continuation entry for End B, the values are set to the values given for end A. For the continuation entries that have values of X(j)/XB between 0.0 and 1.0 and use the default option (blank field), a linear interpolation between the values at ends A and B is performed to obtain the missing field.
6. The GROUP is associated with a FMS CONNECT statement, which specifies the evaluator. A reserved GROUP name is "MSCBML0". Users may create their own cross-section types. Each of the types will require a one or more subroutines to convert DIMi information to geometric property information contained on a PBEAM entry. See [Building and Using the Sample Programs](#) in the *MSC Nastran Utilities Guide* for a discussion of how to include a user-defined beam library.
7. For GROUP = "MSCBML0", the cross-sectional properties, shear flexibility factors and stress recovery points are computed using the TYPE and DIMi as shown in [Figure 9-114](#). The element coordinate system is located at the shear center.
8. A function of this entry is to derive an equivalent PBEAM entry. Any sorted echo request will also cause printout and/or punch of the derived PBEAM.
9. Beams can have no more than 14 dimensions per station. The total number of dimensions at all stations must be less than 200. The transfer of data with the beam server is limited to 4000 words. None of these limits are exceeded with the MSC beam library, but a user defined beam library could.
10. The finite element formulation (FEF) utilized for the arbitrary beam cross section is selected as the default method for computing sectional properties for all supported cross section types of PBEAML when GROUP=MSCBML0. The original beam equations, which are based on thin-walled assumption can be accessed via Bulk Data entry 'MDLPRM,TWBRBML,1'.
11. For optimization, individual DIMx of PBEAML can be selected as designed properties even with finite element formulation.

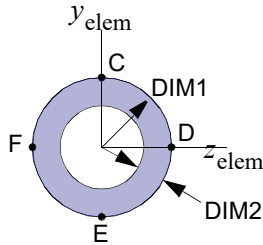
12. The origin of the  $y_{elem}$  and  $z_{elem}$  coordinate system for the section types is at the shear center for all element types when using the FEF. For section types 'L', 'T', 'T1', 'T2' and 'HAT1', no shear center calculation is calculated with the original beam equations and the origins of these types are at the locations specified in the following figures. For these sections types, this can result in a difference in the stress recovery points and the neutral axis locations between the beam equations and the FEF. The FEF results are considered more exact. For the remaining types with the beam equation, the shear center is either computed or is obvious due to symmetry considerations.



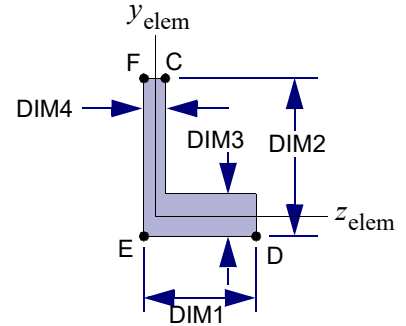
TYPE="ROD"



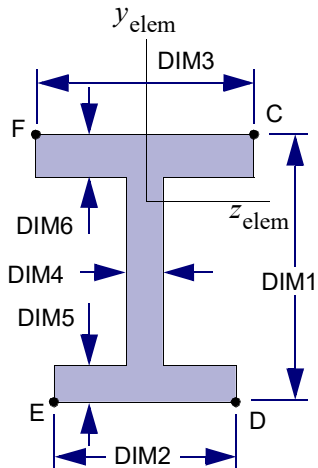
TYPE="TUBE"



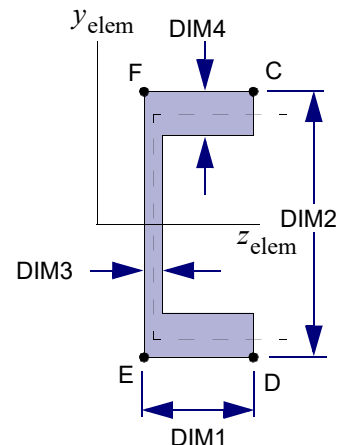
TYPE="TUBE2"



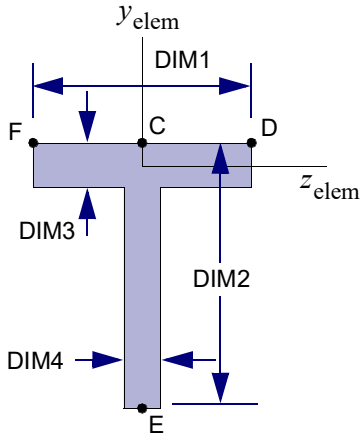
TYPE="L"



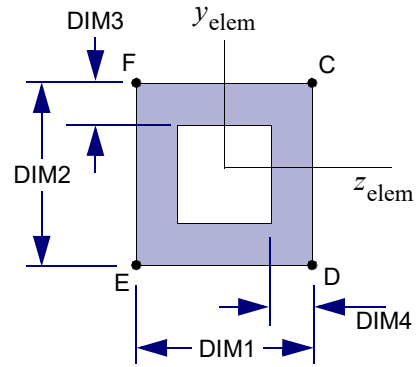
TYPE="I"



TYPE="CHAN"

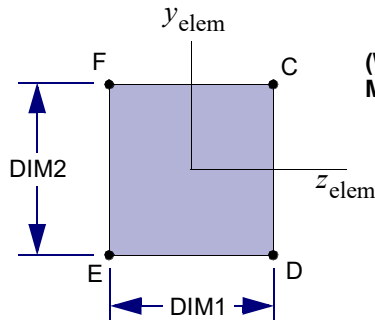


TYPE="T"

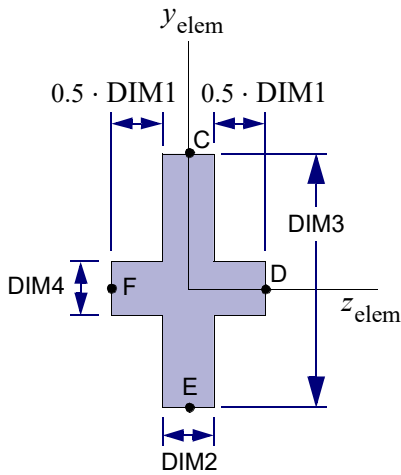


TYPE="BOX"

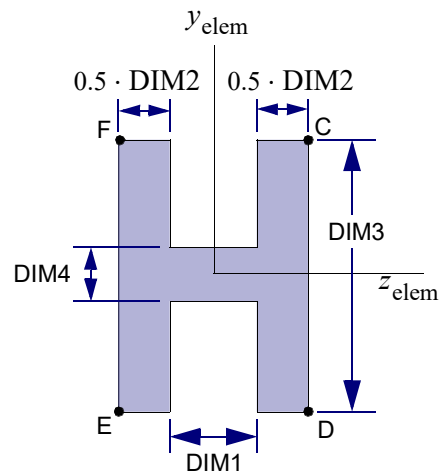
(Warping computed as zero if MDLPRM,TWBRBML,1)



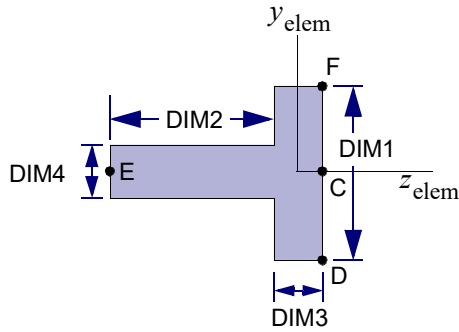
TYPE="BAR"



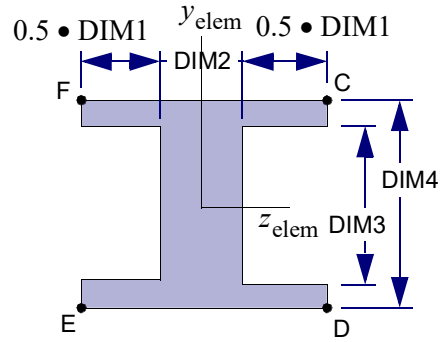
TYPE="CROSS"



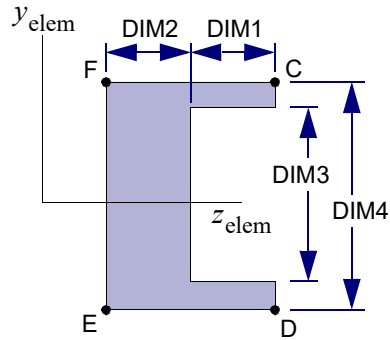
TYPE="H"



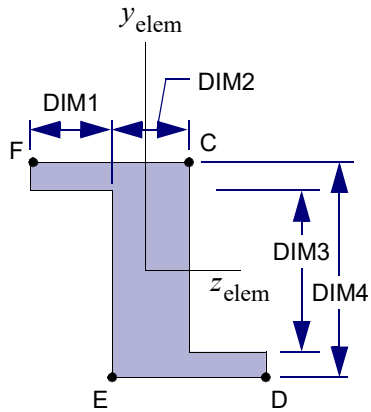
TYPE="T1"



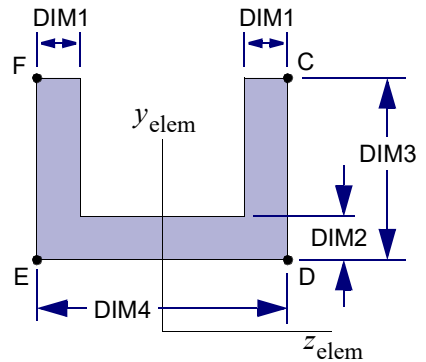
TYPE="I1"



TYPE="CHAN1"

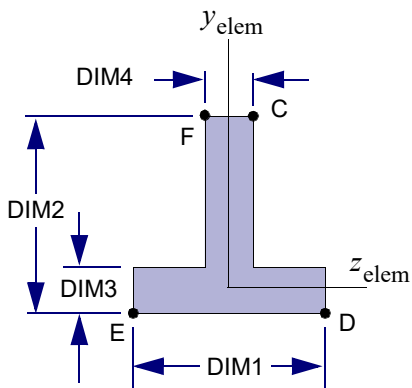


TYPE="Z"

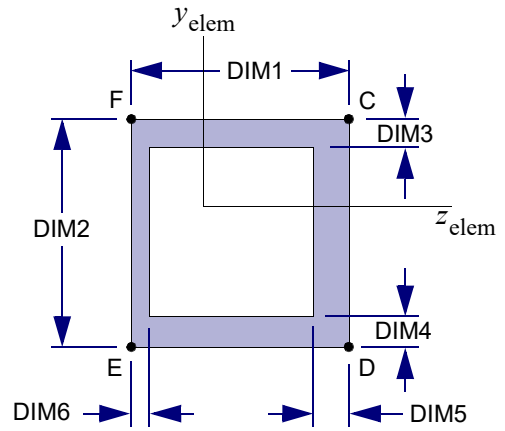


TYPE="CHAN2"

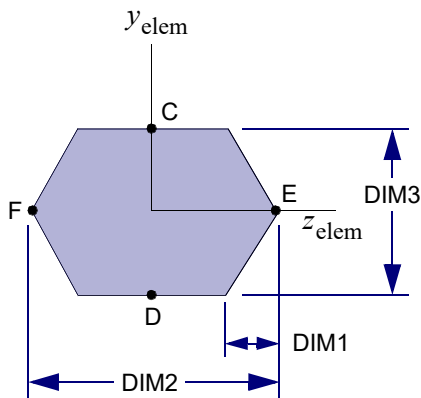




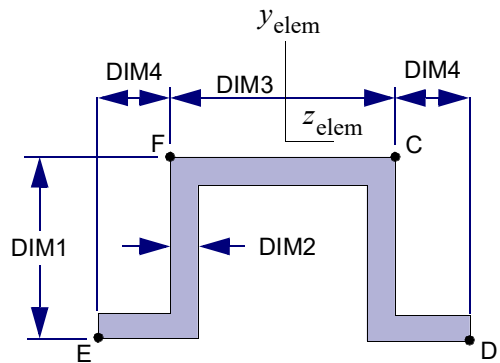
TYPE="T2"



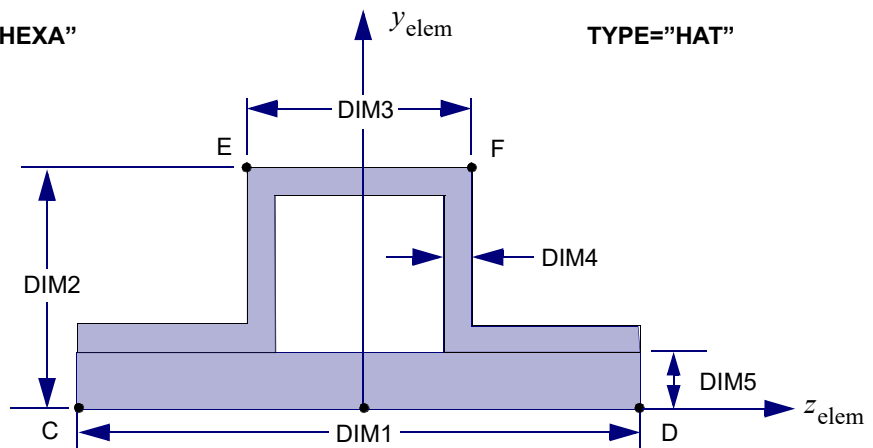
TYPE="BOX1"



TYPE="HEXA"



TYPE="HAT"



TYPE="HAT1"

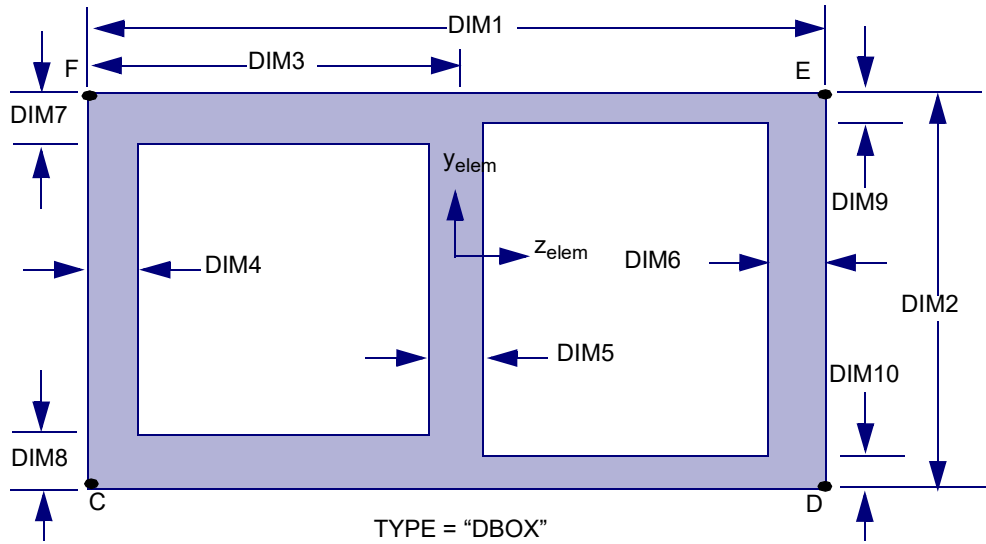


Figure 9-114 Definition of Cross-Section Geometry and Stress Recovery Points for GROUP = "MSCBML0"

## PBELT

## Belt Property- SOL 700

Defines the properties of a belt element referenced by a CROD entry. Used in SOL700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PBELT	PID	LOAD	UNLOAD	DENSITY	DAMP1	DAMP2	SLACK	PRESTRESS	

### Example:

PBELT	9	12	12	2.0E-5	0.1	0.1			

Field	Content
PID	Unique belt property number. (Integer > 0; required)
LOAD	TABLED1 ID defining the force as a function of strain during loading. The strain at time n is specified as engineering strain. (Integer > 0; required)  $\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$
UNLOAD	TABLED1 ID defining the force as a function of strain during unloading. The strain at time is specified as engineering strain. (Integer > 0; required)  $\text{strain}(n) = (\text{length}(n) - \text{length}(0)) / \text{length}(0)$
DENSITY	Density of the belt elements as mass per unit length. (Real > 0.0; required)

Field	Content
DAMP1	<p>A damping force is added to the internal force of the belt elements to damp out high frequency oscillations. (Real &gt; 0.0; default=0.1)</p> <p>The damping force is equal to:</p> $F_{\text{damp}} = \text{DAMP1.mass} \cdot (d\text{vel}/dt)$ <p>Where</p> <p><math>F_{\text{damp}}</math> = damping force</p> <p>DAMP1 = damping coefficient</p> <p>mass = mass of belt element</p> <p>dvel = velocity of elongation</p> <p>dt = time step</p>
DAMP2	<p>The maximum damping force: <math>\text{DAMP2} \cdot F_{\text{belt}}</math> (Real &gt; 0.0; default=0.1)</p> <p>Where</p> <p><math>F_{\text{belt}}</math> = Internal force in the belt element</p>
SLACK	<p>TABLED1 ID defining the slack as a function of time. (Integer &gt; 0; default=not used)</p> <p>The slack must be specified as engineering strain and will be subtracted from the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) - \text{SLACK}(n)$ <p>The force in the element is zero until the element strain exceeds the slack.</p>
PRESTRESS	<p>TABLED1 ID defining a prestress strain as a function of time. (Integer &gt; 0; default=not used)</p> <p>The prestress strain must be specified as engineering strain and will be added to the element strain at time as:</p> $\text{strain}(n) = \text{strain}(n) + \text{PRESTRESS}(n)$

**Remarks:**

1. The loading and unloading curves must start at (0.0, 0.0).

2. During loading, the loading curve is applied to determine the force in the belt element. At unloading, the unloading curve is shifted along the strain axis until it intersects the loading curve at the point from which unloading commences. The unloading table is applied for unloading and reloading, until the strain again exceeds the intersection point. Upon further loading, the loading table is applied. For subsequent unloading, the sequence is repeated.
3. Belt elements are tension only elements.
4. Instantaneous slack of an element can also be initialized per element using the TICEL entry with the keyword SLACK and a corresponding VALUE.

## PBEMN1

## Nonlinear Property Extensions for a PBEAM or PBEAML Entry

Specifies additional nonlinear properties for elements that point to a PBEAM or PBEAML entry in SOL 400.

### Format:

1	2	3	4	5	6	7	8	9	10
PBEMN1	PID	MID		SECT					
	"C2"	BEH2	INT2						

### Example:

PBEMN1	27	93							
--------	----	----	--	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number of an existing PBEAM entry. (Integer > 0)
MID	Material ID. Remark 9.(Integer ≥ 0)
SECT	Section integration. SECT = "S" a smeared cross section is used for integration. SECT = "N" a numerically integrated cross section is used. See Remark 10. (Character Default S or blank)
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 5. (Character Default BEAM)
INT2	Integration scheme. See Remarks 5. and 6. (Character Default LC)

### Remarks:

1. The PID above must point to an existing PBEAM or PBEAML Bulk Data entry and is honored only in SOL 400.
2. Tapering of the CBEAM is ignored. Only section properties at end A are used.
3. MID if blank (or 0) use the MID value on the PBEAM or PBEAML entry. If > 0 it will override the MID value on the PBEAM entry.
4. The MID entry may point to MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials
MAT1
MATVE
MATVP

Implicit Structural Materials
MATEP
MATF
MATS1

5. BEH2 refers to the nonlinear structural behavior of the BEAM element. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHAV CODE	Integration Code	Element Type	# Nodes
Beam	BEAM	<u>LC</u>	BEAM	2
		LCC	BEAM	2
		LCO	BEAM	2
		LS	BEAM	2

6. Integration codes in Remark 5. are:

INT CODE	Integration Type
LC	Linear/Cubic
LCC	Linear/Cubic Closed section
LCO	Linear/Cubic Open section
LS	Linear-shear

7. Integration code LCO requires appropriate scalar point SA and SB entries on the CBEAM entry or a fatal message will result.
8. Normal buckling modes for beams with nonlinear extensions can be computed through the ANALYSIS=BUCK step. The buckle modes can be evaluated through a linear perturbation step about a linear or nonlinear prestressed state. The current limitation is that lateral buckling mode computations (eg. Lateral-torsional buckling seen in compression flanges of open section beams) are not supported for these elements.
9. The structural element damping coefficient, GE, is not supported on elements which reference PBEMN1.

10. Smearing cross section support for integration codes are as follows:

Primary BEAM Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
PBEAM	LC	S	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Both Thin-Walled and Solid Sections with elastic materials. Euler-Bernoulli Beam	Smearing section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LS	S	Linear Interpolation for axial displacements, transverse displacements and rotations	Both Thin-Walled and Solid Sections with elastic materials. Captures transverse shear effects - useful for deep beams.	Smearing section properties only are supported. Only linear elastic materials via MAT1 card can be provided.
	LC or LS	N			Not Supported
	LCC or LCO	S			Not Supported
	LCC or LCO	N			Not Supported
PBEAML	LC or LS	S			Not Supported
	LC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Solid Sections. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported



Primary BEAM Property	INT CODE	SECT	Interpolation	Usage	COMMENTS
	LS	N	Linear Interpolation for axial displacements, transverse displacements and rotations	Solid Sections. Captures transverse shear effects - useful for deep beams.	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported
	LCC or LCO	S			Not Supported
	LCC	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Closed Sections without warping. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported
	LCO	N	Linear Interpolation for axial displacement and Cubic Interpolation for transverse displacements	Thin-Walled Open Sections with warping. Euler-Bernoulli Beam	Numerically integrated sections are supported. Both linear and nonlinear materials can be provided. This includes MAT1, MATS1, MATEP, MATVE, MATVP, MATSMA materials. MATHE/MATHP are not supported

## PBEND

## Curved Beam or Pipe Element Property

Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PBEND	PID	MID	A	I1	I2	J	RB	THETAB	
	C1	C2	DI	D2	E1	E2	F1	F2	
	K1	K2	NSM	RC	ZC	DELTA			

### Example:

PBEND	39	1	0.8	0.07	0.04	0.04	10.		
	0.5	0.4	-0.5	0.4					
	0.6	0.6				0.1			

### Alternate Format and Example for Elbows and Curved Pipes:

PBEND	PID	MID	FSI	RM	T	P	RB	THETAB	
			NSM	RC	ZC				
PBEND	39	1	1	0.5	0.02	10.	10.		
					0.1				

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 1. and 2. (Integer > 0)
A	Area of the beam cross section. (Real > 0.0)
I1, I2	Area moments of inertia in planes 1 and 2. (Real > 0.0)
J	Torsional stiffness. (Real > 0.0)
FSI	Flag selecting the flexibility and stress intensification factors. See Remark 3. (Integer = 1, 2, or 3)
RM	Mean cross-sectional radius of the curved pipe. (Real > 0.0)
T	Wall thickness of the curved pipe. (Real ≥ 0.0; RM + T/2 < RB)
P	Internal pressure. (Real)
RB	Bend radius of the line of centroids. (Real. Optional, see CBEND entry.)
THETAB	Arc angle of element. (Real, in degrees. Optional, see CBEND entry.)
Ci, Di, Ei, Fi	The i, j, z locations from the geometric centroid for stress data recovery. See Remark 8. (Real)

Describer	Meaning
K1, K2	Shear stiffness factor K in $K \cdot A \cdot G$ for plane 1 and plane 2. (Real)
NSM	Nonstructural mass per unit length. (Real)
RC	Radial offset of the geometric centroid from points GA and GB. See <a href="#">Figure 9-116</a> . (Real)
ZC	Offset of the geometric centroid in a direction perpendicular to the plane of points GA and GB and vector v. See <a href="#">Figure 9-116</a> . See <a href="#">Remark 9</a> . (Real)
DELTAN	Radial offset of the neutral axis from the geometric centroid, positive is toward the center of curvature. See <a href="#">Figure 9-116</a> . See <a href="#">Remark 9</a> . (Real; Default is described in <a href="#">Remark 5</a> .)

**Remarks:**

1. For structural problems, MID must reference a MAT1 material entry.
2. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
3. When the alternate PBEND entry with FSI is used the following options are available:  
Using the notation of Dodge and Moore from ORNL-TM-3658, 1972 we define:

The bend characteristic parameter  $\lambda$  as:

$$\lambda = \frac{R_B T}{R_M^2 \sqrt{(1 - \nu^2)}}$$

The internal loading parameter  $\Psi$  as

$$\Psi = \frac{P R_B^2}{E R_M T}$$

And the radius ratio parameter  $\gamma$  as

$$\gamma = \frac{R_B}{R_M}$$

For  $T \neq 0$

$$A = 2\pi R_M T$$

$$I_1 = I_2 = \frac{1}{2} A \left( R_M^2 + \frac{1}{4} T^2 \right)$$

For  $T = 0$

$$A = \pi R_M^2$$

$$I_1 = I_2 = \frac{1}{4} A R_M^2$$

For  $T \neq 0$

$$\Delta N = R_B \left\{ 1 - \frac{1}{2} \left[ \sqrt{\left(1 - \gamma^{-2} - \frac{1}{4} \left(\frac{T}{R_B}\right)^2 + \left(\frac{T}{R_B}\right) \gamma^{-1}\right)} + \sqrt{\left(1 - \gamma^{-2} - \frac{1}{4} \left(\frac{T}{R_B}\right)^2 - \left(\frac{T}{R_B}\right) \gamma^{-1}\right)} \right] \right\}$$

For  $T = 0$

$$\Delta N = \frac{1}{2} R_B \left[ 1 - \sqrt{(1 - \gamma^{-2})} \right]$$

#### Element flexibility calculations:

The shear stiffness factor represents a correction to the section area by computing an effective area  $A_s = KA$ . Here K is defined as:

$$K = \frac{1}{f_s}$$

$f_s$  is the form factor obtained here from the principal of complementary virtual work. There are many other ways to obtain this factor. By complementary virtual work it is defined as:

$$f_s = \frac{A}{I_1^2} \int \frac{Q^2}{b^2} dA$$

where Q is the first moment of area and b is the width of section cut.

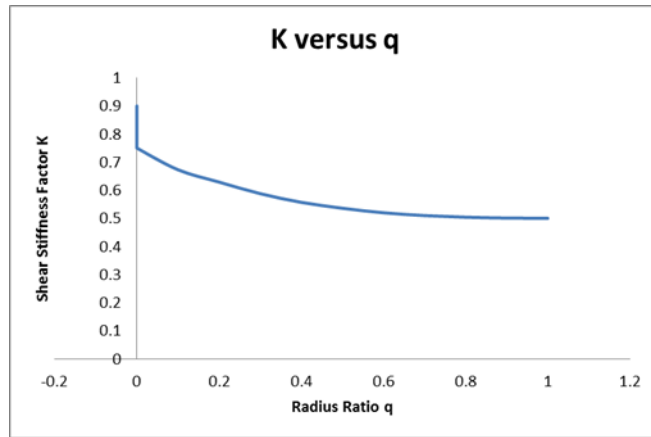
For a solid circular cross-section the above expressions yields a value of  $K = 0.9$ , and for very thin walled pipes the above integral evaluates as  $\pi T R_m^5$  and we get a value of  $K = 0.5$ .

For pipes with general thickness  $T > 0$  we define the radius ratio as:

$$q = \frac{R_i}{R_0} = \frac{1 - \frac{1}{2} \frac{T}{R_M}}{1 + \frac{1}{2} \frac{T}{R_M}}$$

where  $T = R_o - R_i$  and  $R_o$  is the outer radius of the pipe and  $R_i$  is the inner radius of the pipe. The above integral becomes a complicated function in terms of radicals and arc sine functions of  $q$  and its explicit integration form is not warranted in view of the approximate nature for  $f_s$ . We therefore approximate with the curve fit

$$K_1 = K_2 = \frac{0.75}{1 + \frac{q}{1 + q^2}}$$



The in and out of plane flexibility factors  $K_p$  used in computing the element flexibility are calculated as:

For FSI = 1

$$K_p = 1.0$$

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

$$K_p = \frac{1.65R_M^2}{R_B T} \left[ \frac{1}{1 + 6 \frac{PR_M}{TE} \left( \frac{R_M}{T} \right)^{\frac{4}{3}} \frac{1}{\gamma^{\frac{1}{3}}}} \right]$$

where it is recommended that  $\lambda \geq 0.2$

For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$K_p = \frac{1.73}{\lambda} \left[ \frac{1}{1 + 1.75\lambda^{\frac{4}{3}} \exp\left(-1.15\psi^{\frac{1}{4}}\right)} \right]$$

where it is recommended that  $0.05 \leq \lambda \leq 1.0$  and  $0.0 \leq \psi \leq 0.1$ . (Note in the cited Dodge and Moore the  $K_p$  coefficient has a value of 1.66 which was updated in 1991 to the current value of 1.73)

Also we must have  $K_p \geq 1.0$  for all FSI. Therefore, for FSI=2 or FSI=3, when  $K_p$  is computed as  $K_p < 1.0$ , it is set to  $K_p=1$ .

**Stress recovery stress intensification factor calculations:**

Define  $S_1$  as the in plane stress intensification factor and  $S_2$  as the out of plane stress intensification factor.

For FSI = 1

$$S_1 = \frac{I_1}{AR_B} \left[ \frac{1}{r_i} + \frac{R_B - \Delta N}{\Delta N(R_B + r_i)} \right]$$

$$S_2 = 1.0$$

If  $\Delta N = 0.0$ ,  $T = 0.0$ ,  $R_M = 0.0$  then  $S_1 = 1.0$ .

$r_i$  is the C1, D1, E1, F1 of the (r,z) recovery locations.

For FSI = 2 (ASME code Section III, NB-3687.2, NB-3685.2, 1977)

Define:

$$X_1 = 5.0 + 6\lambda^2 + 24\psi$$

$$X_2 = 17.0 + 600\lambda^2 + 480\psi$$

$$X_3 = X_1X_2 - 6.25$$

$$X_4 = (1 - \nu^2)(X_3 - 4.5X_2)$$

and

$$S_{1t} = \sin \phi + [(1.5X_2 - 18.75)\sin 3\phi + 11.25 \sin 5\phi]/X_4$$

$$S_{1n} = (\nu\lambda(9X_2 \cos 2\phi + 225 \cos 4\phi))/X_4$$

$$S_1 = S_{1t} + S_{1n}$$

$$S_{2t} = \cos \phi + [(1.5X_2 - 18.75)\cos 3\phi + 11.25 \cos 5\phi]/X_4$$

$$S_{2n} = (\nu\lambda(9X_2 \sin 2\phi + 225 \sin 4\phi))/X_4$$

$$S_2 = S_{2t} + S_{2n}$$

For data recovery the circumferential angle  $\phi$  is evaluated at 0 deg, 90 deg, 180 deg, and 270 deg. For FSI = 3 (Empirical factors from Welding Research Council Bulletin 179, Dodge and Moore)

$$S_1 = S_2 = \frac{2\lambda^{\frac{2}{3}}(1.0 + 0.25\gamma^{-1})}{1.0 + \lambda^{\frac{4}{3}} \exp\left(-\psi^{\frac{1}{4}}\right)}$$

4. The transverse shear stiffness in planes 1 and 2 are  $K1 \cdot A \cdot G$  and  $K2 \cdot A \cdot G$ , respectively. The default values for K1 and K2 on the first format are zero, which means the transverse shear flexibilities ( $1/Ki \cdot A \cdot G$ ) are set equal to zero. Transverse shear stiffness for the alternate format are automatically calculated for the curved pipe.
5. The neutral axis radial offset from the geometric centroid is default to the

$$\Delta N = \frac{I1}{A \cdot RB}$$

It is recommended that the default be used whenever

$$\frac{(RB)^2 A}{I1} < 15$$

in which case the default value of  $\Delta N$  is within 5% of the exact expression for circular or rectangular cross sections. For the alternate format, the neutral axis offset is calculated from an analytical expression for a hollow or solid circular cross section.

The user may compute an exact value for  $N$  as follows:

$$\Delta N = \frac{RB}{1 + \frac{(RB)^2 A}{Z}}$$

where

$$Z = \int \frac{r^2 dA}{1 + \frac{r}{RB}}$$

The integration is carried out over the cross section of the element.

6. If T is zero, a solid circular cross section of radius RM is assumed and FSI must be 1.
7. If the first format is used, third-order moments are neglected for the consistent mass matrix. These moments are zero whenever the cross section of the beam is symmetric about both the r and z axes.
8. If the circular cross-sectional property entry format is used, the stress points are automatically located at the points indicated in [Figure 9-115](#).
9. Offset vectors are treated like rigid elements and are therefore subject to the same limitations.
  - Offset vectors are not affected by thermal loads.
  - The specification of offset vectors is not recommended in solution sequences that compute differential stiffness because the offset vector remains parallel to its original orientation. (Differential stiffness is computed in buckling analysis provided in SOLs 105 and 200; SOLs 101,103 and 107 through 112 with STATSUB; and also nonlinear analysis provided in SOLs 106, 129, 153, and 159 with PARAM,LGDISP,1.)

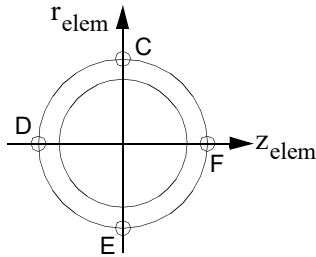


Figure 9-115 PBEND Circular Cross Section



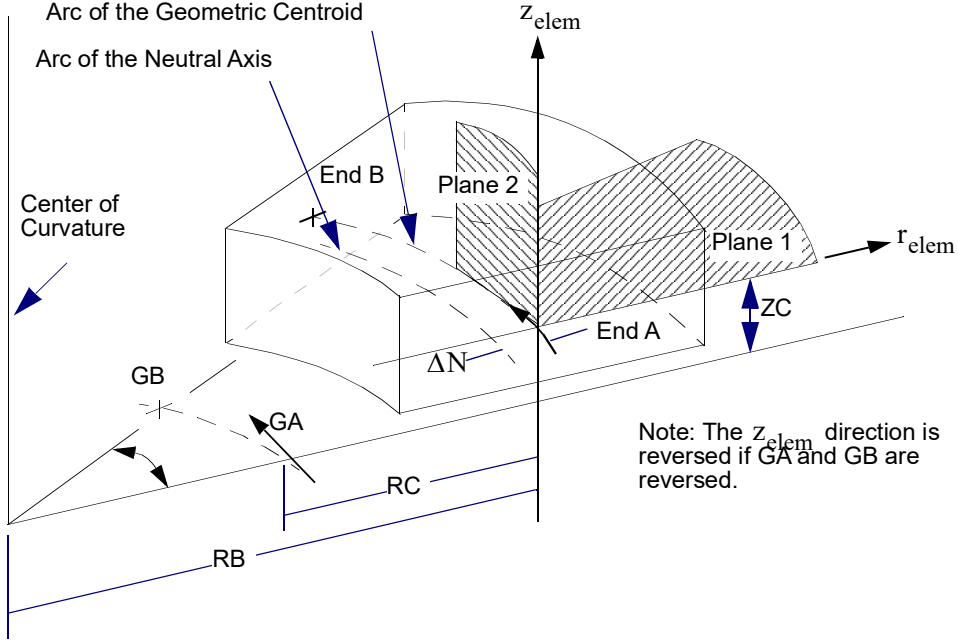


Figure 9-116 PBEND Element Coordinate System

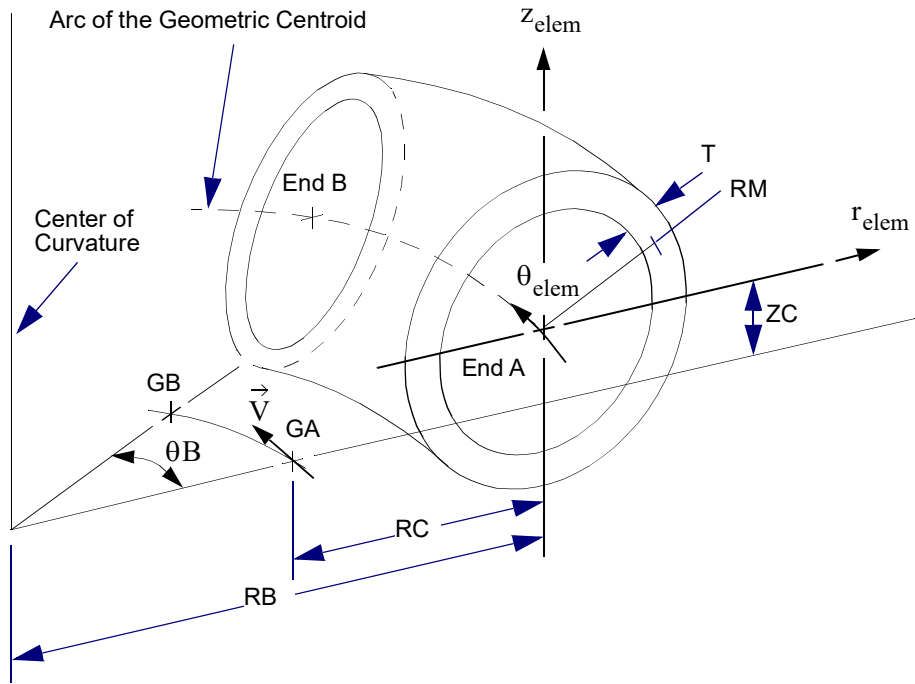


Figure 9-117 PBEND Circular Cross Section Element Coordinate System

10. For RC network solver in thermal analysis, the DELTAN is ignored.
11. PBEND is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBEND property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEND entries.
12. If GRAV or PLOAD1 loads are applied to a CBEND element, then it is recommended that the element should subtend an arc of 20 degrees or less degrees.

## PBMARB6

## Arbitrary Beam/Bar Cross Section in SOL 600

Defines arbitrary BEAM/BAR cross section for use in SOL 600.

### Format:

1	2	3	4	5	6	7	8	9	10
PBMARB6	PID	MID	Nseg	ORDx	ORDy	PreInt	POIS	IPRN	
	Px	Py	F1	F2	F3	F4	F5	F6	
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	1st seg
	X1	Y1	X2	Y2	X3	Y3	X4	Y4	2nd seg

(more rectangular segments, if required)

	X1	Y1	X2	Y2	X3	Y3	X4	Y4	last seg
--	----	----	----	----	----	----	----	----	----------

### Example: (Remark 5)

PBMARB6	20	4	2	3	3			3	
	2.0	1.0							
	0.0	1.0	0.8	0.0	0.8	1.0	0.0	1.0	
	0.0	1.0	1.0	1.0	1.0	2.0	0.0	2.0	

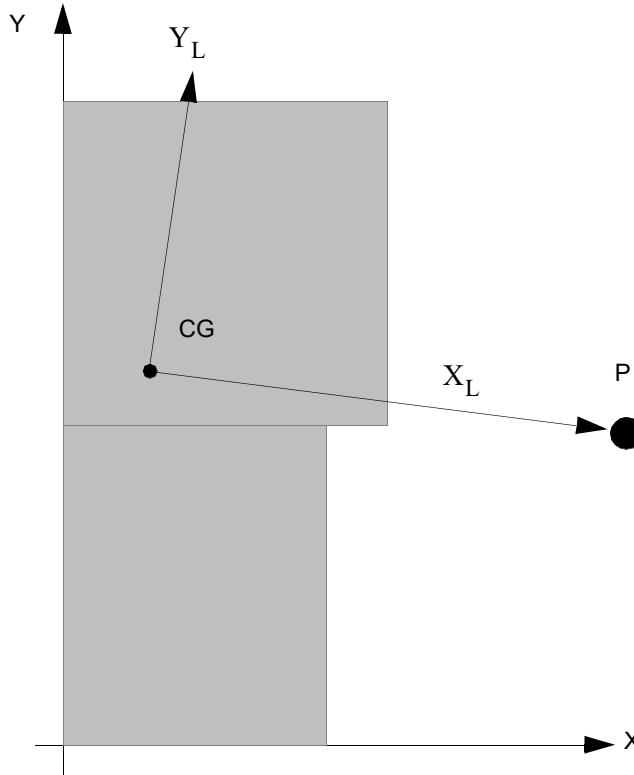
Describer	Meaning
PID	Property identification number. Must be unique among all property ID's. (Integer > 0; no Default)
MID	Material identification number. (Integer > 0; no Default)
Nseg	Number of quadrilateral shaped segments. See Remark 2. (Integer; no Default)
ORDx	Order of the integration rule used for each quadrilateral shaped segment in parametric -direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default)
ORDy	Order of the integration rule used for each quadrilateral shaped segment in parametric -direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is the same rule and order as in parametric -direction and the order cannot be larger than 10. (Integer; no Default)
PreInt	Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)
POIS	For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)

Describer	Meaning
IPRN	<p>Enter 1 to have the principal axis associated with the largest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 2 to have the principal axis associated with the smallest area moment of inertia to be aligned with the local x-axis.</p> <p>Enter 3 to have the x-axis of the coordinate system for which the section is being defined to be aligned with the local x-axis of the element (element coordinates). (Integer; Default = 3)</p>
Px	<p>Enter the x-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the x-coordinate of a point on the positive local x-axis. This coordinate defaults to <math>X_{cg}+1</math>, where <math>X_{cg}</math> is the x-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity.</p>
Py	<p>Enter the y-coordinate of a point that, when projected, lies on the positive side of the local x-axis. If the principal moments of inertia are equal, this defines the y-coordinate of a point on the positive local x-axis. This coordinate defaults to <math>Y_{cg}</math>, where <math>Y_{cg}</math> is the y-coordinate of the center of gravity of the section in the coordinate system in which the section was entered. The default is used when this field is blank or zero or when the user point coincides with the center of gravity.</p> <p>F1 Normal stiffness factor F1. (Real; Default = 1.0)</p> <p>F2 Bending stiffness factor for bending about local X axis. (Real; Default = 1.0)</p> <p>F3 Bending stiffness factor for bending about local Y Axis. (Real; Default = 1.0)</p> <p>F4 Shear stiffness factor for shear in local X direction. (Real; Default = 1.0)</p> <p>F5 Shear stiffness factor for shear in local Y direction. (Real; Default = 1.0)</p> <p>F6 Torsional stiffness factor. (Real; Default = 1.0)</p> <p><math>X_i, Y_i</math> Coordinates of the 4 corners of each rectangle. Enter Nseg sets of 4 coordinates each. See Remark 3. (Real; no Default)</p>

**Remarks:**

1. This option corresponds to Marc’s BEAM SECT Method D.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. The corners are given in counterclockwise order with respect to the local x-y axis.
4. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.

5. For the example, a section is made up of two rectangles as shown in the figure below. The input for the section defines one section with two quadrilateral segments using a 3x3 Simpson integration. The lower segment has size 0.8x1 and the upper segment has size 1x1. The first principal axis defines the local x-axis and the vector from the center of gravity (CG) to the projection of point P (2.0,1.0) onto this principal axis defines its positive direction. The coordinates of corner points of the segments and the coordinates of the point P have been entered with respect to the x,y-system.



6. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.
7. If this entry is used, the following parameters must be included in the bulk data:

PARAM,MARCBEAM,0

PARAM,MROUTLAY,N (where N is the number of integration layers, suggested values are N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N (N should be the same as specified for MROUTLAY)

## **PBMNUM6** Four Specific Numerically Integrated BEAM/BAR Cross Section in SOL 600

Defines four specific numerically integrated BEAM/BAR cross section for use in SOL 600.

### **Format:**

1	2	3	4	5	6	7	8	9	10
PBMNUM6	PID	MID	ITYPE	ORDx	ORDy	PreInt	POIS		
	A	B	C						
	F1	F2	F3	F4	F5	F6			

### **Example:**

PBMNUM6	30	6	2	3	3				
	7.0	2.5							

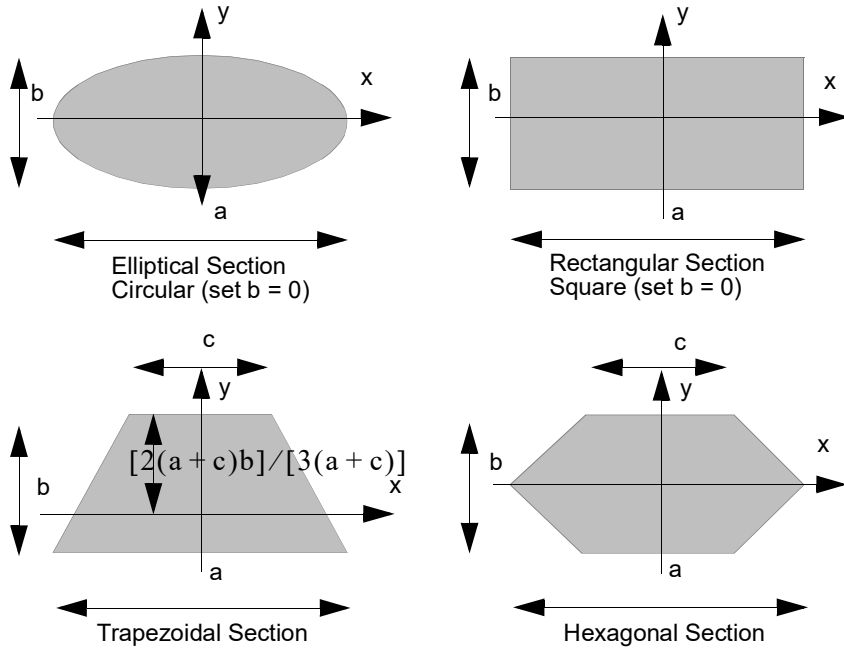
Describer	Meaning
PID	Property identification number. (Integer > 0; no Default) Must be unique among all property ID's
MID	Material identification number. (Integer > 0; no Default)
ITYPE	Type of section, see figure below. (Integer; no Default)
	1            Elliptical
	2            Rectangular
	3            Trapezoidal
	4            Hexagonal
ORDx	For an elliptical section, enter the number of subdivisions in radial direction. The default is 3.  For a rectangular or trapezoidal section, enter the order of the integration rule used in local x-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10.  For a hexagonal section, enter the order of the integration rule used in local x-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is 5 and the order cannot be larger than 10. (Integer; no Default)

Describer	Meaning
ORDy	<p>For an elliptical section, enter the number of subdivisions in circumferential direction of a 90° sector. The default is 2. For a rectangular or trapezoidal section, enter the order of the integration rule used in local y-direction. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is what was entered in the first field and the order cannot be larger than 10.</p> <p>For a hexagonal section, enter the order of the integration rule used in local y-direction over each trapezoidal half. If the number is positive and odd, a Simpson rule is used. If the number is positive and even, a Newton-Cotes rule is used. If the number is negative, a Gauss rule is used. The default is half the order in x-direction plus 1 and the order cannot be larger than 10. For non-Gauss rules, the points in the top row of the lower trapezoid coinciding with the points in the bottom row of the upper trapezoid (i.e., the points coinciding at <math>y = 0</math>) are merged together.</p>
PreInt	Enter 1 to use a pre-integrated section or 0 to use numerical integration through the analysis. If PreInt=0, the section remains elastic. (Integer; Default = 0)
POIS	For a uniform change in cross section, enter the effective Poisson's ratio. This data is not used at this time and its value will be ignored. (Real; Default = 0.0)
A	<p>First dimension of the cross section (Real; no Default)</p> <p>Ellipse      A is the diameter of the circle or the length of the ellipse in local x.</p> <p>Rectangle    A is the length of the square or the rectangle in local x.</p> <p>Trapezoid    A is the width of the trapezoid in local x on minus local y side.</p> <p>Hexagon      A is the width of the hexagon in local x at <math>y = 0</math>.</p>
B	<p>Second dimension of the cross section (Real; Default B=A)</p> <p>Ellipse      B is the height of the ellipse in local y.</p> <p>Rectangle    B is the height of the rectangle in local y.</p> <p>Trapezoid    B is the height of the trapezoid in local y.</p> <p>Hexagon      B is the height of the hexagon in local y.</p>
C	<p>Third dimension of the cross section if required (Real; Default = 0.0)</p> <p>Ellipse      C is not used. Leave blank or enter 0.</p> <p>Rectangle    C is not used. Leave blank or enter 0.</p> <p>Trapezoid    C is the width of the trapezoid in local x on the plus local y side.</p> <p>Hexagon      C is the width of the hexagon in local x on either local y side.</p>
F1	Normal stiffness factor F1 (Real; Default = 1.0)
F2	Bending stiffness factor for bending about local X axis (Real; Default = 1.0)
F3	Bending stiffness factor for bending about local Y Axis (Real; Default = 1.0)
F4	Shear stiffness factor for shear in local X direction (Real; Default = 1.0)

Describer	Meaning
F5	Shear stiffness factor for shear in local Y direction (Real; Default = 1.0)
F6	Torsional stiffness factor. (Real; Default = 1.0)

**Remarks:**

1. This option corresponds to Marc's BEAM SECT Method C.
2. No more than 100 integration points can exist in any cross section. Pre-integrated sections do not allow stress and strain output in section integration points; only generalized stresses and strains can be requested for output. Non pre-integrated sections cannot have more than 100 segments using single point integration each. For pre-integrated sections, there is no limit on the number of segments
3. This option applies only to Marc element types 98 (SOL 600 default) and 52. The difference between types 98 and 52 is that 98 has transverse shear and 52 does not have transverse shear.
4. If this entry is used, do not enter any other properties (such as pbeam, pbeam1, pbar, pbar1, etc.) for the elements using this property.
5. The second continuation entry may be eliminated if F1 through F6 take the default values.
6. For the rectangle, if B=A or B=0.0 the cross section is a square.
7. For the Ellipse, if B=A or B=0.0, the cross section is a circle.



8. If this entry is used, the following parameters must be included in the bulk data:



PARAM,MARCBEAM,0

PARAM,MROUTLAY,N

(where N is the number of integration layers, suggested values are N=5 if there is no plasticity and N=11 with plasticity)

PARAM,MARCSLHT,N

(N should be the same as specified for MROUTLAY)

## PBMSECT

### Arbitrary Cross-Section for CBEAM

Defines the shape of arbitrary cross-section for CBEAM element.

**Format:**

1	2	3	4	5	6	7	8	9	10
PBMSECT	PID	MID	FORM						
Data description for arbitrary section									

**Example:**

PBMSECT	1	10	GS						
OUTP=10,INP=20									
PBMSECT	1	10	CP						
OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)], NSM=0.01									

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.

**Remarks:**

- Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

- OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTER Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.
- OUTM = value(Integer > 0), points to the ID of BEGIN BULK ARBMODEL. OUTM is designed specifically for arbitrary beam cross section with finite element discretization already available. Note that OUTM must not appear together with other keyword, such as OUTP or INP, on a PBMSECT.



For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INner Perimeter by traversing through all the POINTs in the SET.

For OP and CP:

BRP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch. The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

T(id) = [value(Real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the start and end points of line segment(s). The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=( ) do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

- Core(id) = [PCID,PT=(pid1,pid2)]; specifies the composite layout for CORE part of composite. PCID is the ID of a PCOMPi/PCOMPG Bulk Data entry. PT=(pid1,pid2) defines the start and end points of line segment(s) which utilizes PCID.
- Core=PCID is acceptable input and is used as default which is applicable to all segments that are not specifically defined with Core(id).
  - Thickness continuity of a ply must be maintained. Thickness change from segment to segment is not allowed.
  - For CP, the closing segment should always use the default Core=pcid.
  - No ply should go over the profile line which is defined by OUTP and BRP. If situation arises, split a ply into two in such a way the no ply go over the profile line.
- Layer(id) = [PCID,SETID]; specifies the composite layout for additional Layer(s) that wraps around Core. PCID is the ID of a PCOMP/PCOMPG Bulk Data entry. SETID selects a SET1/SET3 with POINT IDs.
- NSM = value(Real > 0.0), specifies non-structural mass per unit length.
- (id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBMSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBMSECT with GS form. The points with maximum and/or minimum coordinates in X and Y axes are more likely to be picked. For PBMSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.

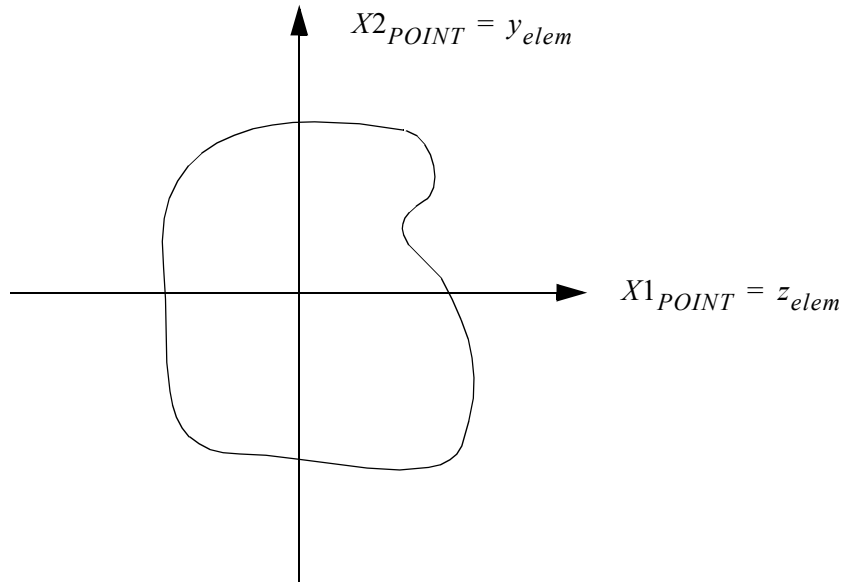


Figure 9-118 Arbitrary Cross-Section Definition

5. Current implementation of PBMSECT supports constant section beam only.
6. Note that keyword Core can be abbreviated as 'C'. Similarly, keyword Layer can be abbreviated as 'L'.
7. If Core and/or Layer appears in PBMSECT, the PBMSECT ID can not be referenced on CBEAM. Instead, it should be referenced on CBEAM3. Note, however, that a CBEAM3 cannot be used with any other of the above options except the Core and/or Layer option. The use of any other option results in a FATAL. In addition, it is recommended that CBEAM3 referencing composite PBMSECT has 3 nodes and 3 warping DOFs.'
8. If OUTM=arbid is utilized on PBMSECT, element connection, grid location, PSHELL and material Bulk Data entries must be provided after 'BEGIN ARBMODEL=arbid'.
9. Note that the 'arbid' used under 'BEGIN ARBMODEL' is considered global and can be referenced by PBMSECT with OUTM=arbid in different 'BEGIN SUPER' Bulk Data Section for Part Superelements (SE).
10. PBMSECT with Core or Layer must be utilized along with 'PARAM,ARB MSTYP,TIMOSHEN' in the Bulk Data Section.
11. The entry computes, based on an internally generated finite element analysis using a 2D mesh of the cross-section, the following:

$$A = \int dy dz$$

$$I1 = \int y^2 dy dz$$

$$I2 = \int z^2 dy dz$$

$$I12 = \int yz dy dz$$

where the above integrals are evaluated by numerical integration.

For a beam cross-section, the warping function,  $\phi$ , satisfies the equation

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0$$

with boundary

$$\left(\frac{\partial \phi}{\partial y} + z\right)n_y + \left(\frac{\partial \phi}{\partial z} - y\right)n_z = 0$$

where  $n_y$  and  $n_z$  are the direction cosine of the normal to the boundary.

Then, the torsion constant is defined as

$$J = I1 + I2 - \int \left[ \frac{\partial \phi}{\partial z} \frac{\partial \phi}{\partial y} \right] \begin{Bmatrix} -y \\ z \end{Bmatrix} dA$$

The load equilibrium of the beam cross-section can be resolved into two Poisson equations for the shear forces in the  $y$  and  $z$  direction as:

$$\nabla^2 f_y = -y/I1$$

$$\nabla^2 f_z = -z/I2$$

then, the shear stiffness factor is defined as

$$K1 = \left[ \frac{A}{I2} \int z f_z dA \right]^{-1}$$

$$K2 = \left[ \frac{A}{I1} \int y f_y dA \right]^{-1}$$

The warping constant is defined as

$$C_w = \int \phi^2 dA - \begin{bmatrix} y_{sc} & z_{sc} \end{bmatrix} [I] \begin{Bmatrix} y_{sc} \\ z_{sc} \end{Bmatrix}$$

The shear center is defined as

$$Q1 = \int z dA \quad Q2 = \int y dA$$

$$N1A = Q1/A \quad N2A = Q2/A$$

12. PBMSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBMSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBEAM, PBEAML, PBCOMP, PBMSECT, PBEAM3 entries.

## PBRSECT

## Arbitrary Cross-Section for CBAR

Defines the shape of arbitrary cross-section for CBAR element.

### Format:

1	2	3	4	5	6	7	8	9	10
PBRSECT	PID	MID	FORM	NSM					
Data description for arbitrary section									

### Example:

PBRSECT	1	10	GS						
OUTP=10,INP=20									
PBRSECT	1	10	CP						
OUTP=10,BRP=20,T=1.0,T(11)=[1.2,PT=(123,204)]									

Descriptor	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
FORM	Cross-section form. (Character) See Remark 1.
NSM	Non-structural mass per unit length. (Real $\geq$ 0.0; Default = 0.0)

### Remarks:

- Options for FORM are

GS	General Section
OP	Open Profile
CP	Closed Profile

- Keywords for describing the arbitrary cross-section:

For GS, OP and CP:

OUTP = value(Integer > 0); points to ID of a SET1 or SET3 that defines the OUTER Perimeter for FORM=GS or the center line for FORM=CP (or OP) by traversing through all the POINTs in the SET.

For GS only:

INP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that defines a INNER Perimeter by traversing through all the POINTs in the SET.



For OP and CP:

BRP(id) = value(Integer > 0); points to the ID of a SET1 or SET3 that specifies a BRanch.  
 The rules for BRP are:

- BRP must start from and/or end on OUTP.
- Segment length, defined as distance between two neighboring points on BRP or OUTP, is preferably to be longer than the segment thickness. Closely spaced points creates denser mesh for the cross section and does not increase accuracy of properties significantly.

T(id) = [value(real > 0.0),PT=(pid1,pid2)]; specifies the thickness of a segment in profile. PT=(pid1,pid2) defines the end points of a straight line segment. The rules for T(id) are:

- For CP and OP, it is a requirement to have a T(id) without PT=(pid1,pid2) to serve as a default thickness for all segments which do not have its thickness specified. This requirement is valid even when the thickness for every segment is specified.
- pid1 and pid2 under PT=( ) do not need to be neighboring points of a SET. However, pid1 and pid2 must hold its position as in SETx for OUTP and/or BRP.
- Same segment may be covered by different T(id). T(id) are processed sequentially as specified. The thickness of the last T(id) for a segment is used as the thickness of a segment. Use 'PARAM,ARBMNOW,1' to turn off segment property overwrite.
- For CP, T(id) of segment that close the profile must have (pid1=ID of last point in OUTP, pid2=ID of first point in OUTP). Note that, if input for pid1 and pid2 are reversed, T(id) will cover all segments in OUTP except the one that close the profile.

NSM = value(Real > 0.0), specifies non-structural mass per unit length.

(id) = integer (>0) identifies INP, BRP or T which is not required if a single entity appears in the PBRSECT entry. For T, the T(id) can be used to identify the particular thickness to be designed in SOL 200.

3. Stress data recovery points are selected automatically from all points of a PBRSECT with GS form. The points with maximum and/or minimum coordinates in X1 and/or X2 axes are more likely to be picked. For PBRSECT with CP or OP form, the stress data recovery points are selected from points with computed coordinates that actually encircle the profile. Similar to GS form, the points with extreme coordinates are more likely to be selected.
4. Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP. In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.
5. See Remark 11. of the PBMSECT entry for the theory used to compute the cross-sectional properties.

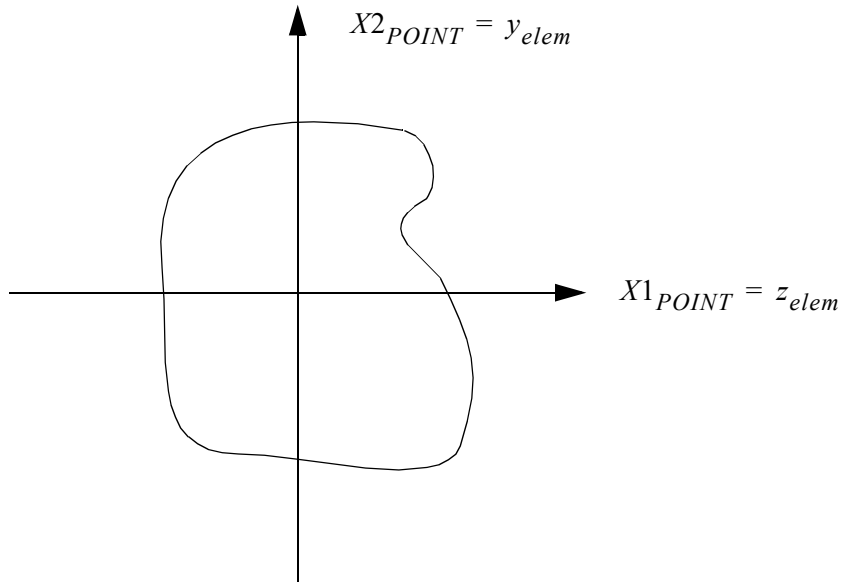


Figure 9-119 Arbitrary Cross-Section Definition

6. PBRSECT is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBRSECT property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBAR, PBARL, PBRSECT entries.

**PBSH2DT****2-D Linear/Nonlinear Connection Properties using Table Lookup**

Defines frequency-dependent properties of a two-dimensional element (CBUSH2D) using TABLED1 lookup.

**Format:**

1	2	3	4	5	6	7	8	9	10
PBSH2DT	PID	“K”	K11	K22	K12	K21			
		“B”	B11	B22	B12	B21			
		“M”	M11	M22	M12	M21			

Describer	Meaning
PID	Property id of a PBUSH2D entry (see Remark 1.)
K11	TABLED1 used to define frequency-dependent stiffness in the T1 direction (Integer >0 or blank)
K22	TABLED1 used to define frequency-dependent stiffness in the T2 direction (Integer >0 or blank)
K12	TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)
K21	TABLED1 used to define frequency-dependent cross coupling stiffness (Integer >0 or blank)
B11	TABLED1 used to define frequency-dependent damping in the T1 direction (Integer >0 or blank)
B22	TABLED1 used to define frequency-dependent damping in the T2 direction (Integer >0 or blank)
B12	TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)
B21	TABLED1 used to define frequency-dependent cross coupling damping (Integer >0 or blank)
M11	TABLED1 used to define frequency-dependent mass in the T1 direction (Integer >0 or blank)
M22	TABLED1 used to define frequency-dependent mass in the T2 direction (Integer >0 or blank)
M12	TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)
M21	TABLED1 used to define frequency-dependent cross coupling mass (Integer >0 or blank)

**Remarks:**

1. A PBUSH2D with the same PID must exist. The values from the PBUSH2D will be used as the nominal stiffness, damping, and mass. Linear statics, normal modes, reduction, and superelement processing will use the nominal values. The values from the selected TABLED1 entries will be used in any frequency-dependent loop.
2. Any field left blank indicates that the associated stiffness, damping, or mass is not frequency-dependent and the nominal values will be used for that term in the solution.
3. PBUSH2DT card will only work if there is rotor definition in the model. If PBUSH2DT is used on non-rotordynamics model, a FATAL message will be issued.

## PBUSH Generalized Spring-and-Damper Property

Defines the nominal property values for a generalized spring-and-damper structural element.

### Format:

1	2	3	4	5	6	7	8	9	10
PBUSH	PID	"K"	K1	K2	K3	K4	K5	K6	
		"B"	B1	B2	B3	B4	B5	B6	
		"GE"	GE1	GE2	GE3	GE4	GE5	GE6	
		"RCV"	SA	ST	EA	ET			
		"M"	M						
		"T"	ALPHA	TREF	COINL				

### Example 1:

Stiffness and structural damping are specified.

PBUSH	35	K	4.35	2.4				3.1	
		GE	.06						
		RCV	7.3	3.3					

### Example 2:

Damping force per unit velocity are specified.

PBUSH	35	B	2.3						
-------	----	---	-----	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
"K"	Flag indicating that the next 1 to 6 fields are stiffness values in the element coordinate system. (Character)
Ki	Nominal stiffness values in directions 1 through 6. See Remarks 2. and 3. (Real; Default = 0.0)
"B"	Flag indicating that the next 1 to 6 fields are force-per-velocity damping. (Character)
Bi	Nominal damping coefficients in direction 1 through 6 in units of force per unit velocity. See Remarks 2., 3., and 9. (Real; Default = 0.0)
"GE"	Flag indicating that the next fields, 1 through 6 are structural damping constants. See Remark 7. (Character)
GEi	Nominal structural damping constant in directions 1 through 6. See Remarks 2. and 3. and 9. (Real; GE1 default = 0.0, GE2-6, defaults described in Remark 9.)

Describer	Meaning
“RCV”	Flag indicating that the next 1 to 4 fields are stress or strain coefficients. See Remarks 4. and 5.(Character).
SA	Stress recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ST	Stress recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
EA	Strain recovery coefficient in the translational component numbers 1 through 3. (Real; Default = 1.0)
ET	Strain recovery coefficient in the rotational component numbers 4 through 6. (Real; Default = 1.0)
“M”	Flag indicating that the following entries are mass properties for the CBUSH element. If inertia properties ( Iij )are desired CONM2 should be used.
M	Lumped mass of the CBUSH. (Real≥0.0; Default=0.0)
“T”	Flag indicating that the following entries are thermal properties for the CBUSH element. See Remark 14.. (Character)
ALPHA	Thermal expansion coefficient for the CBUSH. (Real; Default=0.0)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 14.)
COINL	Length of a CBUSH with coincident grids. (Real; Default=0.0, COINL ≥ 0.0)

**Remarks:**

1. Ki, Bi, or GEi may be made frequency dependent for both direct and modal frequency response by use of the PBUSHT entry.
2. The nominal values are used for all analysis types except frequency response and nonlinear analysis. For modal frequency response, the normal modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
3. If PARAM,W4 is not specified, GEi is ignored in transient analysis.
4. The element stresses are computed by multiplying the stress coefficients with the recovered element forces.  $\sigma_i = F_i \cdot SA$  or  $\sigma_i = M_i \cdot ST$
5. The element strains are computed by multiplying the strain coefficients with the recovered element displacements.  $\varepsilon_i = U_i \cdot EA$  or  $\varepsilon_i = \theta_i \cdot ET$
6. The “K”, “B”, “GE”, “RCV”, or “M” entries may be specified in any order.
7. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
8. Applicable fields refer to directions in the element’s coordinate system.

9. For upward compatibility, if ONLY GE1 is specified on a PBUSH entry and GE<sub>i</sub>, i = 2 – 6 are all blank on the PBUSH entry, then the single structural damping GE1 is applied to all defined K<sub>i</sub> for the PBUSH entry. In this case, GE<sub>i</sub>, i = 2 - 6, blank is not equivalent to 0.0 but a special flag.

For any PBUSH entry that has any GE<sub>i</sub>, i = 2 – 6 numerically specified including a 0.0, the fields GE<sub>i</sub> are all treated separately and any GE<sub>i</sub>, i = 2 - 6 field with no entry specified is defaulted to 0.0. In this case, for any GE<sub>i</sub>, i = 2 – 6, blank is equivalent to 0.0 and the results are radically different from the no GE<sub>i</sub>, i = 2- 6 specified as described above.

Thus for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05						

The GE=.05 will be replicated to fields 5 through 9.

While for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	0.0					

The element x-direction will have a GE=.05 while the other 5 directions will have an associated GE=0.0

The entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05		.02				

is equivalent to the full entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	.0	.02	.0	.0	.0	

IFPSTAR introduced an incompatibility with the above rules in Nastran versions 2014 to 2017. For Nastran version 2018 IFPSTAR, the above original rules have been restored. However, the previous incompatible IFPSTAR results can be recovered with bulk data entry MDLPRM, GEV1417, 1 or NASTRAN, SYSTEM(754)=1 in which case the last example becomes equivalent to the full entry:

1	2	3	4	5	6	7	8	9	10
PBUSH	3303000	K	653.	4000.	460.	10.+3	10.+3	10.+3	
		GE	.05	.05	.02	.05	.05	.05	

10. For SOL 600, it is not necessary to enter PBUSH if a PBUSHT with the same ID is used in the model. Omitting the PBUSH entry if a PBUSHT with the same ID is entered.

11. For SOL 600, the defaults for SA, ST, EA and ET are 1.0E-10.

The mass properties are assumed located at the point defined by S (or S1, S2, S3 if OCID> -1) on the CBUSH entry. They are assumed to be in the element coordinate system. Define  $\alpha = S$  for the case OCID= -1 (default). See CBUSH entry. For the case OCID> -1, define  $r_A, r_B$  to be the vector from grid GA to (S1,S2,S3) and from grid GB to (S1,S2,S3) respectively, then

$$\alpha = \frac{|r_A|}{|r_A| + |r_B|}$$

$$|r_A| = \sqrt{(GA1 - S1)^2 + (GA2 - S2)^2 + (GA3 - S3)^2}$$

$$|r_B| = \sqrt{(GB1 - S1)^2 + (GB2 - S2)^2 + (GB3 - S3)^2}$$

Then the diagonal contribution to GA for the mass matrix in the element coordinate system is:

$$M11A= M22A=M33A= (1- \alpha)*M$$

The diagonal contribution to GB for the mass matrix in the element coordinate system is:

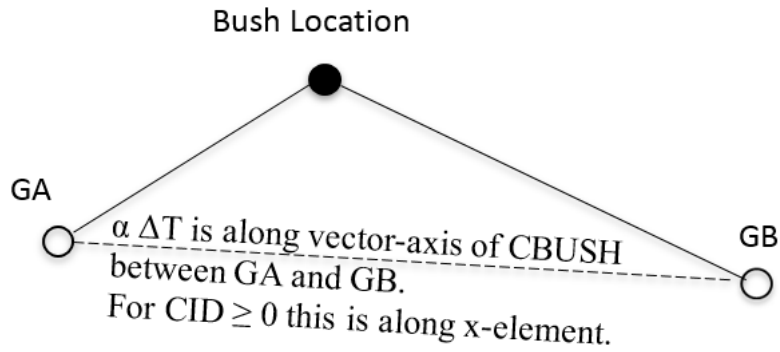
$$M11B= M22B=M33B=( \alpha)*M$$

12. PBUSH is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH entries.
13. The CBUSH is a self-equilibrating element which means that for a CBUSH of finite length, its translational stiffness terms couple with rotational stiffness for rotational degrees of freedom. However, this coupling is dependent on the orientation of the CBUSH element. Thus there may be times when no explicit rotational stiffness is specified along a direction for which translational stiffness was specified that there is also no coupled rotational stiffness for that direction.

In modal analysis, a CBUSH of finite length having rotational degrees of freedom with no rotational stiffness (coupled or explicit) and no associated mass on the grids may cause massless mechanisms. A massless mechanism means the natural frequency for this rotational mode approaches the limit of zero rotational stiffness divided by zero mass, which is an undefined quantity. If the elements lie along a global coordinate axis, the mass term is identically zero, which leads to very large negative or positive eigenvalues and is usually beyond any reasonable search region. If the elements are skewed from the global axes, the eigenvalues may be computed at any value (including negative) because of the indeterminacy caused by numerical truncation.



14. The thermal expansion for the CBUSH will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERATURE Case Control command for thermal loading rules. For a CBUSH with coincident grids, or a single grid, if COINL is provided, the thermal expansion is always just along the element x-axis.



## PBUSH1D

## Rod Type Spring-and-Damper Property

Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PBUSH1D	PID	K	C	M		SA	SE		
	"SHOCKA"	TYPE	CVT	CVC	EXPVT	EXPVC	IDTS		
			IDETS	IDECS	IDETSD	IDECSD			
	"SPRING"	TYPE	IDT	IDC	IDTDU	IDCDU			
	"DAMPER"	TYPE	IDT	IDC	IDTDV	IDCDV			
	"GENER"	TYPE	IDT	IDC	IDTDU	IDCDU	IDTDV	IDCDV	

### Example:

PBUSH1D	35	3000.	200.	300.					
	SHOCKA	TABLE	2.2	1.2	1.		200		

The continuation entries are optional. The four options, SHOCKA, SPRING, DAMPER, and GENER can appear in any order.

Describer	Meaning	Default
PID	Property identification number. (Integer > 0).	Required
K	Stiffness. (Real $\geq 0$ ).	See Remark 1.
C	Viscous damping. (Real $\geq 0$ ).	See Remarks 1. and 2.
M	Total mass of the element. (Real $\geq 0$ ).	Blank
SA	Stress recovery coefficient [1/area]. (Real $\geq 0$ ).	Blank
SE	Strain recovery coefficient [1/length]. (Real $\geq 0$ ).	Blank

Describer	Meaning	Default
SHOCKA	<p>Character string specifying that the next 10 fields are coefficients of the following force versus velocity/displacement relationship. (Character).</p> $F(u, v) = C_v \cdot S(u) \cdot \text{sign}(v) \cdot  v ^{\text{EXPV}}$ <p>The force F, the displacement u, and the velocity v, are in the axial direction of the damper. The axis of the damper is defined by the two connecting grid points GA and GB on the CBUSH1D Bulk Data entry. The displacement u and the velocity v, are the relative displacement and the relative velocity with respect to the grid point GA. The scale factor S(u) must be defined with a table or with an equation.</p>	
TYPE	<p>Character string indicating the type of definition. (Character). For TYPE = EQUAT, the fields IDETS, IDECS, IDETSD, and IDECSD are identification numbers of DEQATN entries. For TYPE = TABLE the field IDTS is an identification number of a TABLEDi entry. If no character string is provided (blanks), TYPE = TABLE is set.</p>	TABLE
CVT	<p>Viscous damping coefficient <math>C_V</math> for tension <math>v &gt; 0</math>, force per unit velocity. (Real).</p>	Required for SHOCKA
CVC	<p>Viscous damping coefficient <math>C_V</math> for compression <math>v &lt; 0</math>, force per unit velocity. (Real).</p>	CVT
EXPVT	<p>Exponent of velocity EXPV for tension <math>v &gt; 0</math>. (Real).</p>	1.
EXPVC	<p>Exponent of velocity EXPV for compression <math>v &lt; 0</math>. (Real).</p>	EXPVT
IDTS	<p>Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE. The TABLEDi entry defines the scale factor S, versus displacement u.</p>	Required for SHOCKA and TYPE=TABLE
IDETS	<p>Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for tension <math>u &gt; 0</math>.</p>	Required for SHOCKA and TYPE=EQUAT
IDECS	<p>Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the scale factor S, versus displacement u, for compression <math>u &lt; 0</math>.</p>	IDETS
IDETSD	<p>Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for tension <math>u &gt; 0</math>.</p>	Required for SHOCKA and TYPE=EQUAT

Describer	Meaning	Default
IDECS	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the scale factor S, with respect to the displacement u, for compression u < 0.	IDETS
SPRING	Character string specifying that the next 5 fields define a nonlinear elastic spring element in terms of a force versus displacement relationship. (Character).  $F(u) = F_T(u)$ <p>Tension is u &gt; 0 and compression is u &lt; 0.</p>	
DAMPER	Character string specifying that the next 5 fields define a nonlinear viscous element in terms of a force versus velocity relationship. (Character).  $F(v) = F_T(v)$ <p>Tension is v &gt; 0 and compression is v &lt; 0.</p>	
GENER	Character string specifying that the next 7 fields define a general nonlinear elastic spring and viscous damper element in terms of a force versus displacement and velocity relationship. (Character). For this element, the relationship can only be defined with TYPE=EQUAT.  $F(u, v) = F_T(u, v)$ <p>Tension is u &gt; 0 and compression is u &lt; 0. For SPRING, DAMPER, and GENER, the remaining fields are</p>	
TYPE	Character string indicating the type of definition. (Character). For TYPE = EQUAT the following fields are identification numbers of DEQATN entries. For TYPE = TABLE the following field is an identification number of a TABLEDi entry. For "GENER", TYPE = blank or "EQUAT" is allowed..	Required for SPRING or DAMPER
IDT	Identification number of a DEQATN entry for tension if TYPE = EQUAT. Identification number of a TABLEDi entry for tension and compression if TYPE = TABLE.	Required for SPRING, DAMPER, and GENER
IDC	Identification number of a DEQATN entry for compression if TYPE = EQUAT. Is ignored for TYPE = TABLE.	IDT

Describer	Meaning	Default
IDTDU	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for tension $u > 0$ . For SPRING and GENER only.	Required if TYPE=EQUAT
IDCDU	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the displacement u, for compression $u < 0$ . For SPRING and GENER only.	IDTDU
IDTDV	Identification number of a DEQATN entry for tension if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for tension $v > 0$ . For DAMPER and GENER only.	Required if TYPE=EQUAT
IDCDV	Identification number of a DEQATN entry for compression if TYPE = EQUAT. The DEQATN entry defines the derivative of the force F with respect to the velocity v, for compression $v < 0$ . For DAMPER and GENER only.	IDCDT

### Remarks:

1. For a linear spring K a default value of 0.0 is allowed by not specifying the K entry. For a linear damper C a default value of 0.0 is allowed by not specifying the C entry. If any nonlinearity is specified (SHOCKA, SPRING, DAMPER, GENER)  $K > 0$  is required. If K is not provided in the presence of any nonlinearity, the element will behave as a linear element.
2. The damping C and mass M are ignored in static solution sequences.
3. The parameters defined on the continuation entries are used in nonlinear solution sequences only.
4. The linear parameters K and C are used in all solution sequences unless parameters on continuation entries are defined and a nonlinear solution sequence is used. Then, the parameters K and C are used for initial values in the first iteration of the first load step and the parameters from continuation entries overwrite the linear parameters thereafter. When SHOCKA, SPRING or GENER are specified, K is overwritten. When SHOCKA, DAMPER or GENER is specified, C is overwritten.
5. PBUSH1D may only be referenced by CBUSH1D elements in the residual structure which do not attach to omitted degrees-of-freedom.
6. The continuation entries SHOCKA, SPRING, DAMPER and GENER may be specified in any order. If more than one continuation entry is defined, then the forces of SHOCKA, SPRING, etc. are added. Multiple continuation entries of the same kind are not allowed, for example, multiple SPRING continuation entries.
7. For TYPE = TABLE, values on the TABLEDi entry are for tension and compression. If table values  $f(u)$  are provided only for positive values  $u > 0$ , then it is assumed that  $f(-u) = -f(u)$ .

8. For TYPE = EQUAT, the equations for tension and compression can be different. If the identification numbers for compression are left blank, it is assumed that the equation for tension is also valid for compression.
9. PBUSH1D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH1D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH1D entries.
10. The force output is:

```

F O R C E S   I N   B U S H 1 D   E L E M E N T S   ( C B U S H 1 D )

AXIAL      AXIAL      AXIAL      AXIAL      AXIAL      PLASTIC
FORCE      DISPLACEMENT VELOCITY   STRESS     STRAIN     STRAIN     STATUS
  
```

Where items in red are always output for linear analysis. The Stress output is:

```

S T R E S S E S   ( F O R C E S )   I N   B U S H 1 D   E L E M E N T S   ( C B U S H 1 D )

AXIAL      AXIAL      AXIAL      AXIAL      AXIAL      PLASTIC
FORCE      DISPLACEMENT VELOCITY   STRESS     STRAIN     STRAIN     STATUS
  
```

Where items in red are always output for linear analysis.

$$(\text{Axial stress}) = SA \times (\text{Axial Force}), (\text{Axial strain}) = SE \times (\text{Axial Displacement}).$$

In nonlinear dynamics, the element force is the sum of all springs and dampers.

In linear dynamic solution sequences, the damping forces are not included in the element force output.

Failed status is not currently used.

11. While computing Single Point Forces of Constraint (SPCFORCES), the constraint relationships exist in stiffness for linear spring. However, the nonlinear elements treat the constraints as loads on set. As a result, the SPCFORCES results are different between linear and nonlinear elements.

## PBUSH2D 2-D Linear/Nonlinear Connection Properties

Defines linear and nonlinear properties of a two-dimensional element (CBUSH2D entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PBUSH2D	PID	K11	K22	B11	B22	M11	M22		
	SQUEEZE	BDIA	BLEN	BCLR	SOLN	VISCO	PVAPCO		
	NPORT	PRES1	THETA1	PRES2	THETA2	OFFSET1	OFFSET2		

### Example:

PBUSH2D	1000	50.0	150.0	0.02	0.02				
	SQUEEZE	1.0	2.0	0.05	LONG	2.1	300.0		
	2	100.0	30.0	120.0	90.0	0.01	0.0		

OR (to include cross coupling terms)

PBUSH2D	PID	K11	K22	B11	B22	M11	M22		
	CROSS	K12	K21	B12	B21	M12	M21		

Descriptor	Meaning
PID	Property identification number (Integer > 0; Required).
K11	Nominal stiffness in T1 rectangular direction (Real; Required).
K22	Nominal stiffness in T2 rectangular direction (Real; Required).
B11	Nominal damping in T1 rectangular direction (Real; Default = 0.0).
B22	Nominal damping in T2 rectangular direction (Real; Default = 0.0).
M11	Nominal acceleration-dependent force in T1 direction (Real; Default = 0.0).
M22	Nominal acceleration-dependent force in T2 direction (Real; Default = 0.0).
'SQUEEZE'	Indicates that squeeze-film damper will be specified (Character; Required).
BDIA	Inner journal diameter. (Real > 0.0; Required)
BLEN	Damper length. (Real > 0.0; Required).
BCLR	Damper radial clearance (Real > 0.0; Required).
SOLN	Solution option: LONG or SHORT bearing (Character; Default = LONG).
VISCO	Lubricant viscosity (Real > 0.0; Required).
PVAPCO	Lubricant vapor pressure (Real; Required).
NPORT	Number of lubrication ports: 1 or 2 (Integer; no Default).
PRES1	Boundary pressure for port 1 (Real ≥ 0.0; Required if NPORT= 1 or 2).

Describer	Meaning
THETA1	Angular position for port 1 ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT= 1 or 2). See Remark 2.
PRES2	Boundary pressure for port 2 ( $\text{Real} \geq 0.0$ ; Required if NPORT= 2).
THETA2	Angular position for port 2 ( $0.0 \leq \text{Real} < 360.0$ ; Required if NPORT= 2). See Remark 2.
OFFSET1	Offset in the SFD direction 1, see Remark 3. (Real; Default = 0.0).
OFFSET2	Offset in the SFD direction 2, see Remark 3. (Real; Default = 0.0)
CROSS	Indicates that cross coupling terms will be provided.
K12, K21	Cross coupling stiffness terms.
B12, B21	Cross coupling damping terms.
M12, M21	Cross coupling mass terms.

**Remarks:**

1. Either SQUEEZE or CROSS option can be used. Both options cannot be specified. SQUEEZE option will be supported only in SOL400. When SQUEEZE option is used, the squeeze film damper model force calculation is invoked using all the entries specified.
2. The angular measurement is counterclockwise from the displacement x-axis for the XY plane, the y-axis for the YZ plane, and the z-axis for the XZ plane.
3. Offsets are measured from GB relative to GA. For example, if direction 2 is in the vertical direction and a gravity load is placed on GA, OFFSET2 will be a positive value (GB ‘moves’ toward GA in the positive direction 2).
4. PBUSH2D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PBUSH2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PBUSH2D entries.
5. PBUSH2D may be used in conjunction with THPAD obtained from ROMAC for rotordynamic applications. Also see ELEMUDS and THPAD bulk data entries.
6. The template used for specifying matrices using cross entries is:

$$[I]_{2 \times 2} = \begin{bmatrix} I_{11} & I_{12} \\ I_{21} & I_{22} \end{bmatrix}$$

where I=K,B or M



**PBUSHT**

Frequency Dependent or Nonlinear Force Deflection Spring and Damper Property

Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.

**Format:**

1	2	3	4	5	6	7	8	9	10
PBUSHT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“B”	TBID1	TBID2	TBID3	TBID4	TBID5	TBID6	
		“GE”	TGEID1	TGEID2	TGEID3	TGEID4	TGEID5	TGEID6	
		“KN”	TKNID1	TKIND2	TKNID3	TKIND4	TKIND5	TKIND6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FSRS	LRGR					

**Example:**

PBUSHT	35	K	72						
		B	18						

Descriptor	Meaning
PID	Property identification number that matches the identification number on a PBUSH entry. (Integer > 0)
“K”	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)
TKID <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)
“B”	Flag indicating that the next 1 to 6 fields are force per velocity frequency table identification numbers. (Character)
TBID <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the force per unit velocity damping vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)
“GE”	Flag indicating that the next field is a structural damping frequency table identification number. (Character)
TGEID <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the non-dimensional structural damping vs. frequency relationship. (Integer ≥ 0; TGEID1 default = 0, for TGEID2-6 see defaults described in Remark 6.)
“KN”	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)
TKNID <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)



Describer	Meaning
FDC	Force deflection curve rule. Specifies a dependence between displacement components. See Remark 16. for use with the FUSE option. (Character or blank default blank)
"NR" or blank	Implies there is no force deflection rule. T1-R6 directions are all independent of each other.
"TRXY"	A radial dependence exists between $u_x$ and $u_y$ element displacements. Either K1 or K2 may be specified on the PBUSH entry. If both are specified then K1 will be used. On the PBUSHT "KN" entry either TKNID1 or TKNID2 may be specified. If both are specified then TKNID1 is used. The expression $u_r = \sqrt{(u_x)^2 + (u_y)^2}$ is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_x$ and $f_y$ in the springs "x" and "y" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current $u_r$ .
"TRXZ"	A radial dependence exists between $u_x$ and $u_z$ element displacements. Either K1 or K3 may be specified on the PBUSH entry. If both are specified then K1 is used. On the PBUSHT "KN" entry either TKNID1 or TKNID3 may be specified. If both are specified then TKNID1 is used. The expression $u_r = \sqrt{(u_x)^2 + (u_z)^2}$ is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_x$ and $f_z$ in the springs "x" and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current $u_r$ .
"TRYZ"	A radial dependence exists between $u_y$ and $u_z$ element displacements. Either K2 or K3 may be specified on the PBUSH entry. If both are specified then K2 is used. On the PBUSHT "KN" entry either TKNID2 or TKNID3 may be specified. If both are specified then TKNID2 is used. The expression $u_r = \sqrt{(u_y)^2 + (u_z)^2}$ is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using $u_r$ . Under the OPTION=RULTLD, the true forces $f_y$ and $f_z$ in the springs "y" and "z" are used for the fuse calculation. New K2 is returned from slope of load-deflection curve at slope at current $u_r$ .

Describer	Meaning
	<p>“TS”</p> <p>A spherical dependence exists between <math>u_x</math>, <math>u_y</math>, and <math>u_z</math> element displacements. Either K1, K2 or K3 may be specified on the PBUSH entry. If all three are specified K1 is used. If only K2 and K3 are specified then K2 is used. On the PBUSHT “KN” entry TKNID1, or TKNID2 or TKNID3 may be specified. If all three are specified then TKNID1 is used. If only TKNID2 and TKNID3 are specified then TKNID2 is used. The expression</p> $u_s = \sqrt{(u_x)^2 + (u_y)^2 + (u_z)^2}$ <p>is used for table lookup. Under the OPTION=ULTLD, the force used for the fuse calculation is computed using <math>u_p</math>. Under the OPTION=RULTLD, the true forces <math>f_x</math>, <math>f_y</math> and <math>f_z</math> in the springs "x", "y", and "z" are used for the fuse calculation. New K1 is returned from slope of load-deflection curve at slope at current <math>u_s</math></p>
FUSE	<p>0 The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)</p> <p>1 The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for postprocessing.</p> <p>2 The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from postprocessing.</p>
DIR	<p>The fuse direction component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks; Default = 0)</p> <p>0 Any direction is allowed to fuse. Same if user entered 123456.</p> <p>1 through 6 may be placed in the field with no embedded blanks. See Remark 13. for use with FDC.</p>
OPTION	<p>Specifies a failure mode. (Character - Default="RELDIS")</p> <p>“ULTLD” The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship.</p> <p>“RELDIS” The specified max relative <math>\pm</math> displacement will be used to define failure.</p> <p>“RULTLD” The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.</p>
LOWER	<p>Lower failure bound. (Real; Default=0.0)</p> <p>If OPTION = “ULTLD” then LOWER specifies a lower failure load</p> <p>If OPTION = “RELDIS” then LOWER specifies a minimum relative displacement before failure.</p>
UPPER	<p>Upper failure bound. (Real; Default=0.0)</p>

Describer	Meaning
	If OPTION = "ULTLD" then UPPER specifies an upper failure load. If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.
FSRS	Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real $\geq$ 0.0; Default = 1.-5)
LRGR	Specifies if large rotation is to occur at end A. (Integer $\geq$ 0; Default = 0). See Remark 14.

**Remarks:**

1. The "K", "B", and "GE" fields are associated with same entries on the PBUSH entry.
2. PBUSHT may only be referenced by CBUSH elements in the residual structure which do not attach to any omitted degrees-of-freedom of ASET and/or dependent DOFs of MPC/Rigid elements. Otherwise, the solution is an approximation based on nominal properties.
3. For nonlinear analysis the nominal values are used at the beginning of the analysis to compute initial loading of the CBUSH element and meaningful values should be supplied for stiffness. As the nonlinear analysis proceeds, the user supplied load deflection curves are used to compute new stiffness values. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency.
4. The "K", "B", "GE" or "KN" fields may be specified in any order.
5. The PBUSHT is used in SOL 108 and SOL 111 when any "K", "B", or "GE" is specified. It is used in SOL 106, SOL 129 or SOL 400 when "KN" is specified. It is ignored in all other solution sequences.
6. For upward compatibility, if ONLY TGEID1 is specified on a PBUSHT entry and TGEID<sub>i</sub>, i = 2 – 6 are all blank on the PBUSHT entry, then the single structural damping table is applied to all defined GE<sub>i</sub> for the PBUSH entry. In this case, TGEID<sub>i</sub>, i = 2 – 6, blank is not equivalent to 0 but a special flag.

A PBUSHT entry that has any TGEID<sub>i</sub>, i = 2 – 6 numerically specified including a 0, all the fields TGEID<sub>i</sub> are then treated separately and any TGEID<sub>i</sub>, i=2 - 6 field with no entry specified is defaulted to a 0 value and the results are radically different from the no TGEID<sub>i</sub>, i=2 – 6 specified as described above.

Thus for the entry:

1	2	3	4	5	6	7	8	9	10
PBUSHT	3303000	K	33030001						
		GE	33030002						

The GE table ID 3303002 will be replicated to fields 5 through 9. While:

1	2	3	4	5	6	7	8	9	10

PBUSHT	3303000	K	33030001						
		GE	33030002	0					

The 0 in FIELD 5 of the above PBUSHT above negates table 3303002 being replicated, hence, fields 5 through 9 will not have any associated table lookup.

7. For nonlinear analysis, only the “KN” line and optionally its two continuation lines are used.
8. For frequency responses, only the “K”, “B” and/or “GE” fields are used.
9. The continuations entries to “KN” are optional.
10. When the FDC field specifies one of the optional radial or spherical rules, stiffness entries not involved in the rule may still have their own independent force-deflection curves.
11. If FUSE=0, the remaining entries are ignored. If FUSE > 0, the “OPTION”, “UPPER”, and “LOWER” fields must be specified.
12. UPPER > LOWER required.
13. The following relationships exists between FDC and DIR

FDC = “TRXY”	1 or 2 or 12 in the DIR field applies to the radial rule	a 3 in the DIR field will be ignored.
FDC = “TRXZ”	1 or 3 or 13 in the DIR field applies to the radial rule	a 2 in the DIR field will be ignored.
FDC = “TRYZ”	2 or 3 or 23 in the DIR field applies to the radial rule	a 1 in the DIR field will be ignored.
FDC = “TS”	Any unique combination of the integers 1 through 3 with no embedded blanks in the DIR field applies to the spherical rule	

14. LRGR = 0 (Default) the element coordinate system is rotated with the rotation of grid A for both the CID and the V vector. LRGR = 1 will suppress large rotation at end A. The initial CID and the V vector will remain unchanged. LRGR = 2 a mid-increment method is used to rotate the element system for the V vector. LRGR = 2 is deactivated if CID is not blank. When CID is blank, LRGR = 2 is recommended for SOL400 when large rotations are present as the mid-increment method “averages” the rotations at both ends of the CBUSH and allows rotation when end A might have a rotational constraint.
15. When the fuse criteria is reached in one of the stiffness components, the entire element is assumed to fail. The component stiffnesses are then computed to be  $K = K \cdot \text{FSRS}$  at initial failure.

16. The figures below depict, using FDC=TRYZ as a typical example, the OPTION="ULTLD" and OPTION="RULTLD". Under the OPTION="ULTLD" the resultant Fr load when FUSED will be

$$F_r = \sqrt{F_y^2 + F_z^2} \approx \text{LIMIT} .$$

Under the OPTION="RULTLD" the resultant Fr load when

FUSED will in general be  $F_r = \sqrt{F_y^2 + F_z^2} > \text{LIMIT}$  with one of the  $F_y$  or  $F_z \approx \text{LIMIT}$  . For FDC=TRYZ and DIR=123, the T1 direction is not included in the force direction curve rule, thus possible fusing in the T1 direction is always based on the actual T1 spring displacement and  $F_x$  if active will be  $F_x \approx \text{LIMIT}$  . For FDC=NR fusing is always calculated based on each individual spring displacement and the actual individual spring loading.

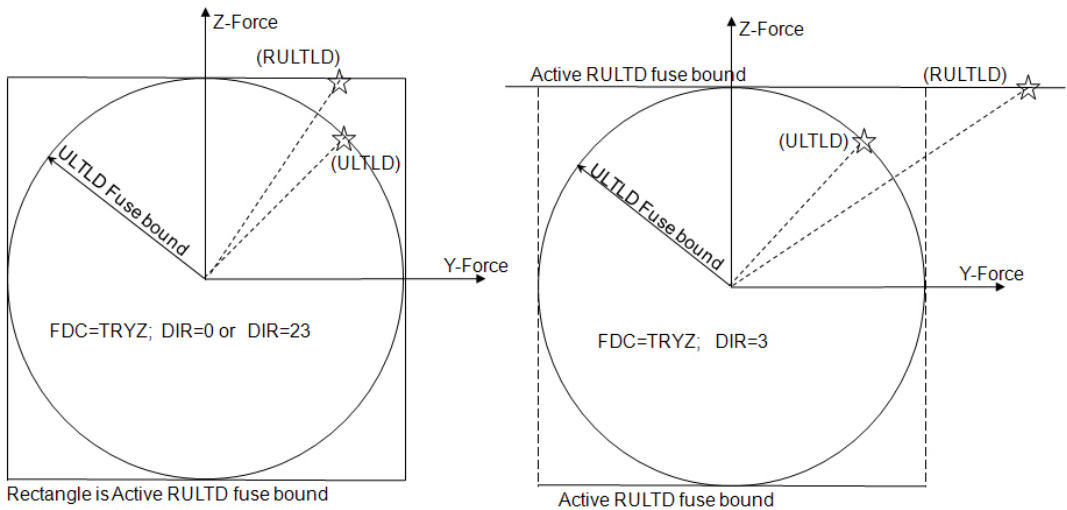


Figure 9-120 FUSE Models

## PCOHE

### Interface Cohesive Zone Modeling Element Properties

Defines the properties of a fully nonlinear element used to simulate the onset and progress of delamination in SOL 400.

#### Format:

1	2	3	4	5	6	7	8	9	10
PCOHE	PID	MID	INT	T	OUTPUT	SECANT			

#### Example:

PCOHE	700	701							
-------	-----	-----	--	--	--	--	--	--	--

Describer	Meaning
PID	Property identification. (Integer > 0)
MID	Identification number of a MCOHE entry. (Integer > 0)
INT	Integration scheme. 0 or blank Gauss integration, 1 for Newton-Coates/Lobatto integration. See Remark 3. (Integer 0 or 1; Default 0)
T	Thickness. See Remark 2. (Real > 0; Default = 1.0)
OUTPUT	Location selection for stress/strain output. If OUTPUT=GRID or blank, output is at the corner grid points. If OUTPUT=GAUSS output at the Gauss points. If INT=1, OUTPUT will be set to GRID. (Character or Blank; Default Grid)
SECANT	Tangent matrix scheme. 0 or blank a secant-type matrix is used to set up the element stiffness matrix, 1 a tangent-type matrix is used to set up the element stiffness matrix. (Integer 0 or 1; Default 0)

#### Remarks:

1. PCOHE can be referenced by CIFQUAD, CIFHEX, CIPENT, and CIFQDX entries.
2. The thickness T applies only to CIFQUAD elements.
3. For initially very stiff interface elements, the Newton-Coates/Lobatto integration scheme may be preferred.
4. PCOHE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOHE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PCOHE entries

## PCOMP Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

### Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	MID3	T3	THETA3	SOUT3	-etc.-				

### Example of multiple plies per line format:

PCOMP	181	-0.224	7.45	10000.0	HOFF				
	171	0.056	0.	YES			45.		
			-45.				90.		

### Example of single ply per line format:

PCOMP	181	-0.224	7.45	10000.	HOFF				
	171	.056	0.	YES					
	171	.056	45.	YES					
	171	.056	-45.	YES					
	171	.056	90.	YES					

Descriptor	Meaning
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remarks 10. and 11. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 7. "HILL" for the Hill theory. "HOFF" for the Hoffman theory. "TSAI" for the Tsai-Wu theory. "STRN" for the Maximum Strain theory. "HFAIL" for the Hashin failure criterion "HTAPE" for the Hashin tape criterion "HFABR" for the Hashin fabric criterion
TREF	Reference temperature. See Remark 3. (Real; Default = 0.0)



Describer	Meaning
GE	Damping coefficient. See Remarks 4. and 12. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 13. and 14.
	<p>“Blank” All plies must be specified and all stiffness terms are developed.</p> <p>“SYM” Only plies on one side of the element centerline are specified. The plies are numbered starting with 1 for the bottom layer. If an odd number of plies are desired, the center ply thickness (T1) should be half the actual thickness.</p> <p>“MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.</p> <p>“BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.</p> <p>“SMEAR” All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set as blanks).</p> <p>“SMCORE” All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.</p>
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDi must refer to MAT1, MAT2, MAT8, or MATDIGI Bulk Data entries. See Remarks 1. 15. and 18. For use with MAT4 and MAT5, see Remark 20. (Integer > 0 or blank, except MID1 must be specified.)
Ti	Thicknesses of the various plies. See Remarks 1. (Real or blank, except T1 must be specified.)
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUTi	Stress or strain output request. See Remarks 5. and 6. (Character: “YES” or “NO”; Default = “NO”)

**Remarks:**

1. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
2. At least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.
3. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMP entry is used for all plies of the element. If not specified, it defaults to “0.0.”

If the PCOMP references temperature dependent material properties, then the TREF given on the PCOMP will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMP entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILON\_T is also set to INTEGRAL, TREF is not applicable.

4. GE given on the PCOMP entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMP entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GE<sub>ij</sub> values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMP entry and use the given GE<sub>ij</sub> values.
5. Stress and strain output for individual plies are available in all superelement static, normal modes, buckling, frequency response (direct and modal), transient response (direct and modal) and nonlinear static analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XY PLOT command.
6. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark 10.
7. In order to get failure index output the following must be present:
  - a. STRESS or STRAIN Case Control commands,
  - b. SB, FT, and SOUT<sub>i</sub> on the PCOMP Bulk Data entry,
  - c. X<sub>t</sub>, X<sub>c</sub>, Y<sub>t</sub>, Y<sub>c</sub>, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or X<sub>t</sub>, X<sub>c</sub>, Y<sub>t</sub>, S, and STRN=1.0 if strain allowables are used.
  - d. -1 - failure in the fiber direction.  
-2 - failure in the matrix direction  
-12 - failure in the inplane shear
  - e. For Hashin Failure criterion, HFi (i=1,2,3,4,10,11) must be present in all referenced MAT8 bulk entry.
  - f. For Hashin Tape criterion, HTi (i=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.
  - g. For Hashin Fabric criterion, HFBi (i=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
- 1 - failure in the tensile first fiber mode.
  - 1 - failure in the compressive first fiber mode.
  - 2 - failure in the tensile second fiber mode.
  - 2 - failure in the compressive second fiber mode.
  - 3 - failure in the tensile matrix mode.
  - 3 - failure in the compressive matrix mode.
8. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
  9. The failure index for the bonding material is calculated as  $\text{Failure Index} = (\tau_{1z}, \tau_{2z})/SB$ .
  10. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry.
  11. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
  12. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
  13. The SYM option for the LAM option computes the complete stiffness properties while specifying half the plies. The MEM, BEND, SMEAR and SMCORE options provide special purpose stiffness calculations. SMEAR ignores stacking sequence and is intended for cases where this sequence is not yet known, stiffness properties are smeared. SMCORE allows simplified modeling of a sandwich panel with equal face sheets and a central core.
  14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
  15. Temperature-dependent ply properties only available in SOL 106 and SOL 400. See PARAM,COMPMTT for details.
  16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.

17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMPF entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.
18. PCOMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.
19. For PCOMP, the non-default Zo option should not be used in conjunction with MEM, BEND, SMEAR or SMCORE as wrong results may occur. These four options provide special purpose stiffness calculations wherein stacking sequence effects are ignored (membrane-bending coupling terms are set to zero) at the preliminary design stage level. A better choice for offsets is to use the ZOFF option on the connectivity entries (CQUAD4, CTRIA3 etc). A User Fatal Message is issued if the user specifies a non-default value of Zo for any of the above smearing options. The fatal message may be changed to a User Warning Message by specifying a positive value for system cell 668 (e.g. nastran system(668)=1) to allow the program to continue.
20. In SOL400 only, this entry may refer to MAT4 or MAT5 entries. If it does, and the user has not supplied PSHLN1 entries, Nastran will internally generate a PSHLN1 entry. For details, the user should refer to Remark 7. of the PSHLN1 entry.
21. In layer composite output, the traditional element model outputs stress or strain separately by using STRESS or STRAN Case Control Command, respectively.  
  
The advanced element model does not recognize STRAIN Case Control Command. As long as user gives STRESS Case Control Command, both stress and strain are outputted together (similar to NLSTRESS).

**PCOMPA**

## Additional Data for Layered Composite Element Property - SOL 700

Defines additional properties of a multi-ply laminate composite material. Used in SOL 700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
PCOMPA	PID	FORM	SHFACT	REF	STRDEF	DT1D	STRNOUT	CLT	
+	SPINCOR								

**Example:**

PCOMPA	1	BLT						1	
+	YES								

Describer	Meaning
PID	Unique property number referring to a PCOMP property number. (Integer > 0; Required)
FORM	Element formulation. (Character; See Remark 1.)
SHFACT	Shear correction factor, see Remark 4. (Real $\geq$ 0.0; default=0.83333)
REF	Reference surface. (Character; default=MID)
	TOP            Reference surface is the top of the surface.
	MID            Reference surface is the central surface.
	BOT            Reference surface is the bottom surface.
STRDEF	Definition in stress-strain output. (Character; default=FIBER)
	FIBER            Stresses defined in the fiber and matrix directions.
	ELEM            Stresses defined in the element coordinate system
DT1D	Time step skip for one-dimensional failure modes. See Remark 2. (Character; default=NO)
	YES            Skip one-dimensional failure modes.
	NO            Normal time-step calculation.
STRNOUT	Strain output option. See Remarks 3. and 4. (Character; default=YES)
	YES            Total strain is calculated.
	NO            No strain is stored in memory.
ICLT	Option to use Classical Lamination Theory. See Remark 4. (Integer > 0; default=0)
	1            Use the Classical Lamination Theory.
	0            Use the integration technique.
SPINCOR	Spin correction. See Remark 4. (Character; default=NO)

Describer	Meaning
NO	No SPINCOR applied
YES	SPINCOR applied

**Remarks:**

- For CQUAD4 elements, the default formulation is Key-Hoff. For CTRIA3 elements, the default formulation is C0-TRIA.
- If the failure mode is such that fiber and shear strength or matrix and shear strength are lost in all layers, the element is not included in the time-step calculation. If the element fails completely, the element is omitted from the time-step calculations, irrespective of the value entered in this field.
- If the STRNOUT field is NO, the strain cannot be output.
- If ICLT is set to 1, the analysis is performed with classical lamination theory. In this case, it is not possible to request the total strain output. The (transverse) shear correction factor input is ignored since it is calculated inside Nastran. There is no update of the cross-sectional properties due to failure. The failure flag only indicates that the failure condition is satisfied. Additional output for element variables is available, namely the stress resultants (NXX, NYY, NXY, MXX, MYY, MXY, QYZ, and QZX). Also the ABD-Q matrices of each element can be requested for output. These data are only stored in the first layer. The variable names are AijM, BijM, DijM, and QsijM for the components of the A-, B-, D- and Q-matrices, respectively. For example, to request the A11 of the A-matrix, the variable name is A11M01.
- The options for SPINCOR are:
 

NO	No SPINCOR correction is applied.
YES	A SPINCOR correction is applied.

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

The SPINCOR option is required for fabric models and is turned on by default to accurately keep track of the fiber directions.

**PCOMPF****Integration Procedure Used in Conjunction with PCOMP or PCOMPG**

Defines the integration procedure for through the thickness integration of composite shells in SOL 600 and SOL 400 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID1	THRU	PID2	BY	N			

**Alternate Formats (SOL 400):**

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	PID1	PID2	PID3	THRU	PID4	PID5	THRU	
	PID6	PID7	TO	PID8	PID9	PID10	PID11	PID12	
	THRU	PID13	BY	N					

1	2	3	4	5	6	7	8	9	10
PCOMPF	INT	ALL							

**Examples:**

PCOMPF	2	100	THRU	200	BY	10			
--------	---	-----	------	-----	----	----	--	--	--

**Example of Application to Single PID (SOL 400):**

PCOMPF	1		23	TO	25		33		
	44	THRU	54	BY	2				
	100								

Describer	Meaning
INT	1 Conventional through the thickness integration of each layer, allows all available material behavior through the thickness. (Default) 2 Linear elastic material, fast-integrated through the thickness - thermal strains and temperature dependent material properties are not allowed. 3 Linear elastic material, fast integrated through the thickness.
PIDi	Property identification number. (0 < Integer < 10000000) corresponds to a matching PCOMP or PCOMPG entry.
N	Property identification number increment. See Remark 7.. (Integer or blank)

### Remarks:

1. If nonlinear behavior is set on a MATS1 (or other option), but INT is  $> 1$ , then the nonlinear material behavior is ignored.
2. In SOL 600 only, if temperature dependent behavior is specified on a MATT1 or similar option and INT=2, the material values specified on the MATT1, MATT2, MATT8 option are ignored (the values on MAT1, MAT2 and MAT8 are used). In SOL 400, temperature loading or temperature dependent behavior should not be used with INT=2, a user fatal will be issued in this case. INT=3 can be used instead.
3. If more than one PCOMPF exists with different INT values, and there is an overlap in PID's, that is to say a given PID could have been assigned different values of INT, then a user fatal will be issued.
4. With the "THRU" and "THRU", "BY" forms, blank fields are allowed for readability. Any combination of a list of ID's and "THRU" and "THRU", "BY" is allowed. The "THRU" and "BY" lists may have missing ID's. That is the list of ID's in a THRU range need not be continuous.
5. In SOL 600 only, if all composite shells are to use the same INT value, it may be entered with PARAM,MFASTCMP instead of PCOMPF.
6. In SOL 600 only, if a "THRU" or "THRU", "BY" range is entered, all items associated with the range must be on the same line (or for large field a line and the continuation entry of that line).
7. For automatic generation of property identification numbers, the default increment value is 1 if property identification numbers are increasing or -1 if property identification numbers are decreasing (i.e., the user need not specify BY and the increment value).



## PCOMPFG      Frequency Dependent Composite structural damping Property

Defines the frequency dependent properties for a PCOMP/PCOMPG Bulk Data entry.

### Format:

1	2	3	4	5	6	7	8	9	10
PCOMPFG	PID						GE		

### Examples:

PCOMPFG	33						15		
---------	----	--	--	--	--	--	----	--	--

Describer	Meaning
PID	Property identification number that matches the identification number on PCOMP/PCOMPG entry. (Integer > 0)
GE	Identification number of a TABLEDi entry that defines the non-dimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0)

### Remarks:

1. The Fields 8 of this entry corresponds to Field 8 of a PCOMP or PCOMG entry. The value in Field 8 of the PCOMP or PCOMPG entry is replaced by the table referenced in the corresponding field of this entry.
2. If the PCOMP/PCOMG MIDi fields point to MAT2 entries with the 2nd continuation fields containing GEij entries, then this entry is ignored for that MIDi entry and GE frequency dependency must be indicated by use of the MAT2F entry with same ID as the MIDi entry.
3. IF GE=0.0 on corresponding PCOMP or PCOMPG then GE table must be blank or 0.

## PCOMP

## Layered Composite Element Property (Alternate to PCOMP Entry)

Defines global (external) ply IDs and properties for a composite material laminate.

### Format:

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
	GPLYID1	MID1	T1	THETA1	SOUT1	GEFLG			
	GPLYID2	MID2	T2	THETA2	SOUT2				

### Example of single ply per line format:

PCOMP	181	-0.224	7.45	10000.	HOFF				
	1001	171	.056	0.	YES				
	101	171	.07	45.	YES				
	2002	171	.056	-45.	YES				
	102	171	0.55	90.	YES				

Describer	Meaning
PID	Property identification number. (0 < Integer < 10000000)
Z0	Distance from the reference plane to the bottom surface. See Remark 11. (Real; Default = -0.5 times the element thickness.)
NSM	Nonstructural mass per unit area. (Real)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). Required if FT is also specified. (Real > 0.0)
FT	Failure theory. The following theories are allowed (Character or blank. If blank, then no failure calculation will be performed) See Remark 8. "HILL" for the Hill theory. "HOFF" for the Hoffman theory. "TSAI" for the Tsai-Wu theory. "STRN" for the Maximum Strain theory. "HFAIL" for the Hashin failure criterion "HTAPE" for the Hashin tape criterion "HFABR" for the Hashin fabric criterion
TREF	Reference temperature. See Remark 4. (Real; Default = 0.0)
GE	Damping coefficient. See Remark 5. (Real; Default = 0.0)
LAM	Laminate Options. (Character or blank, Default = blank). See Remarks 14. "Blank" All plies must be specified and all stiffness terms are developed.

Describer	Meaning
	<p>“MEM” All plies must be specified, but only membrane terms (MID1 on the derived PSHELL entry) are computed.</p> <p>“BEND” All plies must be specified, but only bending terms (MID2 on the derived PSHELL entry) are computed.</p> <p>“SMEAR” All plies must be specified, stacking sequence is ignored MID1=MID2 on the derived PSHELL entry and MID3, MID4 and TS/T and 12I/T**3 terms are set to zero).</p> <p>“SMCORE” All plies must be specified, with the last ply specifying core properties and the previous plies specifying face sheet properties. The stiffness matrix is computed by placing half the face sheet thicknesses above the core and the other half below with the result that the laminate is symmetric about the mid-plane of the core. Stacking sequence is ignored in calculating the face sheet stiffness.</p>
GPLYIDi	User-defined Global (External) Ply ID. See Remark 1. (Integer > 0)
MIDi	Material ID of the various plies. The plies are identified by serially numbering them from 1 at the bottom layer. The MIDs must refer to MAT1, MAT2, or MAT8 Bulk Data entries. See Remarks 2. and 15. For use with MAT4 and MAT5, see Remark 20. (Integer > 0 or blank, except MID1 must be specified.)
Ti	Thicknesses of the various plies. See Remark 2. (Real or blank, except T1 must be specified.)
THETAi	Orientation angle of the longitudinal direction of each ply with the material axis of the element. (If the material angle on the element connection entry is 0.0, the material axis and side 1-2 of the element coincide.) The plies are to be numbered serially starting with 1 at the bottom layer. The bottom layer is defined as the surface with the largest -Z value in the element coordinate system. (Real; Default = 0.0)
SOUTi	Stress or strain output request. See Remarks 6. and 7. (Character: “YES” or “NO”; Default = “NO”)
GEFLG	Ply structural damping flag may be only entered only once and if entered must be entered on the first ply. At least two layers are required else frequency dependency is ignored. For a one-layer PCOMPG use PCOMFQ entry for frequency dependency. (Integer -2, or -1; Default=0) See Remark 19.

### Remarks:

1. The global ply identification number must be unique with respect to other plies in the entry. The plies are defined in stacking sequence starting with the bottom layer.
2. The default for MID2, ..., MIDn is the last defined MIDi. In the example above, MID1 is the default for MID2, MID3, and MID4. The same logic applies to Ti.
3. The global ply ID (GPLYIDi) and at least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist. The minimum number of plies is one.

4. The TREF specified on the material entries referenced by plies are not used. Instead TREF on the PCOMPG entry is used for all plies of the element. If not specified, it defaults to “0.0.”

If the PCOMPG references temperature dependent material properties, then the TREF given on the PCOMPG will be used as the temperature to determine material properties.

TEMPERATURE Case Control commands are ignored for deriving the equivalent PSHELL and MAT2 entries used to describe the composite element.

If for a nonlinear static analysis the parameter COMPMATT is set to YES, the temperature at the current load step will be used to determine temperature-dependent material properties for the plies and the equivalent PSHELL and MAT2 entries for the composite element. The TREF on the PCOMPG entry will be used for the initial thermal strain on the composite element and the stresses on the individual plies. If the parameter EPSILONT is also set to INTEGRAL, TREF is not applicable.

5. GE given on the PCOMPG entry will be used for the element and the values supplied on material entries for individual plies are ignored. The user is responsible for supplying the equivalent damping value on the PCOMPG entry. If PARAM,W4 is not specified GE is ignored in transient analysis. When GEij values are present on the MAT2 entry, Nastran will ignore the GE value given on the first continuation entry field (6) of the MAT2 and the GE entry given in field (8) of the PCOMPG entry and use the given GEij values.
6. Stress and strain output for individual plies are available in all superelement static, frequency response (direct and modal), transient response (direct and modal) and normal modes analysis and requested by the STRESS and STRAIN Case Control commands. Composite lamina stress/strain are available in random analysis by XYLOT command.
7. For Nastran conventional elements, if PARAM,NOCOMPS is set to -1, stress and strain output for individual plies will be suppressed and the homogeneous stress and strain output will be printed. See also Remark 11.
8. In order to get failure index output the following must be present:
  - a. STRESS or STRAIN Case Control commands,
  - b. SB, FT, and SOUTi on the PCOMPG Bulk Data entry,
  - c. Xt, Xc, Yt, Yc, and S on all referenced MAT8 Bulk Data entries if stress allowables are used, or Xt, Xc, Yt, S, and STRN=1.0 if strain allowables are used.
  - d. -1 - failure in the fiber direction  
-2 - failure in the matrix direction  
-12 - failure in the inplane shear.
  - e. For Hashin Failure criterion, HFi (i=1,2,3,4,10,11) must be present in all referenced MAT8 bulk entry.
  - f. For Hashin Tape criterion, HTi (i=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.
  - g. For Hashin Fabric criterion, HFBi (i=1,2,3,4,5,6,10,11,12) must be present in all referenced MAT8 bulk entry.

- h. Hashin failure theory output is captured with following code.
- 1 - failure in the tensile first fiber mode.
  - 1 - failure in the compressive first fiber mode.
  - 2 - failure in the tensile second fiber mode.
  - 2 - failure in the compressive second fiber mode.
  - 3 - failure in the tensile matrix mode.
  - 3 - failure in the compressive matrix mode.
9. A function of this entry is to derive equivalent internal PSHELL and MATi entries to describe the composite element. Any sorted echo request will also cause printout and/or punch of the derived PSHELL and MATi entries in User Information Message 4379 and/or the punch file. Use the NASTRAN system cell (361) PRTPCOMP=1 to print equivalent PSHELL/MAT2 Bulk Data entries to the .f06 file. Use the ECHO=PUNCH Case Control command to write them to the .pch file.
  10. The failure index for the bonding material is calculated as  $\text{Failure Index} = (\tau_{1z}, \tau_{2z})/SB$ .
  11. If the value specified for Z0 is not equal to -0.5 times the thickness of the element and PARAM,NOCOMPS,-1 is specified, then the homogeneous element stresses are incorrect, while lamina stresses and element forces and strains are correct. For correct homogeneous stresses, use ZOFFS on the corresponding connection entry. Use of Z0 to offset a laminate does not change the reference plane. Z0 offsets the bottom of the plies from the reference plane.
  12. An unsymmetric layup or the use of Z0 to specify an unsymmetric layup, is not recommended in buckling analysis or the calculation of differential stiffness. Also, Z0 should not be used to specify an unsymmetric layup. The presence of coupling between bending and extension generally increases deflections. Hence, coupling decreases the effective stiffness of a laminate, reduces buckling loads and vibration frequencies significantly.
  13. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
  14. Element output for the SMEAR and SMCORE options is produced using the PARAM NOCOMPS -1 methodology that suppresses ply stress/strain results and prints results for the equivalent homogeneous element.
  15. Temperature-dependent ply properties only available in SOL 106 and SOL400. See PARAM,COMPMMAT for details.
  16. For SOL 600, the default for SOUT is YES for the top and bottom layer and NO for all layers.
  17. For SOL 600, LAM=BLANK if SMEAR is specified on the SOL 600 Executive Control statement. Other LAM options are not available using SOL 600. The default option for SOL 600 is to use complete through the thickness integration for all layers. This is achieved by not entering SMEAR on the SOL 600 entry. Options to speed up complete through the thickness integration by making certain assumptions such as no plasticity in the layers are available using the PCOMP entry. Please note, the meaning of SMEAR for SOL 600 and for other MSC Nastran solution sequences is not the same.

18. PCOMPG is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPG property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHELL, PCOMP, PCOMPG entries.
19. Defines structural damping matrix data for composites. This entry if used MUST appear on the first ply entry ONLY and all subsequent plies follows depending if -1 or -2 described below

GEFLG Value	Meaning
0	The GE value specified in Field 8 of the PCOMPG parent entry is used for computation of all structural damping.
-1	The GE <sub>i</sub> value associated with the MID1 of this ply and the MID <sub>i</sub> with MID( <i>i</i> > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MID <sub>i</sub> for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry, the GE value specified in Field 8 of the PCOMPG parent entry will be used. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping.
-2	The GE <sub>i</sub> value associated with the MID1 of this ply and the MID <sub>i</sub> with MID( <i>i</i> > 1) for all subsequent plies will be used in computing the K4 structural damping properties for the composite. For any MID <sub>i</sub> for which there is no associated GE value on the MAT1, MAT2, or MAT8 entry then that ply will not contribute to the computation of the K4 structural damping of the composite. The resulting smeared value GE will be placed in field 8 of the PCOMPG and used for computing structural damping.
When GEFLG is used, the smeared GE value will appear in the second continuation entry field (6) of the four internally generated equivalent MAT2 entries	
<b>If GEFLG appears with extended MAT2 entries the GEFLG is ignored for that PCOMPG: See Remark of MAT2 entry.</b>	

20. In SOL400 only, this entry may refer to MAT4 or MAT5 entries. If it does, and the user has not supplied PSHLN1 entries, Nastran will internally generate a PSHLN1 entry. For details, the user should refer to Remark 7. of the PSHLN1 entry.

## PCOMPLS

## Layered Solid Composite Element Property

Defines global (external) ply IDs and properties for a composite material laminate in SOL 600, SOL 400, all linear solution sequences between SOL 101 and SOL 112, and analysis only for SOL 200.

### Format:

1	2	3	4	5	6	7	8	9	10
PCOMPLS	PID	DIRECT	CORDM	SB	ANAL				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	"C20"	BEH20	INT20	BEH20H	INT20H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

### Example:

PCOMPLS	782	1							
	1001	171	.3	12.3					
	100	175	.7	77.7					

Describer	Meaning
PID	Property identification number. (Integer > 0)
DIRECT	The layer direction for BEHi = SLCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer inputs is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer ± 1, ±2, or ±3; Default +1)
CORDM	Identification number of the material coordinate system. See Remark 10. (Integer; Default = 0, which is basic)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real ≥ 0.0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL='ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C8	Keyword indicating that two items following apply to elements with eight corner grids. (Character)
C20	Keyword indicating that two items following apply to elements with eight corner grids and twelve mid-side grids. (Character)
BEHi	Element structural behavior. See Remarks 4. and 7. (Character default: SLCOMP for BEH8 and BEH20)
INTi	Integration scheme. See Remark 9. (Character default: L for INT8, Q for INT20)



BEHiH	Element heat behavior. See Remarks 4. and 8. (Character Default: SLCOMP for BEH8H and BEH20H)
INTiH	Integration scheme. See Remark 9. (Character Default: L for INT8H, Q for INT20H)
IDi	Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)
MIDi	Material ID for the ply. See Remark 3. (Integer > 0)
Ti	Either fractional percent of the total element thickness or actual thickness of that ply depending on $\pm$ value of DIRECT. See Remarks 5. and 6.. (Real > 0.0)
THETAi	Orientation angle of the ply in the plane of the plies. Measured relative to the projection z-axis defined by CORDM on the plane defined by DIRECT. See Remark 1. (Real; Default = 0.0)

**Remarks:**

1. The PCOMPLS can only be referenced by a CHEXA entry. To view the composite results of large models using solid composite elements in PATRAN or SimXpert, the following environment variable setting is required when starting the session: DRANAS\_NAST\_MEM=2048MB.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across all PCOMP, PLCOMP, and PCOMPLS entries.
3. The MIDi entry may point to MAT1, MAT9, MATORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Implicit Structural Materials					
MAT1	MAT9	MATORT	MATHE	MATUSR	MATDIGI
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			
<MATVE> refers to the ALTERNATE format for type ORTHO.					

Heat Materials	
MAT4	MAT5





If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.  
 If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID.  
 MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, MATG and MATDIGI entries.  
 If MATDIGI is being used, the ANAL field must be set to IS.  
 MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries, between themselves, may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=SLCOMP.

For INT8=L and INT20=Q, a total of 510 layers are allowed for any one element.

For INT8=ASTN, a total of 2040 layers are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

Layer orientation			
DIRECT	Normal to Layer Plane	Layers run parallel from face (ply numbering starts here)	to face (ends here)
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and +1. The ply thickness is entered in one of two ways:
  - a. Relative thickness where the numbers entered, are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percents of thickness must sum to 1.0.
  - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plies and then figures the fractional percent of each individual ply as in method 6a.

7. In the following table, BEHi refers to the structural behavior of 3D-solid elements. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Solid continuum composite	SLCOMP	<u>L</u>	HEX	8
		ASTN*	HEX	8
		Q	HEX	20
*Only DIRECT=1 or -1 is allowed				

8. In the following table, BEHiH refers to the heat behavior of 3D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Solid continuum composite	SLCOMP	<u>L</u>	HEX	8
		Q	HEX	20

9. Integration codes in Remark 7. and 8. are:

INT CODE	Integration Type
L	Linear
ASTN	Assumed STRaiN enhanced formulation solid shell
Q	Quadratic

10. The material coordinate system CORDM may be the basic system (0 or blank) or the element coordinate system (-1) or any defined coordinate system (> 0) (Default = 0).

THETAi in conjunction with CORDM is used to define the ply orientation.

- a. For BEH = SLCOMP and INT = ASTN, the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETAi is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.
- b. For BEH = SLCOMP and INT = L or Q, the Material Coordinate System is a function of how NLMOPTS, INLAM, CPROJ is set.

If CPROJ is ON then the projection scheme (see Figure 9-121) is activated. With the activation of projection scheme the X axis of the CORDM system is projected onto the layer plane to form the local X axis. The normal direction of the element (thickness direction = local Z axis) is crossed with the local X axis to yield the local Y axis. THETA<sub>i</sub> is then measured positive counter-clockwise about the local Z axis. If the X axis of the CORDM is in the same direction as the thickness direction, then the analysis will stop with an error.

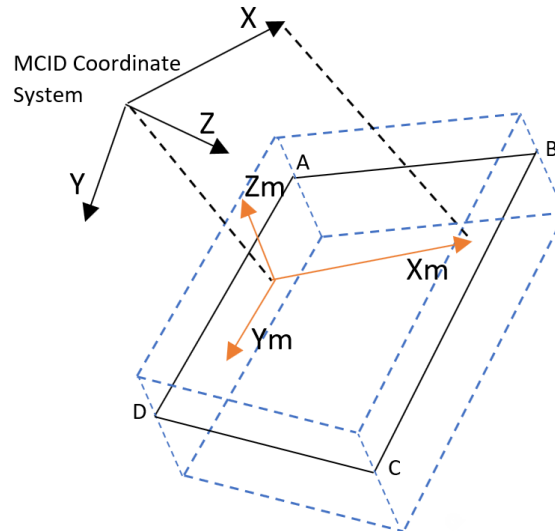


Figure 9-121 Projection Scheme

In Figure 9-121, plane ABCD is the layer plane of interest.  $Z_m$  is normal to the layer plane (or is the thickness direction).  $X_m$  is the projection of the X axis of the MCID coordinate system on the layer plane.  $Z_m \times X_m$  gives the  $Y_m$ . Angle theta is measured counter-clockwise from  $X_m$  axis about the  $Z_m$  axis.

If CPROJ is OFF (default), THETA<sub>i</sub> is then measured positive counter-clockwise about the local Z axis. The local Z is perpendicular to layer plane defined by the layup direction via DIRECT describer.

11. DIRECT, SB, ANAL, “C8” and “C20” are not supported in SOL 600 will be ignored if entered.
12. For SOL 600, PCOMPLS is made into PSOLID and PCOMP internally. Existing PSOLID and PCOMP entries with the same PID may be overwritten. The combination PSOLID and PCOMP is preferred over PCOMPLS for SOL 600, either option (but not both) may be used.
13. PCOMPLS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PCOMPLS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS, PLCOMP, PCOMPG entries.

## PCONEAX Conical Shell Element Property

Defines the properties of a conical shell element described on a CCONEAX entry.

### Format:

1	2	3	4	5	6	7	8	9	10
PCONEAX	ID	MID1	T1	MID2	I	MID3	T2	NSM	
	Z1	Z2	PHI1	PHI2	PHI3	PHI4	PHI5	PHI6	
	PHI7	PHI8	PHI9	PHI10	PHI11	PHI12	PHI13	PHI14	

### Example:

PCONEAX	2	4	1.0	6	16.3	8	2.1	0.5	
	0.001	-0.002	23.6	42.9					

### Describer

### Meaning

ID	Property identification number. (Unique Integer > 0)
MID <sub>i</sub>	Material identification number for membrane, bending, and transverse shear. (Integer ≥ 0)
T1	Membrane thickness. (Real > 0.0 if MID1 = 0)
T2	Transverse shear thickness. (Real > 0.0 if MID3 = 0)
I	Moment of inertia per unit width. (Real)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances from the middle surface for stress recovery. (Real)
PHI <sub>i</sub>	Azimuthal coordinates (in degrees) for stress recovery. (Real)

### Remarks:

1. PCONEAX is allowed only if an AXIC entry is also present.
2. PCONEAX entries may reference MAT1 or MAT2 material entries. However, only orthotropic material properties are consistent with axisymmetry. Therefore, G13 and G23 values on the MAT2 entry referenced by MID1 or MID2 and the G12 value on the MAT2 entry referenced by MID3 should be set to 0.0. In addition, the MID3 entry, if it references a MAT2 material matrix, should be of size 2 x 2.
3. If either MID1 = 0 or blank or T1 = 0.0 or blank, then both must be zero or blank.
4. If either MID2 = 0 or blank or I = 0.0 or blank, then both must be zero or blank.
5. If either MID3 = 0 or blank or T2 = 0.0 or blank, then both must be zero or blank.
6. A maximum of 14 azimuthal coordinates (PHI<sub>i</sub>) for stress recovery may be specified.
7. For a discussion of the conical shell problem, see Section 5.3.3 of the *MSC Nastran Reference Guide*.

8. The following elastic relationships are assumed:

- In-plane forces per unit width

$$\{F\} = T1[G_1]\{\varepsilon\}$$

where  $\{\varepsilon\}$  is the vector of strains in the middle surface.

- Bending moments per unit width

$$\{M\} = I[G_2]\{\chi\}$$

where  $\{\chi\}$  is the vector of curvatures.

- Transverse shear forces per unit width

$$\{V\} = T2[G_3]\{\gamma\}$$

where  $\{\gamma\}$  is the vector of transverse shear strains.

$[G_1]$ ,  $[G_2]$  and  $[G_3]$  are the stress-strain matrices defined by MID1, MID2, and MID3, respectively.

## PCONV Convection Property Definition

Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF	FTYPE	TID			
	CHLEN	GIDIN	CE	E1	E2	E3			

### Examples:

PCONV	53	2	0	.25					
PCONV	4				1	101			
PCONV	38	21			2	54			
	2.0	235	0	1.0	0.0	0.0			

### Alternate Format and Examples:

1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF	"3"	H1	H2	H3	
	H4	H5	H6	H7	H8				
PCONV	20				3	10.0			
PCONV	7				3	10.32	10.05	10.09	
	10.37								

Descriptor	Meaning
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. See Remarks 2. (Integer > 0)
FORM	Type of formula used for free convection. (Integer 0, 1, 10, 11, 20, or 21; Default = 0)
EXPF	Free convection exponent as implemented within the context of the particular form that is chosen. See Remark 3. (Real ≥ 0.0; Default = 0.0)
FTYPE	Formula type for various configurations of free convection. See Remarks 2. and 5. (Integer ≥ 0; Default = 0)
TID	Identification number of a TABLEHT entry that specifies the two-variable tabular function of the free convection heat transfer coefficient or Nusselt number. See Remark 5. (Integer ≥ 0 or blank)
CHLEN	Characteristic length. See Remarks 6. and 8. (Real > 0.0 or blank)

Describer	Meaning
GIDIN	Grid ID of the referenced inlet point. See Remarks 7. and 8. (Integer > 0 or blank)
CE	Coordinate system for defining the direction of boundary-layer flow. See Remarks 7. and 8. (Integer ≥ 0; Default = 0)
Ei	Component of the vector for defining the direction of boundary-layer flow in coordinate system CE. See Remarks 7. and 8. (Real or blank)
Hi	Free convection heat transfer coefficient. See Remark 5. (Real for H1 and Real or blank for H2 through H8; Default for H2 through H8 is H1)

### Remarks:

- Every surface to which free convection is to be applied must reference a PCONV entry. PCONV is referenced on the CONV Bulk Data entry.
- MID is used to supply the convection heat transfer coefficient (H) for FTYPE=0, or the thermal conductivity (K) for FTYPE=2. MID is ignored for FTYPE=1 and FTYPE=3 and may be blank.
- EXPF is the free convection temperature exponent.

- If FORM = 0, 10, or 20, EXPF is an exponent of (T - TAMB), where the convective heat transfer is represented as

$$q = H \cdot u_{\text{CNTRLND}} \cdot (T - \text{TAMB})^{\text{EXPF}} \cdot (T - \text{TAMB}).$$

- If FORM = 1, 11, or 21,

$$q = H \cdot u_{\text{CNTRLND}} \cdot (T^{\text{EXPF}} - \text{TAMB}^{\text{EXPF}})$$

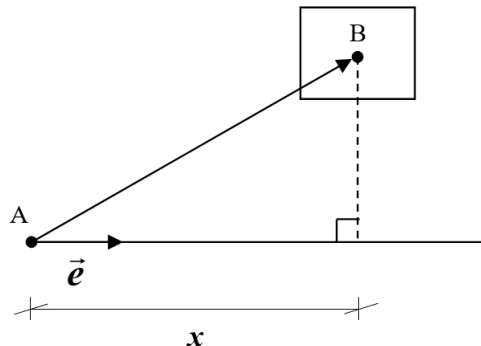
where T represents the elemental grid point temperatures and TAMB is the associated ambient temperature.

- FORM specifies the formula type and the reference temperature location used in calculating the convection film coefficient if FLMND = 0.
  - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperatures (average).
  - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
  - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperatures).
- FTYPE defines the formula type used in computing the convection heat transfer coefficient *h*.
  - If FTYPE = 0, *h* is specified in the MAT4 Bulk Data entry referenced by MID.
  - If FTYPE = 1, *h* is computed from  $h = f(T_w, T_a)$ , where *f* is a two-variable tabular function specified in the TABLEHT Bulk Data entry referenced by TID,  $T_w$  is the wall temperature, and  $T_a$  is the ambient temperature.
  - If FTYPE = 2, the program computes *h* from Nusselt number  $Nu = f(T_w, T_a)$ , where *f* is a two-variable tabular function specified in the TABLEHT Bulk Data entry referred by TID,  $T_w$  is the wall temperature, and  $T_a$  is the ambient temperature.

- a. For average heat transfer coefficient  $\bar{h}$ , specify the tabular data of the average Nusselt number  $\overline{Nu}_L = \bar{h} L / K$  and CHLEN for the characteristic length L (See Remarks 6. and 8.).
  - b. For local spatial dependent heat transfer coefficient  $h_x$ , define the tabular function of local Nusselt number  $Nu_x = h_x x / K$  along with GIDIN, CE, and Ei to calculate the distance x from the leading edge of heat transfer. (See Remarks 7. and 8.)
- If FTTYPE = 3, h is the free convection heat transfer coefficient applied to grid point Gi of the referenced HBDY surface element.
6. CHLEN specifies the characteristic length used to compute the average heat transfer coefficient  $\bar{h}$ . The following table lists typical values of CHLEN for various convection configurations.

Convection Configuration	Characteristic Length CHLEN
Free convection on a vertical plate or cylinder	Height of the plate or cylinder
Free convection from horizontal tubes	Diameter of the pipes
Free convection from horizontal square plates	Length of a side
Free convection from horizontal rectangular plates	Average length of four sides
Free convection from horizontal circular disks	0.9d, where d is the diameter of the disk.
Free convection from horizontal unsymmetric plates	A/P, where A is the surface area and P is the perimeter of the surface.

7. GIDIN, CE and Ei are used to define the distance from the leading edge of heat transfer. GIDIN specifies the referenced grid ID where heat transfer starts. CE and Ei define the direction of boundary-layer flow. If CE field is blank, the default is CE=0 for basic coordinate system. If E1, E2, and E3 fields are blank, the defaults are Ei = < 1.0, 0.0, 0.0 >, i.e. the flow is in the x direction. The distance x from the leading edge of heat transfer is computed as follows.





$$x = \overrightarrow{AB} \bullet \vec{e}$$

Where A is the location of GIDIN, B is the centroid of the convection element, and  $\vec{e}$  is the unit vector in the direction of boundary-layer flow.

8. CHLEN, GIDIN, CE, and Ei are required only for free convection from flat plates with FTYPE = 2. In this case, if the heat transfer coefficient is spatial dependent, GIDIN must be specified. Otherwise, CHLEN has to be defined for the computation of average heat transfer coefficient  $\bar{h}$ . For free convection from tubes (CHBDYP elements with TYPE="ELCY", "TUBE" or "FTUBE"), CHLEN, GIDIN, CE, and Ei need not be specified, because MSC Nastran will use the average diameter of tubes as the characteristic length while computing Nu. CHLEN, GIDIN, CE, and Ei are ignored for FTYPE ≠ 2 .
9. For RC network solver in thermal analysis, the FORM, EXPF, FTYPE, TID, CHLEN, GIDIN, CE, E1, E2 and E3 are ignored.

## PCONVM Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
PCONVM	PCONID	MID	FORM	FLAG	COEF	EXPR	EXPPI	EXPPO	

### Example:

PCONVM	3	2	1	1	.023	0.80	0.40	0.30	
--------	---	---	---	---	------	------	------	------	--

Describer	Meaning
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)
FLAG	Flag for mass flow convection. (Integer = 0 or 1; Default = 0)
COEF	Constant coefficient used for forced convection. (Real > 0.0)
EXPR	Reynolds number convection exponent. (Real ≥ 0.0; Default = 0.0)
EXPPI	Prandtl number convection exponent for heat transfer into the working fluid. (Real ≥ 0.0; Default = 0.0)
EXPPO	Prandtl number convection exponent for heat transfer out of the working fluid. (Real ≥ 0.0; Default = 0.0)

### Remarks:

- Every surface to which forced convection is applied must reference a PCONVM entry. PCONVM is referenced on the CONVM entry.
- MID specifies material properties of the working fluid at the temperature of the point FLMND. FLMND is specified on the CONVM entry.
- The material properties are used in conjunction with the average diameter and mass flow rate (mdot). MID references the material properties and supplies the fluid conductivity (k), heat capacity (cp), and viscosity (μ) needed to compute the Reynolds (Re) and Prandtl (Pr) numbers as follows:

$$Re = 4 \cdot |\text{mdot}| / (\pi \cdot \text{diameter} \cdot \mu)$$

$$Pr = cp \cdot \mu / k$$

- FORM controls the type of formula used in determination of the forced convection film coefficient h. There are two cases:

- If FORM = 0, 10, or 20 than  $h = \text{coef} \cdot Re^{EXPR} \cdot Pr^{EXPPI}$ .

- If FORM = 1, 11, or 21 then the above h is multiplied by k and divided by the average hydraulic diameter.
  - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.
  - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).
  - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
  - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).
5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.
  6. FLAG controls the convective heat transfer into the downstream point (the second point as identified on the CHBDYi statement is downstream if mdot is positive).
    - FLAG = 0, no convective flow (stationary fluid).
    - FLAG = 1, convective energy flow that is consistent with the Streamwise Upwind Petrov Galerkin (SUPG) element formulation.
  7. No phase change or internal heat generation capabilities exist for this element.
  8. For RC network solver in thermal analysis, the FORM, FLAG, COEF, EXPR, EXPPI and EXRPO are ignored.

## PCONV1

## Thermal Convection Calculation Properties

Defines the properties required to calculate convective heat transfer. Can exist in a simple mode with convection coefficient defined in the MID or in advanced mode where the H value is calculated using the geometric parameters and referenced material.

### Format:

1	2	3	4	5	6	7	8	9	10
PCONV1	PID		CorrID	MID	Mdot	Velocity	Length or Diameter	Flow Cross Section	
			Length function type	Flow Cross Sec type	Mdot f	Velocity f	Length or Diameter f	Flow Cross Sec f	
	C1	C2	C3	C4	C5	C6	C7	C8	
	C9	C10	C11	C12	C13	C14	C15	C16	
	C17	C18	C19	C20	C21	C22	C23	C24	

### Example:

PCONV1	2		701	2			1.0		
--------	---	--	-----	---	--	--	-----	--	--

### Describer

### Meaning

PID	Property identification number, referenced by CHBDYP and/or CONV. (Integer > 0)
Corr ID	ID of convection type in correlation library (Integer ≥ 0; Default 0)
MID	Material identification number for convection calculation (Integer > 0)
Mdot	Mass flow rate (Real; Default 0.0)
Velocity	Fluid velocity (Real ≥ 0.0; Default 0.0)
Length or Diameter	Geometry, based on Corr ID (Real ≥ 0.0; Default 0.0)
Flow Cross Section	Flow Cross Section (Real ≥ 0.0; Default 0.0)
Length function type	Geometry function type, 1=time 2=temperature (Integer > 0; Default 2)
Flow Cross Sec type	Flow Cross Section function type, 1=time 2=temperature (Integer > 0; Default 2)
Mdot f	Mass flow rate TABLEDj table ID (Integer ≥ 0; Default 0)
Velocity f	Fluid velocity TABLEDj table ID (Integer ≥ 0; Default 0)
Length or Diameter f	Geometric function TABLEMj or TABLEDj table ID (Integer ≥ 0; Default 0)
Flow Cross Sec f	Flow Cross Section function TABLEMj or TABLEDj table ID (Integer ≥ 0; Default 0)
C1 thru C24	Geometric properties based on chosen convection correlation (Real; Default 0.0)

**Remarks:**

1. This entry is for RC Network solver only.
2. The PCONV1 entry contains the properties for a CONV and CHDBYP, and can be used for connecting with a PRJCON. PID must be unique to both the PCONVID in PCONV and the PID in PHBDY. This will be the ID referenced by CONV, PRJCON, and CHBDYP.
3. MATID must reference a MAT4 fluid material.
4. For Corr. ID and C1 thru C24, please reference *MSC SINDA User's Guide and Library Reference* or *P/Thermal User's Guide*.

## PDAMP Scalar Damper Property

Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.

### Format:

1	2	3	4	5	6	7	8	9	10
PDAMP	PID1	B1	PID2	B2	PID3	B3	PID4	B4	

### Example:

PDAMP	14	2.3	2	6.1					
-------	----	-----	---	-----	--	--	--	--	--

Describer	Meaning
PID <sub>i</sub>	Property identification number. (Integer > 0)
B <sub>i</sub>	Force per unit velocity. (Real)

### Remarks:

1. Damping values are defined directly on the CDAMP2 and CDAMP4 entries, and therefore do not require a PDAMP entry.
2. A structural viscous damper, CVISC, may also be used for geometric grid points.
3. Up to four damping properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
5. PDAMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PDAMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PDAMP entries.

## PDAMP5 Scalar Damper Property for CDAMP5

Defines the damping multiplier and references the material properties for damping. CDAMP5 is intended for heat transfer analysis only.

### Format:

1	2	3	4	5	6	7	8	9	10
PDAMP5	PID	MID	B						

### Example:

PDAMP5	2	3	4.0						
--------	---	---	-----	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT4 or MAT5 entry. (Integer > 0)
B	Damping multiplier. (Real > 0.0)

### Remarks:

1. B is the mass that multiplies the heat capacity CP on the MAT4 or MAT5 entry.
2. RC network solver does not support PDAMP5 for thermal analysis.

**PDAMPT**

## Frequency-Dependent Damper Property

Defines the frequency-dependent properties for a PDAMP Bulk Data entry.

**Format:**

1	2	3	4	5	6	7	8	9	10
PDAMPT	PID1	TBID1							

**Example:**

PDAMPT	12	34							
--------	----	----	--	--	--	--	--	--	--

**Describer****Meaning**

PID

Property identification number that matches the identification number on a PDAMP entry. (Integer > 0)

TBID1

Identification number of a TABLEDi entry that defines the damping force per-unit velocity versus frequency relationship. (Integer  $\geq$  0; Default = 0)

**Remarks:**

1. PDAMPT may only be referenced by CDAMP1 or CDAMP3 elements in the residual structure, which do not attach to any omitted degrees-of-freedom.
2. The PDAMPT entry is ignored in all solution sequences except frequency response analysis.



## PDISTB Element property distributions.

Defines element distributions of property data.

### Format:

1	2	3	4	5	6	7	8	9	10
PDISTB	ID	TYPE	PID	PNAME	LOCNUM	SMULT	DELTA		
	EID1	V1	EID2	V2	EID3	V3	EID4	V4	
	EID5	V5	EID6	V6	EID7	V7	EID8	V8	
	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-			

### Example:

PDISTB	7008	PCOMP G	700	GE		.9			
	205	.013	224	.015	362	.135			
PDISTB	70014	PCOMP G	700	T	1				
	205	.013	224	.015	362	.135			
PDISTB	70034	PCOMP G	700	T	3				
	205	.013	224	.015	362	.135	400	.067	

### Descriptor      Meaning

ID	Unique identification number. (Integer > 0, no default)
TYPE	Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL, PSOLID and PTUBE (Character-Required).
PID	Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default)
PNAME	Property name, such as "T", "I12" (Character). See Remark 8.
LOCNUM	The Numerical order of Ply in PCOMP or PCOMPG. The Station number on a PBEAM or PBEAM3. (Integer ≥ 0).
SMULT	Scale factor of distribution entries. (Real, Default=1.0)
DELTA	Delta value of distribution entries. (Real, Default=0.0)
EID <sub>i</sub>	Element ID associated with Property PID. (Integer > 0, no default)
V <sub>i</sub>	The value of item to be changed. (Real for not PSOLID, Integer ≥ -2 for PSOLID)

### Remarks:

1. The ID of the PDISTB is for user convenience of grouping identification and is not used for case control selection.

2. The PID must point to an existing “Property” entry as shown in the Table of Remark 8. entry below. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
3. If no distribution is given, the value associated with the named property entry will be used.
4. If an EID occurs on this entry for an element not associated with referred to property the EID is ignored.
5. If an EID is referred to twice on this entry or another PDISTB entry with the same PID and PNAME, Or in the case of a PCOMP or PCOMPG with same PID, PNAME, and LOCNUM referral a fatal message will be issued.
6.  $VALUE_i = SMULT * V_i + DELTA$
7. The PID referred to may not have an associated PCOMPFQ entries. A fatal message will be issued if there is such an association.
8. For most Primary property entries, the QRG symbol is used for valid PNAME. See values in table below. For more difficult entries such as a PCOMP the PNAME for example of the thickness of PLY 3 would be entered as PNAME=T and LOCNUM=3. Note on PCOMPG it is the numerical PLY listed not the GPLYID. Thus for example if the second ply listed on a PCOMPG entry had GPLYID=4 and the thickness is to be changed then PNAME=T and LOCNUM=2 not 4.

Property	PNAME	Corresponding QRG Description
<b>Current Supported Items are:</b>		
PSHELL	T	T - Membrane thickness
	I12	12I/T**3 Bending moment of inertia ratio
	TS	TS/T Transverse shear thickness ratio
	NSM	Nonstructural mass.
	Z1, Z2	Fiber distances for stress calculations
PCOMP (G)	Z0	Distance from the reference plane to the bottom surface
	NSM	Nonstructural mass.
	SB	Allowable shear stress of the bonding material
	TREF	Reference temperature
	GE	Damping coefficient
	T	Thicknesses of the various plies (Ply Identified by LOCNUM)
	THETA	Orientation angle of the various ply. (Ply Identified by LOCNUM)
PBEAM	LOCNUM=1 corresponds to END A; LOCNUM=12 corresponds to END B; LOCNUM=2-11 Intermediate stations left to right.	
	A	Area of bar cross section (Identified by LOCNUM)

Property	PNAME	Corresponding QRG Description
<b>Current Supported Items are:</b>		
	I1, I2, I12	Area moments of inertia (Identified by LOCNUM)
	J	Torsional constant (Identified by LOCNUM)
	NSM	Nonstructural mass. (Identified by LOCNUM)
	K1, K2	Shear stiffness factor K in $K^*A^*G$ for plane 1 and plane 2.
PROD	A	Area of bar cross section
	J	Torsional constant
	NSM	Nonstructural mass.
PBAR	A	Area of bar cross section
	I1, I2, I12	Area moments of inertia
	J	Torsional constant
	NSM	Nonstructural mass.
	K1, K2	Area factor for shear
PTUBE	OD	Outside diameter of tube
	T	Thickness of tube
	NSM	Nonstructural mass.
	OD2	Diameter of tube at second grid point
PSHEAR	T	Thickness of shear panel.
	NSM	Nonstructural mass.
	F1, F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4.
PBEAM3	LOCNUM=1 corresponds to END A; LOCNUM=2 corresponds to END B; LOCNUM=3 "middle" station C	
	A	Area of the beam cross-section (Identified by LOCNUM)
	IZ, IY, IYZ	Area moments of inertia (Identified by LOCNUM)
	J	Torsional constant (Identified by LOCNUM)
	NSM	Nonstructural mass. (Identified by LOCNUM)
	KY, KZ	Shear effectiveness factors for local y- and z-directions.
	CW	Warping coefficient (Identified by LOCNUM).
PSOLID	CORDM	Identification number of the material coordinate system. Leave SMULT and DELTA fields blank.

## PDISTBM Element property material distributions.

Defines element distributions of property data.

### Format:

1	2	3	4	5	6	7	8	9	10
PDISTBM	ID	TYPE	PID	MIDNUM					
	EID1	MIDV1	EID2	MIDV2	EID3	MIDV3	EID4	MIDV4	
	EID5	MIDV5	EID6	MIDV6	EID7	MIDV7	EID8	MIDV8	
	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-	-etc.-			

### Example:

PDISTBM	70013	PSHELL	700	2					
	205	16							

### Descriptor Meaning

ID	Unique identification number. (Integer > 0, no default)
TYPE	Name of a property entry, PBAR, PBEAM, PBEAM3, PCOMP, PCOMPG, PROD, PSHEAR, PSHELL, PSOLID, and PTUBE (Character-Required).
PID	Property identification number that matches the identification number of an existing Property entry. (Integer > 0, no default)
MIDNUM	MID <sub>i</sub> name (Integer ≥ 0, Default=1) See Remark 6.
EID <sub>i</sub>	Element ID associated with Property PID. (Integer > 0, no default)
MIDV <sub>i</sub>	The replacement value of the MID <sub>i</sub> . (Integer>0, no default)

### Remarks:

1. The PID must point to an existing “PSHELL”, “PCOMPG”, “PCOMP”, and “PSOLID”, entry. If this entry is used, Nastran will check that all elements have a unique element ID across all element types. If a non-unique element ID is found a fatal message will be issued.
2. If no distribution is given, the value associated with the named property entry will be used.
3. If an EID occurs on this entry for an element not associated with referred to property entry the EID is ignored.
4. If an EID is referred to twice on this entry or another PDISTBM entry with the same PID and MIDNUM referral a fatal message will be issued.
5. The MIDV<sub>i</sub> the user supplies must be a valid MAT<sub>i</sub> entry unique across all existing MAT<sub>i</sub> entries. The MAT<sub>i</sub> may have associated MAT<sub>i</sub>F entries, however, TABLED5 (Frequency as a function of temperature) entry is not currently supported with spatial entries.

6. The MIDNUM is property dependent. For example a PSHELL has MID1 through MID4, so a MIDNUM=3 means that the MID3 field of the PSHELL is being changed. For PCOMP or PCOMPG the MIDNUM refers to the MIDi of the Numerical order of ply.

## PDUMi Dummy Element Property

Defines the properties of a dummy element ( $3 \leq i \leq 7$ ). Referenced by the CDUMi entry.

### Format:

1	2	3	4	5	6	7	8	9	10
PDUMi	PID	MID	A1	A2	A3	A4	A5	A6	
	A7	-etc.-							

### Example:

PDUM3	108	2	2.4	9.6	1.E4	15.		3.5	
	5		2						

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
Aj	Additional fields. (Real or Integer)

### Remark:

1. The additional fields are defined in the user-written element subroutines.
2. The fields on this entry are required to be defined on the corresponding ADUMi entry. This entry requires a license for "USER MODIFIABLE Nastran" Other than the PID field, all field checking is the responsibility of the user supplied code.

## PELAS Scalar Elastic Property

Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PELAS	PID1	K1	GE1	S1	PID2	K2	GE2	S2	

### Example:

PELAS	7	4.29	0.06	7.92	27	2.17	0.0032		
-------	---	------	------	------	----	------	--------	--	--

Describer	Meaning
PIDi	Property identification number. (Integer > 0)
Ki	Elastic property value. (Real)
GEi	Damping coefficient, $g_e$ . See Remarks 5. and 6. (Real)
Si	Stress coefficient. (Real)

### Remarks:

1. Be careful using negative spring values.
2. Spring values are defined directly on the CELAS2 and CELAS4 entries, and therefore do not require a PELAS entry.
3. One or two elastic spring properties may be defined on a single entry.
4. For a discussion of scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPI\)](#) in the *MSC Nastran Reference Guide*.
5. If PARAM,W4 is not specified, GEi is ignored in transient analysis. See [Parameters](#).
6. To obtain the damping coefficient GE, multiply the critical damping ratio  $C/C_0$  by 2.0.
7. If PELAS is used in conjunction with PELAST,  $K_i > 0$ , and the initial slope of the nonlinear force-displacement relationship defined by the PELAST should agree with  $K_i$ .
8. PELAS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PELAS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PELAS entries.

## PELAS1

Defines a spring property designated by a force-deflection curve for SOL 700. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PELAS1	PID	TID							

### Example:

PELAS1	22	33							
--------	----	----	--	--	--	--	--	--	--

### Describer

### Meaning

PID

Property identification number. (Integer, no Default, > 0)

TID

Identification number of a TABLED1 entry which defines the force deflection curve. (Integer, no Default, > 0)

### Remarks:

1. Unlike PELAST, when PELAS1 is used, no PELAS entry is made.
2. All PELAS and PELAS1 ID's must be unique.
3. This entry may only be referenced by a CELAS1D entry.



## PELAST Frequency Dependent or Nonlinear Elastic Property

Defines the frequency dependent or nonlinear properties for a PELAS Bulk Data entry.

### Format:

1	2	3	4	5	6	7	8	9	10
PELAST	PID	TKID	TGEID	TKNID					

### Example:

PELAST	44	38							
--------	----	----	--	--	--	--	--	--	--

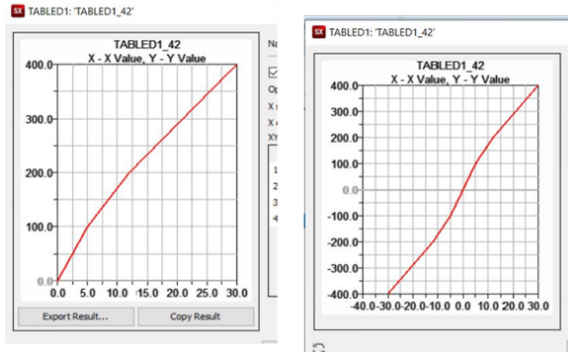
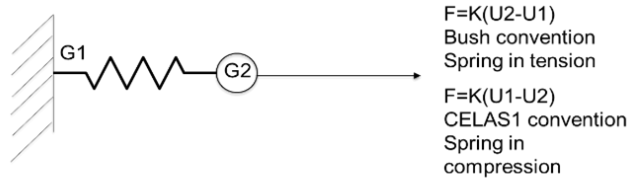
Describer	Meaning
PID	Property identification number that matches the identification number on a PELAS entry. (Integer > 0)
TKID	Identification number of a TABLEDi entry that defines the force per unit displacement vs. frequency relationship. (Integer > 0; Default = 0)
TGEID	Identification number of a TABLEDi entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer > 0; Default = 0)
TKNID	Identification number of a TABLEDi entry that defines the nonlinear force vs. displacement relationship. (Integer > 0; Default = 0)

### Remarks:

1. The PELAST entry may only be referenced by CELAS1 or CELAS3 elements in the residual structure which do not attach to any omitted degrees-of-freedom.
2. For frequency dependent modal frequency response the modes are computed using the nominal Ki values as specified on the PELAS entry.
3. The nominal values are used for all analysis types except frequency response and nonlinear analyses. For frequency dependent modal frequency response the system modes are computed using the nominal Ki values. The frequency-dependent values are used at every excitation frequency. For nonlinear analysis the nominal values for Ki should agree with the initial slope of the nonlinear force-displacement relationship defined by the PELAST, or the results will be unpredictable.
4. The following table summarizes the usage PELAST entry in various solution sequences

Field	Frequency Response	NONLINEAR (See Remark 6)	Linear (Non-Frequency Response)
TKID	Used	Ignored	Ignored
TGEID	Used	Ignored	Ignored
TKNID	Ignored	Used	Ignored

5. PELAST is used in SOL 108 and SOL 111 when TKID or TGEID is specified. It is used in SOL 106, SOL 129 and SOL 400 when TKIND is specified. It is ignored in all other solution sequences.
6. This entry is not available in SOL 600 and if entered will cause the job to terminate. If PARAM,MRPELAST,1 is entered this entry will be ignored in SOL 600.
7. The CELAS1/3 elements linear force computation  $F=K(U1-U2)$ , conflicts with the below figure, in that an obvious tension loading produces a compressive force.



The PELAST, TKNID option, for this loading, internally uses the  $F=K(U2-U1)$  when computing the element tangent matrix. This allows for tension only loading, the use of a single sided load-deflection curve to compute the element stiffness.

For element material nonlinear force and stress output, the default PELAST, TKNID option, the sign convention used is  $F=K(U2-U1)$ .

For a user desiring that the material nonlinear force and strain output, be consistent with the linear CELAS1/3 elements, the Nastran system cell NASTRAN DELTAU=1 is provided.

The tangent stiffness is still computed using  $F=K(U2-U1)$ , but for nonlinear force and stress output the convention is  $F= - K(U2-U1)$ .

While the PELAST, TKNID default option is more consistent with the figure above, in general for  $U1>U2$ , a compressive force will result. When tension or compression may occur, then the load deflection curves must be two sided curves as shown in the above figure.

## PERMEAB

## Air Bag Permeability

Defines the permeability of a COUPLE and/or GBAG (sub)surface.

Permeability is the velocity of gasflow through a (sub)surface and is defined as a linear or tabular function of the pressure difference over the surface. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PERMEAB	PID	PERMC	PERMT	FLOW	PENV	RHOENV	SIEENV	CP	

### Example:

PERMEAB	201	0.5		OUT	1.E5	1.128	2.21E5	1001.	
---------	-----	-----	--	-----	------	-------	--------	-------	--

Describer	Meaning
PID	Unique identification number of a PERMEAB entry. (Integer > 0, Required)
PERMC	Permeability is a linear function of the pressure difference. $\text{permeability} = \text{PERM} - C * \text{abs}(\text{Pinside} - \text{PENV})$ For Pinside > PENV: outflow For Pinside < PENV: inflow See Remark 3. (Real ≥ 0)
PERMT	Permeability is a tabular function of the pressure difference: table contains: permeability versus  Pinside – PENV  For Pinside > PENV: outflow For Pinside < PENV: inflow See Remark 3. (Integer ≥ 0)
FLOW	Defines the allowed directions of the flow. (Character, Default = BOTH) BOTH      In- and outflow are allowed. IN          Only inflow allowed. OUT        Only outflow allowed.
PENV	Environmental pressure. (Real > 0, Required)
RHOENV	Environmental density. (Real > 0, Required)
SIEENV	Environmental specific internal energy. (Real > 0, Required)
CP	Environmental specific heat at constant pressure. See Remark 5. (Real > 0)

### Remarks:

1. The PERMEAB entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver or Roe solver using an EOSGAM (ideal gas) equation of state.

3. Either PERM-C or PERM-T must be specified.
4. The values for the environment  $p_{env}$  (PENV),  $\rho_{env}$  (RHOENV),  $e_{env}$  (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The  $\gamma_{env}$  is calculated and is used when inflow occurs. Inflow occurs when  $p_{env} > p_{inside}$ .

5. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.

## PERMGBG

## Air Bag Permeability

Defines a permeable area of a COUPLE and/or GBAG surface, connected to another GBAG.

The velocity of the gas flow through the surface is defined as a linear or tabular function of the pressure difference. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PERMGBG	PID	PERMC	PERMT	FLOW	GBID				

### Example:

PERMGBG	12		10		2				
---------	----	--	----	--	---	--	--	--	--

Describer	Meaning						
PID	Unique identification number of a PERMEAB entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or from a LEAKAGE to model the flow between an Eulerian air bag and a GBAG. (Integer > 0; Required)						
PERMC	Permeability is a linear function of the pressure difference. permeability = PERM – C*abs (Pinside – Pgbid) The gas flow is from the higher to the lower pressure. See Remark 3. (Real > 0)						
PERMT	Permeability is a tabular function of the pressure difference: table contains: permeability versus  Pinside – Pgbid  The gas flow is from the higher to the lower pressure. See Remark 3. (Integer > 0)						
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH)  <table border="0"> <tr> <td>BOTH</td> <td>In- and outflow are allowed.</td> </tr> <tr> <td>IN</td> <td>Only inflow allowed into the GBAG or the coupling surface that references this entry.</td> </tr> <tr> <td>OUT</td> <td>Only outflow allowed into the GBAG or the coupling surface that references this entry.</td> </tr> </table>	BOTH	In- and outflow are allowed.	IN	Only inflow allowed into the GBAG or the coupling surface that references this entry.	OUT	Only outflow allowed into the GBAG or the coupling surface that references this entry.
BOTH	In- and outflow are allowed.						
IN	Only inflow allowed into the GBAG or the coupling surface that references this entry.						
OUT	Only outflow allowed into the GBAG or the coupling surface that references this entry.						
GBID	Number of a GBAG entry. This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)						

### Remarks:

1. The PERMGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler, the entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.

3. Either PERMC or PERMT must be specified.

## PEULER

## Eulerian Element Properties

Defines the properties of Eulerian element. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PEULER	PID	MID	TYPE						

### Example:

PEULER	100	25							
--------	-----	----	--	--	--	--	--	--	--

### Describer

### Meaning

PID	Unique property number. (Integer > 0, Required)
MID	Number of a MATDEUL entry defining the constitutive model. (Integer ≥ 0, Required)
TYPE	The type of Eulerian material being used. (Character, Default = HYDRO)
HYDRO	Hydrodynamic material with no shear strength + void.
1STORDER	Single material, 1 <sup>st</sup> order accurate Riemann solution-based fluids- & gases Euler solver.
2NDORDER	Single material, 2 <sup>nd</sup> order accurate Riemann solution-based fluids- & gases Euler solver.
STRENGTH	Structural material with shear strength + void.
MMHYDRO	Multimaterial hydrodynamic material with no shear strength + void.
MMSTREN	Structural multimaterial with shear strength + void.

### Remarks:

1. Make the property number unique with respect to all other property numbers.
2. The elements that reference this property use the Eulerian formulation.
3. If TYPE is set to HYDRO, only one material number for all the Eulerian elements of TYPE is used and a hydrodynamic yield model is chosen.
4. If the TYPE is set to either 1st Order or 2nd Order, only one material for all Eulerian elements of TYPE is used and the Riemann solution-based solver is chosen.
5. If TYPE is set to STRENGTH, only one material number for all the Eulerian elements of TYPE is used and a nonhydrodynamic yield model is chosen.
6. If TYPE is set to MMHYDRO, different material numbers for all Eulerian elements of TYPE are used and a hydrodynamic behavior is chosen for each material.
7. If TYPE is set to MMSTREN, different material numbers for all Eulerian elements of TYPE are used and a yield model is chosen for each material.

8. In a multimaterial Euler calculation, the options MMSTREN and MMHYDRO cannot be mixed; they are mutually exclusive.
9. If the material number is blank or zero, the corresponding elements are void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1st Order or 2nd Order scheme, an error message will be issued and the analysis will stop.
10. Initial conditions are defined on the TICEL Bulk Data entry.



## PEULER1

## Eulerian Element Properties

Eulerian element properties. The initial conditions of these elements are defined in geometric regions. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PEULER1	PID		TYPE	SID	SID2				

### Example:

PEULER1	100		HYDRO	300					
---------	-----	--	-------	-----	--	--	--	--	--

### Describer

### Meaning

PID	Unique property number. (Integer > 0; Required)
TYPE	They type of Eulerian material(s) being used. (Character; Default = HYDRO)
HYDRO	Hydrodynamic material + void.
1STORDER	Single material, 1 <sup>st</sup> order accurate Riemann solution-based fluids- & gases solver.
2NDORDE R	Single material, 2 <sup>nd</sup> order accurate Riemann solution-based fluids- & gases solver.
STRENGTH	Structural material with shear strength + void.
MMHYDRO	Multimaterial hydrodynamic material with no shear strength + void.
MMSTREN	Structural multimaterial with shear strength + void.
SID	Number of a TICEUL1 entry specifying the materials and geometric grouping criteria. (Integer > 0; Required)
SID2	Number of a EULFOR1 entry defining acceleration field. (Integer>=0 Default=0)

### Remarks:

1. Remarks 1 through 6 of the PEULER definition apply also here.
2. Initial conditions and/or material assignments are defined on the TICEUL1 Bulk Data entry.

## PFAST

## CFAST Fastener Property

Defines the CFAST fastener property values.

### Format:

1	2	3	4	5	6	7	8	9	10
PFAST	PID	D	MCID	MFLAG	KT1	KT2	KT3	KR1	
	KR2	KR3	MASS	GE	ALPHA	TREF	COINL		

### Example:

PFAST	7	1.1	70		100000.	46000.	12300.		
-------	---	-----	----	--	---------	--------	--------	--	--

### Describer

### Meaning

PID	Property identification number. (Integer > 0)
D	Diameter of the fastener. See Remark 2. (Real > 0)
MCID	Specifies the element stiffness coordinate system. See Remark 1. (Integer ≥ -1 or blank; Default = -1)
MFLAG	Defines if the coordinate system defined by MCID is absolute or relative. See Remark 1. (Integer 0 or 1; Default = 0) If MFLAG = 0, MCID defines a relative coordinate system. See Remark 1a. If MFLAG = 1, MCID defines an absolute coordinate system. See Remark 1c.
KT <sub>i</sub>	Stiffness values in directions 1 through 3. (Real)
KR <sub>i</sub>	Rotational stiffness values in directions 1 through 3. (Real; Default = 0.0)
MASS	Lumped mass of fastener. (Real; Default = 0.0)
GE	Structural damping. (Real; Default = 0.0)
ALPHA	Thermal expansion coefficient for the CFAST. (Real; Default=0.0)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0, See Remark 8.)
COINL	Length of a CFAST with coincident grids. (Real; Default=0.0, COINL ≥ 0.0)

### Remarks:

1.
  - a. If  $MCID \geq 0$  and  $MFLAG = 0$  (Default), then the  $KT_1$  stiffness will be applied along the  $x_{elem}$  axis direction of the fastener defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\|\vec{x}_B - \vec{x}_A\|}$$

The T2 direction defined by MCID will be used to define the orientation vector  $\vec{v}$  of the fastener. Then the element  $z_{elem}$  axis will be defined as

$$\vec{e}_3 = \frac{\vec{e}_1 \times \vec{v}}{\|\vec{e}_1 \times \vec{v}\|}$$

The KT3 stiffness will lie along the  $z_{elem}$  axis. The element  $y_{elem}$  axis is defined as

$$\vec{e}_2 = \vec{e}_3 \times \vec{e}_1$$

The KT2 stiffness will lie along the  $y_{elem}$  axis

This option allows the user to define orthotropic material properties normal to the axis of the fastener that will “slide” with the curve of the patches.

- b. If MCID = -1, MFLAG is ignored, and the following element system is defined: the  $x_{elem}$  axis direction of the fastener defined as

$$\vec{e}_1 = \frac{\vec{x}_B - \vec{x}_A}{\|\vec{x}_B - \vec{x}_A\|}$$

Relative to the basic system, find the smallest component  $j$  of the element  $x_{elem}$  axis unit vector. If two such components are equal, take the first one. Form a unit vector in the basic system. For example, assuming the  $j = 3$  component of  $\vec{e}_1$  was the smallest.

$$b_j = b_3 = \begin{Bmatrix} 0 \\ 0 \\ 1 \end{Bmatrix}$$

Form the following orthogonal vector:

$$\hat{e}_2 = \vec{b}_j - \frac{\vec{e}_1 \cdot \vec{b}_j}{\vec{e}_1 \cdot \vec{e}_1} \vec{e}_1$$

$$\vec{e}_2 = \frac{\hat{e}_2}{\|\hat{e}_2\|}$$

Form  $\vec{e}_3$  as

$$\vec{e}_3 = \vec{e}_1 \times \vec{e}_2$$

- c. If MCID  $\geq 0$  and MFLAG = 1, then the MCID will be used to compute stiffness. KT1 will be applied along the MCID T1 axis, KT2 along the MCID T2 axis, and KT3 along the MCID T3 axis. The element forces will be computed in the coordinate system defined in Remark 1b.

- d. If the length of GA - GB is zero, then the element  $x_{elem}$  axis is defined to lie along the projected normal to patch A.
2. The diameter D is used along with the piercing points of GA and GB to determine the location of fictitious grid points to form a fictitious hexa volume that determines the elements and physical grids used for the fastener element. Four points are positioned at  $\pm a$  positions parallel to the element axis where  $a = f(D)$ . The stiffness contribution of the fastener depends on both the stiffness values specified and the diameter D. It is a function of D, because the  $\pm a$  positions are used along with the surface shape functions of the fictitious hexa to weight the contribution of the physical grids used to the grids GA and GB of the fastener element.
3. The CFAST element (see Figure 9-122), for stiffness and structural damping calculations, is designed to satisfy rigid body equilibrium requirements. When  $\vec{x}_B - \vec{x}_A$  has finite length, internal rigid links connect grids GA and GB. This may result in coupling between translational and rotational degrees-of-freedom even when no rotational stiffness (KR1-KR3) are specified.  
For mass calculations, half the specified mass value is placed directly onto the projected grid A and grid B translational degrees-of-freedom.

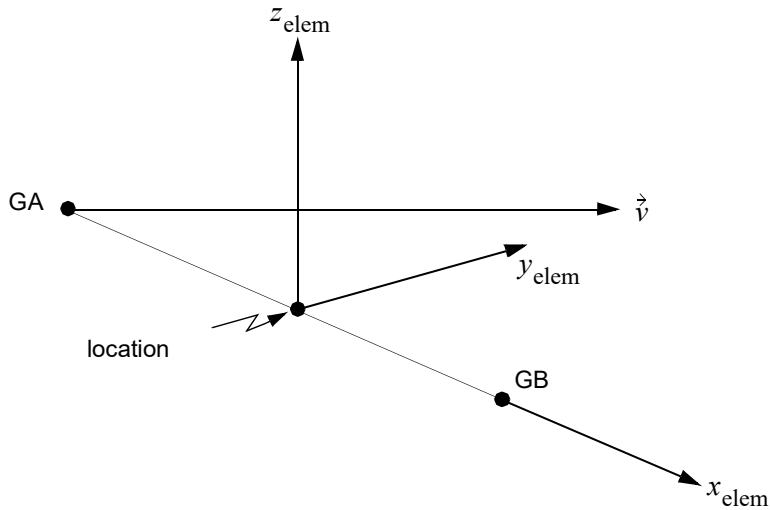


Figure 9-122 CFAST Element

4. The CFAST element lies midway between GA and GB.
5. Values for  $KTi$  and  $KRi$  are specified at the user's discretion. Assuming a short stubby beam where shear is dominate, possible values might be:

$$KT1 = \frac{EA}{L}$$

$$KT2 = \frac{G_2 A_s}{L}$$

$$KT3 = \frac{G_3 A_s}{L}$$

$$KR1 = \frac{GJ}{L}$$

$$KR2 = \frac{EI}{L} + \frac{G_2 A_s L}{3}$$

$$KR3 = \frac{EI}{L} + \frac{G_3 A_s L}{3}$$

where:

$$A = \pi D^2 / 4$$

$$I = \pi D^4 / 64$$

$$J = \pi D^4 / 32$$

$$L = |\vec{x}_B - \vec{x}_A|$$

$$A_s = A_s = A / \alpha_s$$

$$\alpha_s = 4/3$$

$E$ ,  $G_2$ ,  $G_3$ , and  $G$  are the material properties of the fastener.

The fastener stiffness is not, however, independent of the surrounding structure. The values of stiffness specified should not overwhelm the stiffness of the local structure or max ratio's will occur.

One possible way to estimate the local stiffness  $S$  is by the relationship.

$$S = \frac{t_p E_p E}{E_p + E}$$

where  $t_p$  is a shell thickness and  $E_p$  is the modulus of the shell.

6. The element force and strain are computed as follows:

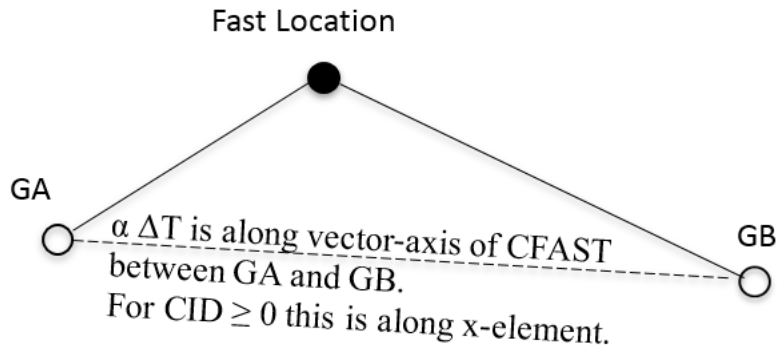
$$\{f_e\} = [K_e] \{u_e\} \text{ for statics}$$

$$\{f_e\} = ([K_e] + i(g + g_e)[K_e])(\{u_e\}_{\text{real}} + i\{u_e\}_{\text{imag}}) \text{ for frequency}$$

$$\{f_e\} = [K_e] \{u_e\} + \left( \frac{g}{w3} + \frac{g_e}{w4} \right) [K_e] \{v_e\} \text{ for transient}$$

where  $[K_e]$  is the 6 x 6 element stiffness matrix,  $\{u_e\} = \{u_b\} - \{u_a\}$  relative displacement in the element coordinate system, and  $\{v_e\} = \{v_b\} - \{v_a\}$  relative velocity in the element coordinate system. The subscripts  $a$  and  $b$  stand for end A and end B of the fastener.  $g$  is defined by param,g;  $w3$  is defined by param,w3;  $w4$  is defined by param,w4; and  $g_e$  is the GE entry of the PFAST.  $\{u_e\}$  is the strain output. Stress output is the same as force output.

7. PFAST is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PFAST property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PFAST entries.
8. The thermal expansion for the CFAST will be calculated if the user supplies the thermal expansion coefficient ALPHA and TEMPERATURE(LOAD) is requested. See TEMPERATURE Case Control command for thermal loading rules. For a CFAST with coincident grids, if COINL is provided, the thermal expansion is always just along the element x-axis.



## PFASTT

### Frequency Dependent and material nonlinear CFAST Property

Defines the frequency dependent and material nonlinear properties for a PFAST Bulk Data entry.

**Format:**

1	2	3	4	5	6	7	8	9	10
PFASTT	PID	“K”	TKID1	TKID2	TKID3	TKID4	TKID5	TKID6	
		“GE”	TGEID						
		“KN”	TKNID1	TKNID2	TKNID3	TKNID4	TKNID5	TKNID6	
			FDC	FUSE	DIR	OPTION	LOWER	UPPER	
			FSR	LRGR					

**Examples:**

PFASTT	33	“GE”	158						
--------	----	------	-----	--	--	--	--	--	--

Describer	Meaning						
PID	Property identification number that matches the identification number on PFAST entry. (Integer > 0)						
“K”	Flag indicating that the next 1 to 6 fields are stiffness frequency table identification numbers. (Character)						
TKID <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the stiffness vs. frequency relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)						
“GE”	Flag indicating that the next field is a structural damping frequency table identification number. (Character)						
TGEID	Identification number of a TABLED <sub>i</sub> entry that defines the nondimensional structural damping coefficient vs. frequency relationship. (Integer ≥ 0; Default = 0)						
“KN”	Flag indicating that the next 1 to 6 fields are nonlinear force-deflection table identification numbers. (Character)						
TKIND <sub>i</sub>	Identification number of a TABLED <sub>i</sub> entry that defines the force vs. deflection relationship in directions 1 through 6. (Integer ≥ 0; Default = 0)						
FDC	Force deflection curve rule. Specifies dependence between displacement components. See PBUSHT FDC entry of detail of rules.						
FUSE	<table style="width: 100%; border: none;"> <tr> <td style="width: 10%; text-align: center; vertical-align: top;">0</td> <td style="padding-left: 20px;">The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)</td> </tr> <tr> <td style="text-align: center; vertical-align: top;">1</td> <td style="padding-left: 20px;">The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for post processing.</td> </tr> <tr> <td style="text-align: center; vertical-align: top;">2</td> <td style="padding-left: 20px;">The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from post processing.</td> </tr> </table>	0	The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)	1	The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for post processing.	2	The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from post processing.
0	The element remains active irrespective of failure level. (Integer 0, 1, or 2, Default = 0)						
1	The element is deactivated if maximum failure as specified in OPTION is reached. Elements remains for post processing.						
2	The element is deactivated if maximum failure as specified in OPTION is reached. Element is removed from post processing.						



Describer	Meaning
DIR	The fuse direction. ( $0 < \text{Integer} < 6$ ; Default = 0). 0 or 123456 imply any direction may fuse. Values of 1 through 6 may be placed in the field with no embedded blanks. See Remark 13. of PBUSHT.
OPTION	Specifies a failure mode. (Character - Default="RELDIS") "ULTLD" The specified failure load in compression or tension will be used to define failure using load computed from the appropriate radial dependence of displacement relationship. "RELDIS" The specified max relative + displacement will be used to define failure. "RULTLD" The specified failure load in compression or tension will be used to define failure using the actual current physical load in each spring.
LOWER	Lower failure bound. (Real; Default=0.0) If OPTION = "ULTLD" then LOWER specifies a lower failure load If OPTION = "RELDIS" then LOWER specifies a minimum relative displacement before failure.
UPPER	Upper failure bound. (Real; Default=0.0) If OPTION = "ULTLD" then UPPER specifies an upper failure load. If OPTION = "RELDIS" then UPPER specifies a maximum relative displacement before failure.
FSR	Fuse Stiffness Retention Factor is a factor which scales the stiffness so that the stiffness does not instantly drop to a zero value. (Real > 0.0; Default = 1.-5)
LRGR	Specifies if large rotation is to occur at end A. (Integer > 0; Default = 0). Remark 14. of the PBUSHT.

**Remarks:**

1. For frequency dependent modal frequency response the modes are computed using the nominal  $K_i$  values as specified on the PFAST entry.
2. The nominal values are used for all analysis types except frequency response. For frequency dependent modal frequency response the system modes are computed using the nominal  $K_i$  values. The frequency-dependent values are used at every excitation frequency.
3. The PFASTT entry may only be referenced by CFAST elements in the residual structure of SOL400 which do not attach to any omitted degrees-of-freedom.
4. For nonlinear analysis the nominal values for  $K_i$  on the PFAST should agree with the initial slope of the nonlinear force-displacement relationship defined by the PFASTT, or the results will be unpredictable.



## PFTG Fatigue Properties.

Defines fatigue properties of elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PFTG	ID		FINISH	KFINISH	KF	SCALE	OFFSET		
	SHAPE	KTREAT	DIAM	T1	T2	SPTFLG			

### Examples:

PFTG	3		POLISH			1.2			
------	---	--	--------	--	--	-----	--	--	--

Describer	Meaning
ID	Unique ID referenced by a FTGDEF bulk data entry. (Integer>0)
FINISH	Material Surface Finish. This is a result of manufacturing process. Value can be NONE, POLISH, GROUND, MACHINE, POOR, ROLLED, CAST, KROUGH, KSURFC (Character; Default=NONE). See Remark 1.
KFINISH	Roughness factor for FINISH = KROUGH (0.0 < Real < 1.0, no Default). Surface roughness in microns for FINISH = KSURFC (Real > 0.0; no Default; 0.0 ≤ Real ≤ 1.0 for materials not listed in Remark 2).
KF	Fatigue strength reduction factor. (Real ≥ 0.0, Default=1.0). See Remark 3.
SCALE	Factor used to scale the resulting FE stresses of entities associated to this property set (Real; Default=1.0).
OFFSET	Offset used to offset the resulting FE stresses of entities associated to this property set (Real; Default=0.0).
SHAPE	Shape factor (Real ≥ 1.0; No default). See Remark 4.
KTREAT	Treatment factor (Real ≥ 0.0; Default = 1.0).
DIAM	Spot weld nugget diameter. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). See Remark 5. and 6.
T1/T2	Top (T1) and bottom (T2) thickness of shells connecting spot welds. Used in the fatigue analysis of spot welds only. (Real > 0.0 or blank, No Default). Both should be left blank if either one is left blank. See Remark 5.
SPTFLG	Flag to indicate that a lookup table is to be used to define the spot weld nugget diameter. 0 or 1, Default = 0, no lookup). Only used if CWELD elements are used to define spot welds. See Remark 6.

**Remarks:**

1. The KF field can be used in lieu of or in addition to the FINISH & KTREAT field to modify the fatigue limit by multiplying the original fatigue limit by this value. POOR = Poor Machined. ROLLED = As Rolled. CAST = As Cast. KROUGH and KSURFC require that a KFINISH be entered. A material CODE on the MATFTG entry must be supplied to use anything other than NONE or POLISH, otherwise an error is issued.
2. If KFINISH = KSURFC, the user should enter a value for surface roughness  $R_z$  in  $\mu\text{m}$ . This is the average surface roughness according to the German standard DIN 4768. The Surface Roughness Factor  $K_r$  will then be calculated based on the strength and type of material (for example stronger materials are in general more sensitive to surface finish, and cast materials less so). The method for calculating  $K_r$  is taken from the FKM guideline *Analytical Strength Assessment of Components in Mechanical Engineering*.

If  $R_z \leq 1\mu\text{m}$ ,  $K_r = 1$ .

Otherwise:

$$K_r = 1 - a_r \log(R_z) \log(2R_m/R_{m,N,\min})$$

$R_m$  is the UTS in MPa

$R_{m,N,\min}$  and  $a_r$  are constants.

Table 9-33 Constants for Derivation of Surface Roughness Factor  $K_r$  from Roughness  $R_z$

Material	Steel	GS*	GGG†	GT‡	GG**	Wrought Al Alloys	Cast Al alloys
CODE from MATFTG	13,15,16-99	9-12, 15	5-8	2-4	1	100-105	106
$a_r$	0.22	0.20	0.16	0.12	0.06	0.22	0.20
$R_{m,N,\min}$	400	400	400	350	100	133	133

\*GS = cast steel and heat treatable cast steel, for general purpose

†GG = cast iron with lamellar graphite (grey cast iron)

‡GGG = nodular cast iron

\*\*GT = malleable cast iron

3. Fatigue strength reduction factor can take into account notch effects, size effects, and loading type influence.  $K_f = C_{\text{notch}} \cdot C_{\text{size}} \cdot C_{\text{loading}}$  where the latter three are correction factors for each effect, respectively.
4. Setting the shape factor activates the Seeger Heuler plastic limit load correction. Leave this field blank if no plastic limit load correction is required.
5. If T1, T2, and DIAM are specifically supplied, they are used directly in the fatigue analysis of spot welds. If either T1 or T2 are blank, the thicknesses are automatically determined from the PSHELL entries that connect the spot welds.

When performing optimization using SOL 200, it is necessary to leave T1 and T2 blank if the connecting shell thicknesses are being used as design variable, otherwise the fatigue analysis of the spot welds will not use correct thicknesses from one design cycle to another.

6. If DIAM is left blank, the following rule is used:
  - The diameter is derived based on the minimum thickness of the two sheets either side of the weld by performing a lookup on a table. An example table is provided in the `spotweld.sys` file in the `msc20xx/util` directory of a standard MSC Nastran installation. This is true for element types CBAR/CBEAM and CHEXA used to define spot welds. If using CWELD elements the diameter is automatically extracted from the PWELD entry, if SPTFLG=0. If SPTFLG=1, then the lookup table is used for CWELDS. The `spotweld.sys` file is specified by including the `spotweld=<path>/spotweld.sys` on the Nastran command line or defining it in the Nastran RC file as is done with other keywords. See [Executing MSC Nastran](#) (Ch. 1). If not specified, the default table is used as given in the example file mentioned.

In the case of a table lookup to determine DIAM, the thicknesses of the top and bottom sheets must be within the range of the lookup table, which is up to 3mm by default. Otherwise a fatal error is issued. Also the diameter is set as a function of the thickness of the thinnest sheet joined by the spot weld. No interpolation between the data points in the `spotweld.sys` file is done; rather, the thickness of the thinnest sheet from each spot weld is compared to the table, and the value of thickness that is nearest to but less than or equal to the thickness of the sheet is identified. The corresponding diameter from the table is assigned to that spot weld.

Example `spotweld.sys` file:

```
SPOT WELD DEFINITION FILE
NUGGET DIAMETER=BY_THICKNESS
0.3,3.5
0.8,4.0
1.2,5.0
2.0,5.5
3.0,6.0
```

Note that there may be rounding errors in the extraction of sheet thicknesses, so when defining a `spotweld.sys` file, it may be a good idea to reduce the sheet thickness values by a small tolerance in order to avoid anomalous results

Table 9-34 Availability of Settings for Different Analysis Types

Fatigue Property	Analysis / Material Type Category			
	SN	EN	SpotWeld	SeamWeld
FINISH	✓	✓	✓	✓
KFINISH	✓	✓†		
KF	✓	✓†		
SCALE	✓	✓	✓	✓
OFFSET	✓	✓	✓	✓
SHAPE		✓		

Table 9-34 Availability of Settings for Different Analysis Types

Fatigue Property	Analysis / Material Type Category			
	SN	EN	SpotWeld	SeamWeld
KTREAT	✓	✓†		
DIAM			✓	
T1/T2			✓	
SPTFLG			✓	

✓ Available; † Not available for Multi-mean or Multi R-ratio curves

## PGAP Gap Element Property

Defines the properties of the gap element (CGAP entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PGAP	PID	U0	F0	KA	KB	KT	MU1	MU2	
	TMAX	MAR	TRMIN						

### Example:

PGAP	2	.025	2.5	1.E6		1.E6	0.25	0.25	
------	---	------	-----	------	--	------	------	------	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
U0	Initial gap opening. See <a href="#">Figure 9-124</a> . (Real; Default = 0.0)
F0	Preload. See <a href="#">Figure 9-124</a> . (Real ≥ 0.0; Default = 0.0)
KA	Axial stiffness for the closed gap; i.e., $U_a - U_b > U0$ . See <a href="#">Figure 9-124</a> . (Real > 0.0)
KB	Axial stiffness for the open gap; i.e., $U_a - U_b < U0$ . See <a href="#">Figure 9-124</a> . See <a href="#">Remark 2</a> . (Real ≥ 0.0; Default = $10^{-14} \cdot KA$ )
KT	Transverse stiffness when the gap is closed. See <a href="#">Figure 9-125</a> . It is recommended that $KT \geq (0.1 \cdot KA)$ . (Real ≥ 0.0; Default = $MU1 \cdot KA$ )
MU1	Coefficient of static friction ( $\mu_s$ ) for the adaptive gap element or coefficient of friction in the y transverse direction ( $\mu_y$ ) for the nonadaptive gap element. See <a href="#">Remark 3</a> . and <a href="#">Figure 9-125</a> . (Real ≥ 0.0; Default = 0.0)
MU2	Coefficient of kinetic friction ( $\mu_k$ ) for the adaptive gap element or coefficient of friction in the z transverse direction ( $\mu_z$ ) for the nonadaptive gap element. See <a href="#">Remark 3</a> . and <a href="#">Figure 9-125</a> . (Real ≥ 0.0 for the adaptive gap element, $MU2 \leq MU1$ ; Default = $MU1$ )
TMAX	Maximum allowable penetration used in the adjustment of penalty values. The positive value activates the penalty value adjustment. See <a href="#">Remark 4</a> . (Real; Default = 0.0)
MAR	Maximum allowable adjustment ratio for adaptive penalty values KA and KT. See <a href="#">Remark 5</a> . ( $1.0 < Real < 10^6$ ; Default = 100.0)
TRMIN	Fraction of TMAX defining the lower bound for the allowable penetration. See <a href="#">Remark 6</a> . ( $0.0 \leq Real \leq 1.0$ ; Default = 0.001)

### Remarks:

- [Figure 9-123](#), [Figure 9-124](#), and [Figure 9-125](#) show the gap element and the force-displacement curves used in the stiffness and force computations for the element.

2. For most contact problems,  $K_A$  (penalty value) should be chosen to be three orders of magnitude higher than the stiffness of the neighboring grid points. A much larger  $K_A$  value may slow convergence or cause divergence, while a much smaller  $K_A$  value may result in inaccurate results. The value is adjusted as necessary if  $TMAX > 0.0$ .
3. When the gap is open, there is no transverse stiffness. When the gap is closed and there is friction, the gap has the elastic stiffness ( $K_T$ ) in the transverse direction until the friction force is exceeded and slippage starts to occur.
4. There are two kinds of gap elements: adaptive gap and nonadaptive gap. If  $TMAX \geq 0.0$ , the adaptive gap element is selected by the program. When  $TMAX = 0.0$ , penalty values will not be adjusted, but other adaptive features will be active (i.e., the gap-induced stiffness update, gap-induced bisection, and subincremental process). The value of  $TMAX = -1.0$  selects the nonadaptive (old) gap element. The recommended allowable penetration  $TMAX$  is about 10% of the element thickness for plates or the equivalent thickness for other elements that are connected to the gap.
5. The maximum adjustment ratio  $MAR$  is used only for the adaptive gap element. Upper and lower bounds of the adjusted penalty are defined by

$$\frac{K^{init}}{MAR} \leq K \leq K^{init} \cdot MAR$$

where  $K^{init}$  is either  $K_A$  or  $K_T$ .

6.  $TRMIN$  is used only for the penalty value adjustment in the adaptive gap element. The lower bound for the allowable penetration is computed by  $TRMIN \cdot TMAX$ . The penalty values are decreased if the penetration is below the lower bound.

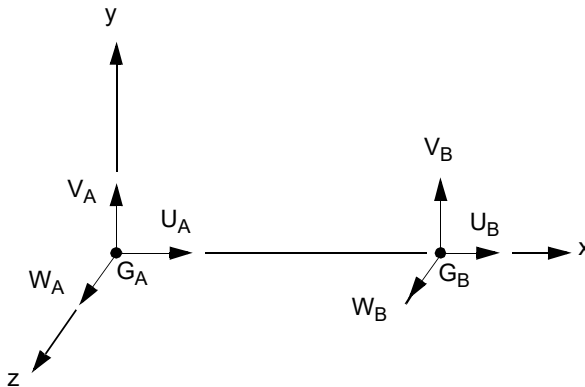


Figure 9-123 The CGAP Element Coordinate System

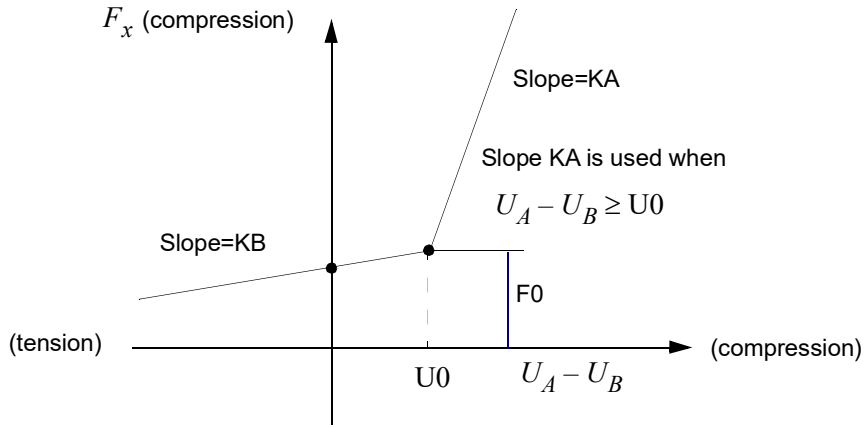


Figure 9-124 CGAP Element Force-Deflection Curve for Nonlinear Analysis

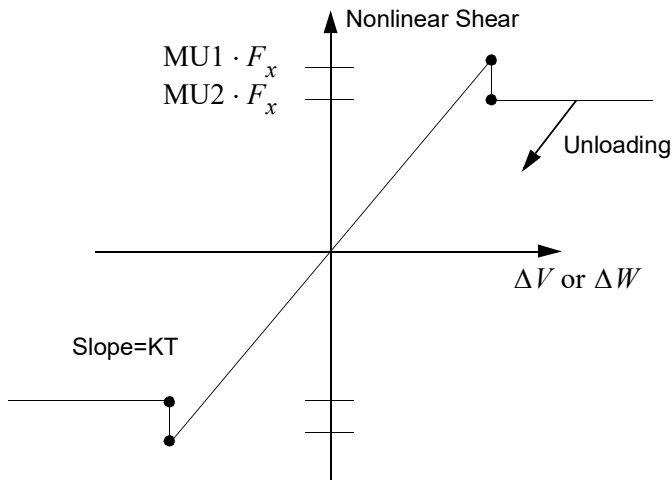


Figure 9-125 Shear Force for CGAP Element

7. If  $U_0$  is specified negative and  $G_A$  and  $G_B$  are not coincident, then the direction for closing must be controlled by the use of the CID field on the CGAP entry.
8. PGAP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PGAP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PGAP entries.

## PHBDY

## CHBDYP Geometric Element Definition

A property entry referenced by CHBDYP entries to give auxiliary geometric information for boundary condition surface elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PHBDY	PID	AF	D1	D2					

### Example:

PHBDY	2	.02	1.0	1.0					
-------	---	-----	-----	-----	--	--	--	--	--

### Describer

### Meaning

PID	Property identification number. (Unique Integer among all PHBDY entries). (Integer > 0)
AF	Area factor of the surface used only for CHBDYP element TYPE = "POINT", TYPE = "LINE", TYPE = "TUBE", or TYPE = "ELCYL". For TYPE = "TUBE", AF is the constant thickness of the hollow tube. (Real > 0.0 or blank)
D1, D2	Diameters associated with the surface. Used with CHBDYP element TYPE = "ELCYL", "TUBE", and "FTUBE". (Real > 0.0 or blank; Default for D2 = D1)

### Remarks:

- The PHBDY entry is used with CHBDYP entries.
- AF
  - For TYPE = "POINT" surfaces, AF is the area.
  - For TYPE = "LINE" or TYPE = "ELCYL" surfaces, AF is the effective width:  $\text{area} = \text{AF} \cdot (\text{length})$ .
  - For TYPE = "FTUBE" and outer TYPE = "TUBE" surfaces

$$\text{area} = \pi \cdot \left( \frac{D1 + D2}{2} \right) \cdot \sqrt{(\text{LGTH})^2 + \left( \frac{D1 - D2}{2} \right)^2}$$

- D1 and D2 are used only with TYPE = "ELCYL", TYPE = "TUBE", and TYPE = "FTUBE" surfaces.
  - For TYPE = "ELCYL" surfaces, D1 and D2 are the two diameters associated with the ellipse.
  - For TYPE = "FTUBE" and outer TYPE = "TUBE" surfaces, D1 and D2 are the diameters associated with the first and second grid points, respectively.
  - For inner TYPE = "TUBE" surfaces, the diameters are reduced by twice the thickness ( $2 \cdot \text{AF}$ ).



**PLCOMP**

## Plane Strain or Axisymmetric Composite Element Property

Defines global (external) ply IDs and properties for a composite material laminate.

**Format:**

1	2	3	4	5	6	7	8	9	10
PLCOMP	PID	DIRECT	THICKOP	SB	ANAL				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

**Example:**

PLCOMP	782	1							
	1001	171	.3	12.3					
	100	175	.7	77.7					

Describer	Meaning
PID	Element property identification number. (Integer > 0)
DIRECT	The layer direction for BEHi=COMPS or AXCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer input is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer $\pm 1$ or $\pm 2$ ; Default + 1)
THICKOP	An out-of-plane thickness. (Real, Default = 1.0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL= "ISH" - Implicit structural and heat elements are being referred to. (Character Default ISH).
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real $\geq 0.0$ )
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four mid-side grids. (Character)
BEHi	Element structural behavior. See Remarks 4. and 7. (Character default: COMPS for BEH4 and BEH8)
INTi	Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H and INT8H)

Describer	Meaning
BEHiH	Element heat behavior. See Remark 8. (Character Default: COMPS for BEH4H and BEH8H)
INTiH	Integration scheme. See Remark 9. (Character Default: L for INT4H, Q for INT8H, Q for INT8H)
IDI	Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)
MIDI	Material ID for the ply. See Remark 3. (Integer > 0)
Ti	Either fractional percent of the total element thickness or actual thickness of that ply depending on $\pm$ value of DIRECT. See Remarks 5. and 6. (Real > 0.0)
THETAi	Orientation angle of the ply in the plane of the plies. These angles are measured about the thickness direction of the element. (Real; Default = 0.0)

**Remarks:**

1. The PLSIMP can only be referenced by a CQUAD, CQUAD4, CQUAD8, or CQUADX entry. Currently it should be used ONLY in SOL400. Erroneous results will occur in other Solution sequences.
2. Global Ply ID is intended as a unique ply identifier for ply alignment across ALL PCOMP, PLSIMP, and PLSIMPLS entries.
3. The MIDI entry may point to MAT1, MAT3, MATHORT, MATHE, MATUSR or MATDIGI entries. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials					
MAT1	MAT3	MATORT	MATHE	MATUSR	MATDIGI
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			
<MATVE> refers to the ALTERNATE format for type ORTHO					

Heat Materials	
MAT4	MAT5



If heat analysis is being performed and the user wishes to override standard MSC Nastran heat elements, the ANAL entry must be set to IH or ISH.  
 If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.  
 MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

4. The keyword entries may occur, between themselves, in any order or not at all. If a keyword entry is missing, its defaults are assumed.
5. The following table describes layer orientation for BEHi=COMPS or AXCOMP. A total of 1026 plys are allowed for any one element.

Note the ply numbering starts from the bottom to the top parallel to the positive thickness direction.

Layer Orientation			
DIRECT	Normal to Layer edge	Layers run parallel from edge (ply numbering starts here)	to edge (ends here)
1	Element Y direction	G1-G2	G4-G3
2	Element X direction	G1-G4	G2-G3

6. The ply thickness of the element is computed using isoparametric coordinates of the element in the DIRECT direction and the element nodes are mapped between -1 and + 1. The ply thickness is entered in one of two ways:
  - a. Relative thickness where the numbers are a fractional percent of the total thickness. This is the preferred method. For this method, the sum of all the fractional percentages of thickness must sum to 1.0.
  - b. Absolute thickness where the layer thickness is entered directly. Using this option, the code sums the total user input thickness across all plys and then figures the fractional percent of each individual ply as in method 6a.
7. In the following table, BEHi refers to the structural behavior of 2D-solid elements. An underlined item delineates a default.

Implicit Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Strain composite	COMPS	<u>L</u>	QUAD	4
		<u>Q</u>	QUAD	8
Axisymmetric composite	AXCOMP	L	QUAD	4
		Q	QUAD	8

8. In the following table, BEHiH refers to the heat behavior of 2D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Strain composite	COMPS	<u>L</u>	QUAD	4
		<u>Q</u>	QUAD	8
Axisymmetric composite	AXCOMP	L	QUAD	4
		Q	QUAD	8

9. Integration codes in Remark 7 are:

INT CODE	Integration Type
L	Linear
Q	Quadratic

10. P<sub>L</sub>COMP is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the P<sub>L</sub>COMP property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between P<sub>S</sub>SOLID, P<sub>C</sub>COMPLS, P<sub>L</sub>COMP, P<sub>C</sub>COMPG, P<sub>S</sub>SHELL, P<sub>L</sub>PLANE entries.

## PLOAD Static Pressure Load

Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PLOAD	SID	P	G1	G2	G3	G4			

### Example:

PLOAD	1	-4.0	16	32	11				
-------	---	------	----	----	----	--	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
P	Pressure. (Real)
Gi	Grid point identification numbers. (Integer > 0; G4 may be zero or blank.)

### Remarks:

- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- The grid points define either a triangular or a quadrilateral surface to which a pressure is applied. If G4 is blank, the surface is triangular.
- In the case of a triangular surface, the assumed direction of the pressure is computed according to the right-hand rule using the sequence of grid points G1, G2, G3 illustrated in [Figure 9-126](#).

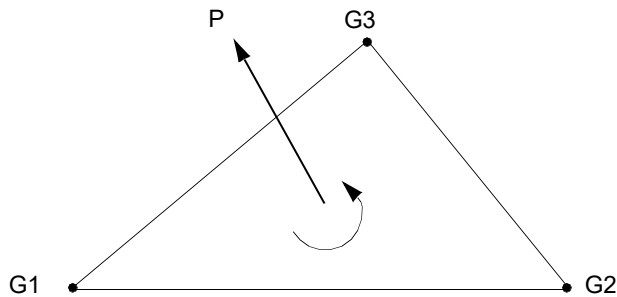


Figure 9-126 Pressure Convention for Triangular Surface of Surface Elements and/or the Faces of Solid Elements

The total load on the surface (see [Figure 9-127](#)),  $AP$ , is divided into three equal parts and applied to the grid points as concentrated loads. A minus sign in field 3 reverses the direction of the load.

4. In the case of a quadrilateral surface, the grid points  $G1$ ,  $G2$ ,  $G3$ , and  $G4$  should form a consecutive sequence around the perimeter. The right-hand rule is applied to find the assumed direction of the pressure. Four concentrated loads are applied to the grid points in approximately the same manner as for a triangular surface. The following specific procedures are adopted to accommodate irregular and/or warped surfaces:
  - The surface is divided into two sets of overlapping triangular surfaces. Each triangular surface is bounded by two of the sides and one of the diagonals of the quadrilateral.
  - One-half of the pressure is applied to each triangle, which is then treated in the manner described in [Remark 2](#).

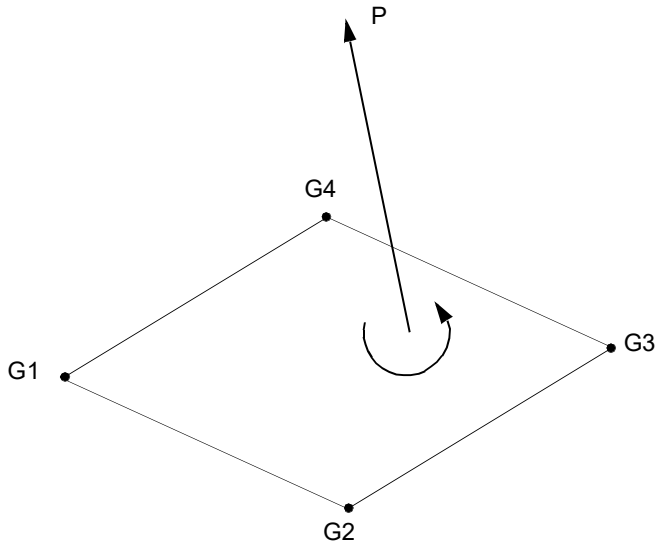


Figure 9-127 Pressure Convention for Quadrilateral Surface of Surface Elements and/or the Faces of Solid Elements

5. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 835](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600 if geometric nonlinear effects are turned on with `PARAM,LGDISP,1`. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

## PLOAD1

### Applied Load on CBAR, CBEAM or CBEND Elements

Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis. For the CBEND element, only distributed loads over an entire length may be defined.

#### Format:

1	2	3	4	5	6	7	8	9	10
PLOAD1	SID	EID	TYPE	SCALE	X1	P1	X2	P2	

#### Example:

PLOAD1	25	1065	MY	FRPR	0.2	2.5E3	0.8	3.5E3	
--------	----	------	----	------	-----	-------	-----	-------	--

#### Describer

#### Meaning

SID	Load set identification number. (Integer > 0)
EID	CBAR, CBEAM, or CBEND element identification number. (Integer > 0)
TYPE	Load type. (Character: "FX", "FY", "FZ", "FXE", "FYE", "FZE", "MX", "MY", "MZ", "MXE", "MYE", "MZE")
SCALE	Determines scale factor for X1, X2. (Character: "LE", "FR", "LEPR", "FRPR")
X1, X2	Distances along the CBAR, CBEAM, or CBEND element axis from end A. (Real; X2 may be blank; $0 \leq X1 \leq X2$ )
P1, P2	Load factors at positions X1, X2. (Real or blank)

#### Remarks:

- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- If  $X2 \neq X1$ , a linearly varying distributed load will be applied to the element between positions X1 and X2, having an intensity per unit length of bar equal to P1 at X1 and equal to P2 at X2, except as noted in Remarks 8. and 9.
- If X2 is blank or equal to X1, a concentrated load of value P1 will be applied at position X1.
- If  $P1 = P2$  and  $X2 \neq X1$ , a uniform distributed load of intensity per unit length equal to P1 will be applied between positions X1 and X2 except as noted in Remarks 8. and 9.
- Load TYPE is used as follows to define loads:  
"FX", "FY" or "FZ": Force in the x, y, or z direction of the basic coordinate system.  
"MX", "MY" or "MZ": Moment in the x, y, or z direction of the basic coordinate system.

- “FXE”, “FYE” or “FZE”: Force in the x, y, or z direction of the element’s coordinate system.  
 “MXE”, “MYE” or “MZE”: Moment in the x, y, or z direction of the element’s coordinate system.
- If SCALE = “LE” (length), the xi values are actual distances along the element axis, and, if  $X1 \neq X2$ , then  $P_i$  are load intensities per unit length of the element.
  - If SCALE = “FR” (fractional), the xi values are ratios of the distance along the axis to the total length, and (if  $X2 \neq X1$ )  $P_i$  are load intensities per unit length of the element.
  - If SCALE = “LEPR” (length projected), the xi values are actual distances along the element axis, and (if  $X2 \neq X1$ ) the distributed load is input in terms of the projected length of the element.

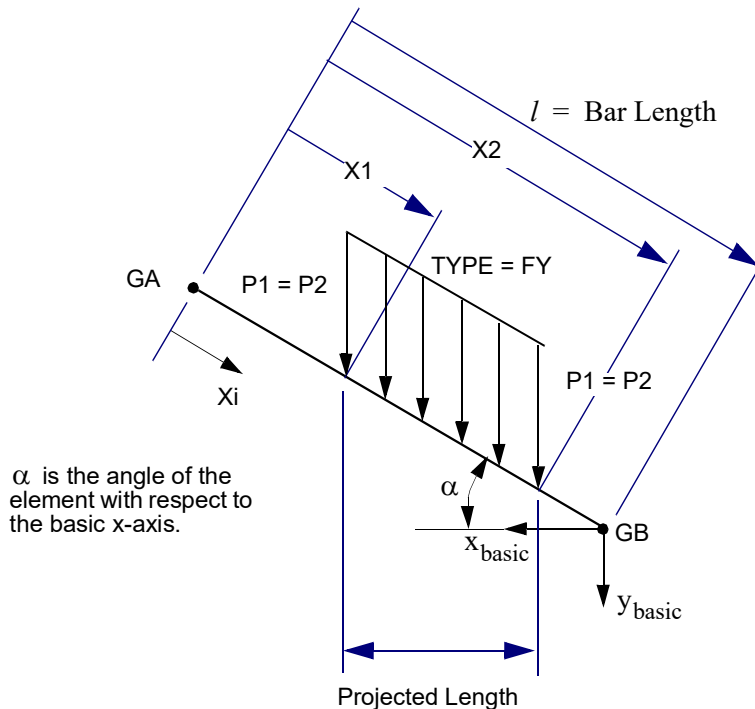


Figure 9-128 PLOAD1 Convention on Beam or Bar Elements

If SCALE = “LE”, the total load applied to the bar is  $P1 (X2 - X1)$  in the y-basic direction.

If SCALE = “LEPR”, the total load applied to the bar is  $P1 (X2 - X1) \cos\alpha$  in the y-basic direction.

- If SCALE = “FRPR” (fractional projected), the Xi values are ratios of the actual distance to the length of the bar (CBAR entry), and if  $X1 \neq X2$ , then the distributed load is specified in terms of the projected length of the bar.
- Element identification numbers for CBAR, CBEAM, and CBEND entries must be unique.
- For the CBEND element, the following coordinate equivalences must be made for the element coordinates



$$R_{elem} \equiv X_{elem}$$

$$\theta_{elem} \equiv Y_{elem}$$

12. Only distributed loads applied over the entire length of the CBEND element may be applied.
13. Projected loads are not applicable to the CBEND element.
14. Loads on CBEAM elements defined with PLOAD1 entries are applied along the line of the shear centers.
15. If a CBARAO or PLOAD1 entry is specified and stress and/or force output is requested, then the stresses and/or forces will be calculated at each location  $X_i$  and output as a separate line. The force and stress locations  $X_i=0$  and  $X_i = l$  will always be output. This output format will be used for all beam and bar elements.
16. If on the TYPE field of the PLOAD1 entry, the element coordinate system direction (e.g. TYPE = FYE) option is selected, then the projection (i.e. SCALE = FRPR or LEPR) option is ignored and the result is the same as the SCALE = FR (or LE) option.

## PLOAD2

### Uniform Normal Pressure Load on a Surface Element

Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements.

#### Format:

1	2	3	4	5	6	7	8	9	10
PLOAD2	SID	P	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	EID8	-etc.-						

#### Example:

PLOAD2	21	-3.6		4	16		2		
--------	----	------	--	---	----	--	---	--	--

#### Alternate Format and Example:

PLOAD2	SID	P	EID1	“THRU”	EID2				
PLOAD2	1	30.4	16	THRU	48				

#### Describer

#### Meaning

SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
EID <sub>i</sub>	Element identification number. (Integer ≥ 0 or blank; for the “THRU” option, EID1 < EID2.)

#### Remarks:

- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOAD<sub>i</sub> or TLOAD<sub>i</sub> entry.
- At least one positive EID must be present on each PLOAD2 entry.
- If the alternate form is used, all elements EID1 through EID2 must be two-dimensional.
- The direction of the pressure is computed according to the right-hand rule using the grid point sequence specified on the element entry. Refer to the PLOAD entry.
- All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
- The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 835](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution

sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with `PARAM,LGDISP,1`. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).

## PLOAD4

### Pressure Load on Surface and Faces of Solid Elements

Defines a pressure load on a face of a CHEXA, CPENTA, CPYRAM, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element. See Remark 17.

#### Format:

1	2	3	4	5	6	7	8	9	10
PLOAD4	SID	EID	P1	P2	P3	P4	G1	G3 or G4	
	CID	N1	N2	N3	SORL	LDIR			

#### Example:

PLOAD4	2	1106	10.0	8.0	5.0		48		
	6	0.0	1.0	0.0					

#### Alternate Format and Example (See Remark 8.):

PLOAD4	SID	EID1	P1	P2	P3	P4	“THRU”	EID2	
	CID	N1	N2	N3	SORL	LDIR			

PLOAD4	2	1106	10.0	8.0	5.0		THRU	1143	
	6	0.0	1.0	0.0					

Describer	Meaning
SID	Load set identification number. (Integer > 0)
EID EID1 EID2	Element identification number. (Integer > 0; for the “THRU” option, EID1 < EID2)
P1, P2, P3, P4	
G1	
G3	
G4	Identification number of the CTETRA grid point located at the corner; this grid point may not reside on the face being loaded. This is required data and is used for CTETRA elements only. (Integer > 0)

Describer	Meaning
CID	Coordinate system identification number. See Remark 2. (Integer $\geq 0$ ; Default = 0)
N1, N2, N3	Components of vector measured in coordinate system defined by CID. Used to define the direction (but not the magnitude) of the load intensity. See Remark 2. (Real)
SORL	The character string SURF or LINE. SURF means the surface load acting on the surface of the element and LINE means the consistent edge loads acting on the edges of the element. The default is SURE. See Remark 13.
LDIR	Denote the direction of the line load (SORL=LINE), character string X, Y, Z, TANG, or NORM. The default is NORM. See Remark 14.

### Remarks:

- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- The continuation entry is optional. If fields 2, 3, 4, and 5 of the continuation entry are blank, the load is assumed to be a pressure acting normal to the face. If these fields are not blank, the load acts in the direction defined in these fields. Note that if CID is a curvilinear coordinate system, the direction of loading may vary over the surface of the element. The load intensity is the load per unit of surface area, not the load per unit of area normal to the direction of loading.
- For the faces of solid elements, the direction of positive pressure (defaulted continuation) is inward. For triangular and quadrilateral faces, the load intensity P1 acts at grid point G1 and load intensities P2, P3, (and P4) act at the other corners in a sequence determined by applying the right-hand rule to the outward normal.
- For plate elements, the direction of positive pressure (defaulted continuation) is in the direction of positive normal, determined by applying the right-hand rule to the sequence of connected grid points. The load intensities P1, P2, P3, (and P4) act respectively at corner points G1, G2, G3, (and G4) for triangular and quadrilateral elements. (See plate connection entries.)
- If P2, P3, and P4 are blank fields, the load intensity is uniform and equal to P1. P4 has no meaning for a triangular face and may be left blank in this case.
- Equivalent grid point loads are computed by linear or bilinear interpolation of load intensity followed by numerical integration using isoparametric shape functions. Note that a uniform load intensity will not necessarily result in equal equivalent grid point loads.
- G1 and G3 are ignored for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements.
- The alternate format is available only for CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR elements. The continuation entry may be used in the alternate format.

9. For triangular faces of CPENTA elements, G1 is an identification number of a corner grid point that is on the face being loaded and the G3 or G4 field is left blank. For faces of CTETRA elements, G1 is an identification number of a corner grid point that is on the face being loaded and G4 is an identification number of the corner grid point that is not on the face being loaded. Since a CTETRA has only four corner points, this point G4 will be unique and different for each of the four faces of a CTETRA element.
10. For the CQUADR and CTRIAR element, only pressure that acts normal to the element is computed properly. Surface tractions are not resolved into moments normal to the element.
11. All referenced elements must exist (closed list) for residual only runs and are not required to exist (open list) for superelement runs; and they cannot be hyperelastic for either.
12. If fields 3 through 5 of the continuation entry are not blank, the load is assumed to have a fixed direction. If fields 2 through 5 of the continuation entry are left blank, the load is assumed to be a pressure load. In this case, follower force effects are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK, 835](#)). In addition, follower force effects are included in the force balance in the nonlinear static and nonlinear transient dynamic solution sequences, SOLs 106, 129, 153, 159, 400 and 600, if geometric nonlinear effects are turned on with `PARAM,LGDISP,1`. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153, 400 and 600) but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
13. The SORL field is ignored by all elements except QUADR and TRIAR. For QUADR or TRIAR only, if `SORL=LINE`, the consistent edge loads are defined by the PLOAD4 entry. P1, P2, P3 and P4 are load per unit length at the corner of the element. If all four Ps are given, then the line loads along all four edges of the element are defined. If any P is blank, then the line loads for only two edges are defined. For example, if P1 is blank, the line loads of the two edges connecting to G1 are zero. If two Ps are given, then the line load of the edge connecting to the two grid points is defined. If only one P is given, the second P value default to the first P value. For example, P1 denotes that the line load along edge G1 and G2 has the constant value of P1.
14. The direction of the line load (`SORL=LINE`) is defined by either (CID, N1, N2, N3) or LDIR. Fatal error will be issued if both methods are given. TANG denotes that the line load is in tangential direction of the edge, pointing from G1 to G2 if the edge is connecting G1 and G2. NORM denotes that the line load is in the mean plan, normal to the edge, and pointing outward from the element. X, Y, or Z denotes the line load is in the X, Y, or Z direction of the element coordinate system. If both (CID, N1, n2, N3) and LDIR are blank, then the default is `LDIR=NORM`.
15. For SOL 600, the SORL field may also be used by CQUAD4 and CTRIA3 in addition to CQUADR and CTRIAR.
16. For SOL 600, the LDIR field must be blank or a fatal error will occur. SOL 600 line loads must use the CID, N1, N2, N3 fields.
17. For SOL 400, with large displacement (`PARAM,LGDISP`), higher-order 6-node triangles (CTRIA6), 8-node quadrilateral (CQUAD8), 20-node hexagonal (CHEXA), or 15-node pentahedral (CPENTA), the load is not calculated correctly. Use lower-order elements if distributed loads are required in a large displacement analysis.

**PLOADB3**

## Applied distributed load on CBEAM3 elements

Defines a distributed load to a CBEAM3 element over entire length of the beam axis.

**Format:**

1	2	3	4	5	6	7	8	9	10
PLOADB3	SID	EID	CID	N1	N2	N3	TYPE	SCALE	
	P(A)	P(B)	P(C)						

**Example:**

PLOADB3	10	1002	LOCAL	1.0			MOMENT		
	100.	90.	70.						

**Describer****Meaning**

SID	Load set identification number. (Integer > 0; Required)
EID	CBEAM3 element identification number. (Integer > 0, Required)
CID	Coordinate system for load definition. (Character or Integer; Default = "BASIC") "LOCAL"            Local coordinate system; "ELEMENT"        Element coordinate system; "BASIC" or 0       Basic coordinate system;
n (n>0):	Any user-specified coordinate system identification number.
N1, N2, N3	Load vector components measured in coordinate system specified by CID. (Real; at least one $N_i \neq 0.0$ )
TYPE	Type of applied load. (Character = "FORCE", "MOMENT" or "BIMOMENT"; Required)
SCALE	Load vector scale factor. (Real; Default = 1.0)
P(j)	Magnitudes of load at j (j=A, B and C). (Real; Default = 0.0)

**Remarks:**

- In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- The load vector is defined by  $\vec{P}_j = SCALE \cdot P_j \cdot \vec{N}$ , ( $j = A, B, C$ ). The orientation of load  $\vec{P}$  is determined by vector  $\vec{N}$  and the magnitude is equal to  $SCALE \cdot P$  times magnitude of vector  $\vec{N}$ .

3. The distributed load is applied over the entire length of the beam axis, along the line of the shear center.



## PLOADX1

### Pressure Load on Axisymmetric Element

Defines surface traction to be used with CAXISYM, CQUADX, CTRIAX, and CTRIAX6 axisymmetric elements.

#### Format:

1	2	3	4	5	6	7	8	9	10
PLOADX1	SID	EID	PA	PB	GA	GB	THETA		

#### Example:

PLOADX1	200	35	3.5	10.5	10	30	20.		
---------	-----	----	-----	------	----	----	-----	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
EID	Element identification number. (Integer > 0)
PA	Surface traction at grid point GA. (Real)
PB	Surface traction at grid point GB. (Real; Default = PA)
GA, GB	Corner grid points. GA and GB are any two adjacent corner grid points of the element. (Integer > 0)
THETA	Angle between surface traction and inward normal to the line segment. (Real; Default = 0.0)

#### Remarks:

1. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. PLOADX1 is intended only for the CAXISYM, CQUADX, CTRIAX, and CTRIAX6 elements.
3. The surface traction is assumed to vary linearly along the element side between GA and GB.
4. The surface traction is input as force per unit area. MSC Nastran converts this traction into equivalent nodal loads by integration over the surface. For AXISYM and hyperelastic QUADX and TRIAX elements this integration is over one radian. For TRIAX6 and axisymmetric harmonic QUADX and TRIAX elements this integration is over two pi radians.
5. THETA is measured counter-clockwise from the inward normal of the straight line between GA and GB, to the vector of the applied load, as shown in [Figure 9-129](#), [Figure 9-130](#), and [Figure 9-131](#). Positive pressure is in the direction of inward normal to the line segment.

For the axisymmetric shell element (CAXISYM), the inward normal is established from the cross product  $\hat{k} \times (\vec{GB} - \vec{GA})$ , where  $\hat{k}$  is the unit out-of-plane vector and  $(\vec{GB} - \vec{GA})$  is the in-plane line segment from GA to GB (see Figure 9-131).

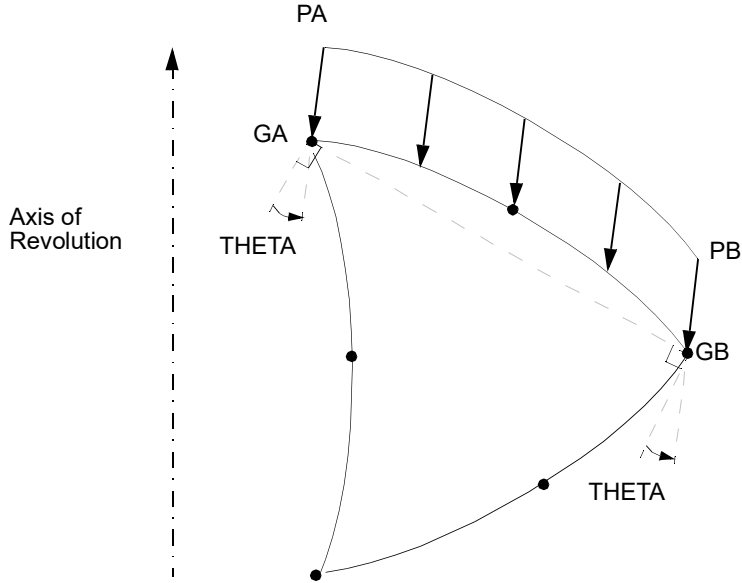


Figure 9-129 Pressure Load on CTRIAX6 or CTRIAX Element

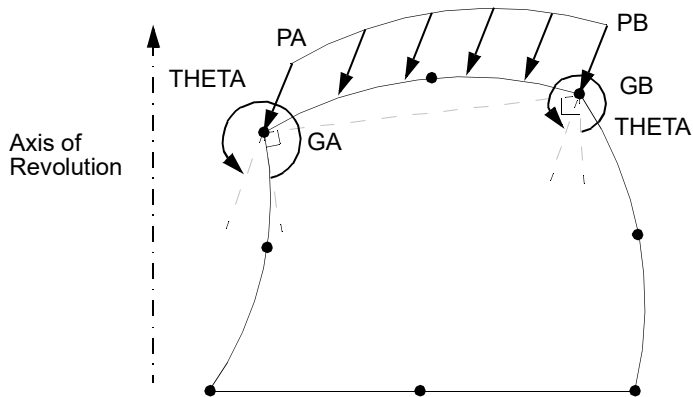


Figure 9-130 Pressure Load on CQUADX Element

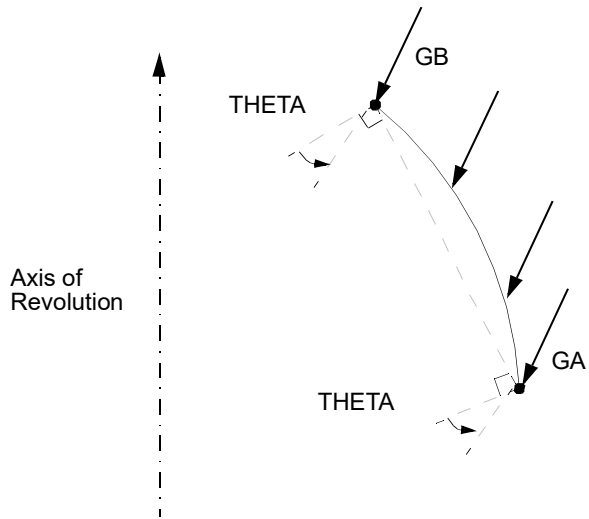


Figure 9-131 Pressure Load on CAXISYM Element

6. Axisymmetric harmonic elements may have Harmonic "N" greater than zero. For this case the "PA" and "PB" fields of the PLOADX1 entry are coefficients of  $\text{COS}(N \cdot \text{theta})$ . The PLOADX1 cannot supply azimuthal tractions on axisymmetric harmonic elements.

## PLOTEL Dummy Plot Element Definition

Defines a one-dimensional dummy element for use in plotting.

### Format:

1	2	3	4	5	6	7	8	9	10
PLOTEL	EID	G1	G2						

### Example:

PLOTEL	29	35	16						
--------	----	----	----	--	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (Integer > 0)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

### Remarks:

1. This element is not used in the model during any of the solution phases of a problem. It is used to simplify plotting of structures with large numbers of colinear grid points, where the plotting of each grid point along with the elements connecting them would result in a confusing plot.
2. Element identification numbers should be unique with respect to all other element identification numbers.
3. Only one PLOTEL element may be defined on a single entry.
4. In superelement analysis, PLOTELS, as well as other elements such as CBAR, CQUAD4, etc., will affect the formation of the superelement tree. The PLOTEL EIDs will also appear in the superelement map output; see the description of PARAM,SEMAPPRT in [Parameters](#).
5. Only grid points connected by structural elements appear on structure plots. This does not include points connected only by rigid or general elements or MPCs. A plot element in parallel with elements that do not plot will cause these points to be present.

## PLPLANE

### Fully Nonlinear Plane Element Properties

Defines the properties of a fully nonlinear (i.e., large strain and large rotation, etc.) plane strain, plane stress, or axisymmetric element. Please refer to PSHLN2.

#### Format:

1	2	3	4	5	6	7	8	9	10
PLPLANE	PID	MID	CID	STR					

#### Example:

PLPLANE	203	204	-2	GRID					
---------	-----	-----	----	------	--	--	--	--	--

#### Describer

#### Meaning

PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP entry for Nastran conventional elements, or MATHE and other linear or nonlinear material entries for Nastran advanced nonlinear elements. (Integer > 0). See Remarks 7.
CID	Identification number of a coordinate system defining the plane of deformation. See Remarks 2. and 3. (Default = 0)
STR	Location of stress and strain output. (Character: "GAUS" or "GRID", Default = "GRID")

#### Remarks:

1. PLPLANE can be referenced by a CQUAD, CQUAD4, CQUAD8, CQUADX, CTRIA3, CTRIA6, or CTRIAX entry.
2. Plane strain hyperelastic elements must lie on the x-y plane of the CID coordinate system. Stresses and strains are output in the CID coordinate system.
3. Axisymmetric hyperelastic elements must lie on the x-y plane of the basic coordinate system. CID may not be specified and stresses and strains are output in the basic coordinate system.
4. Negative values of CID will indicate an axes combination for axisymmetric elements. The following table describes the value and axes. Inside one model, only one CID value is allowed.

Axial	Radial	CID Value
X	Y	-2
X	Z	-3
Y	X	-4

Axial	Radial	CID Value
Y	Z	-5
Z	X	-6
Z	Y	-7

- CID = 0 is the default value which equal to CID = -2.
- PLPLANE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLPLANE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness among PLPLANE, PAXSYMH, and PSHELL entries.
- When using Enhanced materials (SOL400), the MID field may point to additional Material entries such as a MAT1. For a full list see the PSHLN2 entry. If a material type other than the MATHP or MATHE entry is used in solution sequences other than SOL400, wrong results will occur. Note linear stress recovery is not available for any element using the PLPLANE.

## PLSOLID Fully Nonlinear Solid Element Properties

Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.

### Format:

1	2	3	4	5	6	7	8	9	10
PLSOLID	PID	MID	STR						

### Example:

PLSOLID	20	21							
---------	----	----	--	--	--	--	--	--	--

Describer	Meaning
PID	Element property identification number. (Integer > 0)
MID	Identification number of a MATHP entry. (Integer > 0)
STR	Location of stress and strain output. (Character: "GAUS" or "GRID", Default = "GRID")

### Remarks:

1. PLSOLID can be referenced by a CHEXA, CPENTA, or CTETRA entry.
2. Stress and strain are output in the basic coordinate system.
3. PLSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PLSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PLSOLID entries.

## PLTSURF

## Surface plot mesh definition

Defines the surface plot mesh.

### Format:

1	2	3	4	5	6	7	8	9	10
PLTSURF	EID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
	GID8								

### Describer

### Meaning

EID	Identification number of element.
GIDi	Grid identification number of structure/cavity models or trim component surface element.

### Remarks:

1. This element is used only to describe the surface mesh of 3D model. The connection can be 3, 4, 6 or 8 nodes and the connection sequence mirrors those of CTRIA3, CQUAD4, CTRIA6 and CQUAD8.
2. PLTSURF can be utilized to describe surface mesh on ACPEMCP for trim component coupling in PEM job.
3. For trim component, PLTSURF ID must be referenced on SET3,id,ELEM. If SET1 entry is used with PLTSURF ID, fatal error or unexpected results may be the results.
4. PLTSURF entry can also be used to define surface mesh for structure/cavity models and it behaves similarly as PLOTTEL and does not participate in analysis.



## PMARKER

## Property Definition of a Marker Element

Defines the behavior of the marker element in the EULER domain. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PMARKER	ID	TYPE							

### Example:

PMARKER	7	FIXED							
---------	---	-------	--	--	--	--	--	--	--

### Describer

### Meaning

ID	Marker property ID, referred by CMARKB2 and CMARKN1 entries. (Integer > 0, Required)
TYPE	Behavior of the marker grid points in the Euler domain: (Character, Default = FIXED)
	FIXED                      The marker will not move in the Euler domain.
	MOVING                     The marker will be moved by velocities in the Euler domain.

### Remarks:

1. The PMARKER entry will be ignored for elements that are referring to structural grid points. These structural grid points will move with the structure and the Euler velocities do not change their velocity.
2. TYPE = FIXED. This means that the marker is stationary through out the simulation and it is therefore not moving with the Euler velocity. If the marker grid is located outside the Eulerian domain(s), the Marker will still be allowed to exist. It means, however, that no variables are recorded and that the variables will appear as zero on the Time History plots.
3. TYPE = MOVING. The marker is moving along with the Eulerian material. When the grid point approaches a coupling surface there is no mechanism that prevents the marker from passing through the coupling surface. When this happens the marker enters an element that is covered and motion of the grid point will stop. It is allowed that the grid point moves from one Euler domain to the other through a porous hole or a coupling surface with interactive failure.

## PMASS Scalar Mass Property

Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).

### Format:

1	2	3	4	5	6	7	8	9	10
PMASS	PID1	M1	PID2	M2	PID3	M3	PID4	M4	

### Example:

PMASS	7	4.29	6	13.2					
-------	---	------	---	------	--	--	--	--	--

Describer	Meaning
PIDi	Property identification number. (Integer > 0)
Mi	Value of scalar mass. (Real)

### Remarks:

1. Mass values are defined directly on the CMASS2 and CMASS4 entries, and therefore do not require a PMASS entry.
2. Up to four mass values may be defined by this entry.
3. For a discussion of scalar elements, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.
4. PMASS is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PMASS property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PMASS entries.

## PMINC

## Constant Spallation Model

Defines a spallation model where the minimum pressure is constant. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PMINC	PID	VALUE	FVTOL	FVTOL2					

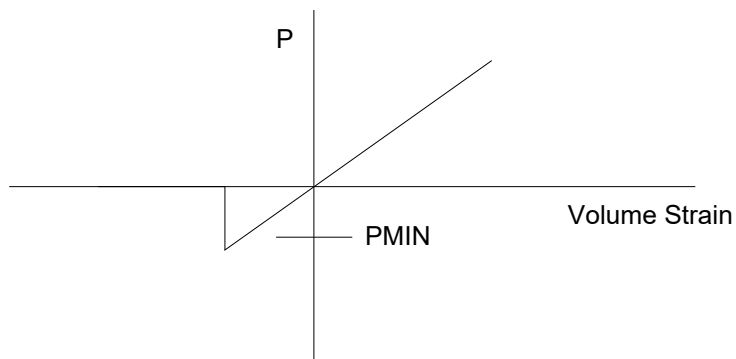
### Example:

PMINC	220	-370							
-------	-----	------	--	--	--	--	--	--	--

Describer	Meaning
PID	Unique PMINC number. Referenced from MATDEUL. (Integer > 0, Required)
VALUE	The value of the minimum pressure. (Real $\leq$ 0.0, Default = 0.0)
FVTOL	Volume cutoff tolerance. (Real > 0, 1.E-4)
FVTOL2	Maximum void fraction that is permissible under tension. (Real $\geq$ 0; Default = 0.0)

### Remarks:

1. If the pressure in an element falls below the minimum pressure, the element spalls. The pressure and yield stress are set to zero.



2. The default for the volume cutoff tolerance is 1.E-4. This value should be decreased in case of large mass increase of material without any reason.
3. If an element spalls a void is created. In order to prevent getting void fractions that are too small, a void fraction is put to zero if it is smaller than FVTOL. The default for FVTOL is 1.E-4 and works only for Eulerian elements. This value should be decreased in case of large mass increase of material without any reason. Voids can be created during transport of material, because of a material failure and by unloading.

4. With  $FVTOL2 = 0$ , any void fraction in an element will lead to failure, and then no tensile stresses are possible. In simulations in which tensile conditions are present, it can be required to allow for tensile stresses in the presence of a small void fraction not exceeding a threshold. This threshold is given by  $FVTOL2$ . A good value for  $FVTOL2 = 2 * FVTOL = e.E-4$ .  $FVTOL 2$  is only used for Eulerian materials.  $FVTOL 2$  should be larger than  $FVTOL$ .

## PMREBAI Defines Rebar Property Information for CMREBAI Elements in SOL 600

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. SOL 600 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PMREBAI	IP	TOL	IFILE	X1	X2	X3	MICRO	FACTOR	
	IM1	POS1	AREA1	SSS1	ANG1	IPOST1	IORI1		
	IM2	POS2	AREA2	SSS2	ANG2	IPOST2	IORI2		
	IM3	POS3	AREA3	SSS3	ANG3	IPOST3	IORI3		
	IM4	POS4	AREA4	SSS4	ANG4	IPOST4	IORI4		
	IM5	POS5	AREA5	SSS5	ANG5	IPOST5	IORI5		
	"HEXA"	ID1	THRU	ID2	IHE3	THRU	IHE4		
	"HEXA"	ID5	THRU	ID6	etc.				

### Example (two rebar layers through matrix CHEXA elements 100-120):

PMREBAI	20		1	1.0	0.0	0.0	1	0.01	
	100	0.25	0.075	3.0	45.0	2000	1		
	101	0.275	0.050	32.0	4 -30.0	2001	1		
	HEXA	100	THRU	120					

Describer	Meaning
IP	Rebar property ID. (Integer, Required, no Default)
TOL	Exterior tolerance. A rebar grid is considered within a matrix CHEXA element if the distance between the element and the grid is smaller than the tolerance times the average edge length of the element unless the grid is actually inside another matrix CHEXA element. (Real, Default = 0.05)
IFILE	Option to create a file for rebar verification. (Integer, Default = 0). If IFILE=1, a file named jid.marc_rebar.mfd will be created for use by Mentat.
X1	First direction cosine of reference axis. (Real, Required, no Default)
X2	Second direction cosine of reference axis. (Real, Required, no Default)
X3	Third direction cosine of reference axis. (Real, Required, no Default)
MICRO	Option to activate "micro buckling" behavior. (Integer, Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
FACTOR	Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real, Default = 0.02)

IMi	Material identification number. (Integer, Required, no Default)
POSi	Relative position of rebar layer at edge 1 (pr/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real, Required, no Default)
AREAi	Rebar cross sectional area. (Real, Required, no Default)
SSSi	Number of rebars per unit length in each layer. (Real, Required, no Default)
ANGi	Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real, Default = 0.0, must be between -90.0 and 90.0), see <a href="#">Figure 9-132</a> .
IPOSTi	Global identification number of the rebar layer used for postprocessing. (Integer, Default = 0)
IORIi	Rebar layer orientation type, see <a href="#">Figure 9-133</a> . (Integer, Required, no Default) 1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face 2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face 3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face
HEXA	Enter the string HEXA to indicate the line that contains matrix CHEXA element ID's through which the rebar passes. (Character, Required)
ID1, ID3, etc.	Starting matrix CHEXA element ID. (Integer ID1 is Required)
THRU	Enter string THRU to indicate that more than one matrix CHEXA element with these rebar properties apply.
ID2, ID4, etc.	Ending matrix CHEXA element ID. (Integer, ID2 is Required)

### Remarks:

1. This entry is the property entry for CMREBAI elements and makes use of Marc's REBAR and INSERT capabilities for rebar membrane element types 147 and 148. Entries CMREBAR and PMREBAR makes use of Marc's REBAR only capability and use rebar elements 23 and 146.
2. If rebar layers i are not desired, the lines 2-5 may be omitted. The line for i=1 is required.
3. See the following two figures for a definition of ANG and IORI.
4. Any material valid for SOL 600 may be used to define IM.
5. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords).  $\alpha$  is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the next figure.

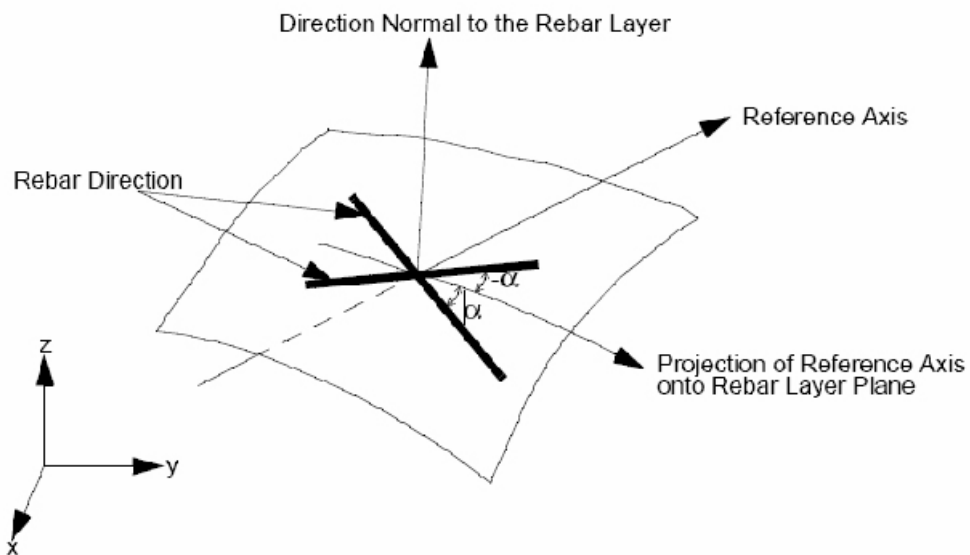


Figure 9-132 Description of rebar orientation on a single rebar layer

### 3-D

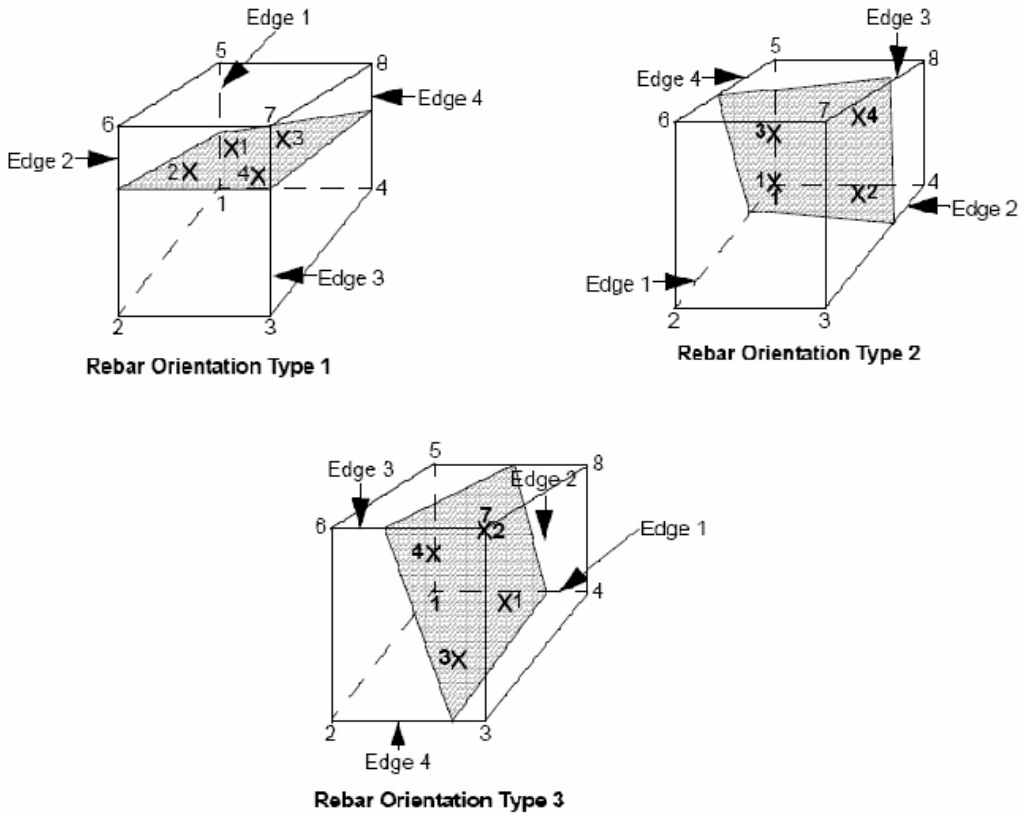


Figure 9-133 Rebar numbering and orientation type (IORI value) - only 3D supported.



**PMREBAR****Defines Rebar Property Information for CMREBAR Elements in SOL 600**

In some cases, particularly for modeling of concrete or tires, it is beneficial to add rebar or cord material to a matrix. The resulting combined material is similar to a composite but it is sometimes easier to postprocess the stresses of the rebar and matrix separately to determine failure conditions. Enter lines 2 through 6 below to describe up to 5 “rebar layers”. If only the structure only contains two rebar layers, only enter lines 1-3 as shown in the example. SOL 600 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
PMREBAR	IP		IFILE	X1	X2	X3	MICRO	FACTOR	
	IM1	POS1	AREA1	SSS1	ANG1	IPOST1	IORI1		
	IM2	POS2	AREA2	SSS2	ANG2	IPOST2	IORI2		
	IM3	POS3	AREA3	SSS3	ANG3	IPOST3	IORI3		
	IM4	POS4	AREA4	SSS4	ANG4	IPOST4	IORI4		
	IM5	POS5	AREA5	SSS5	ANG5	IPOST5	IORI5		

**Example (two rebar layers):**

PMREBAR	20		1	1.0	0.0	0.0	1	0.01	
	100	0.25	0.075	3.0	45.0	2000	1		
	101	0.275	0.050	32.0	4 -30.0	2001	1		

**Describer****Meaning**

IP	Rebar property ID. (Integer; Required; no Default)
IFILE	Option to create a file for rebar verification. (Integer; Default = 0). If IFILE=1, a file named jid.marc_rebar.mfd will be created for use by Mentat.
X1	First direction cosine of reference axis. (Real; Required; no Default)
X2	Second direction cosine of reference axis. (Real; Required; no Default)
X3	Third direction cosine of reference axis. (Real; Required; no Default)
MICRO	Option to activate “micro buckling” behavior. (Integer; Default = 0) If MICRO=0 micro buckling is not activated, if MICRO=1 micro buckling is activated.
FACTOR	Factor to use to reduce rebar stiffness in compression if MICRO=1. (Real; Default = 0.02)
IMi	Material identification number. (Integer; Required; no Default)
POSi	Relative position of rebar layer at edge 1 (p/r/t) - the ratio of the distance between the reference surface and the rebar layer to the distance across the element. (Real; Required; no Default)
AREAi	Rebar cross sectional area. (Real; Required; no Default)

SSSi	Number of rebars per unit length in each layer. (Real; Required; no Default)
ANGi	Angle (degrees) between the rebar and the projection of the reference axis on the rebar layer plane (see the following figure). (Real; Default = 0.0, must be between -90.0 and 90.0), see <a href="#">Figure 9-134</a> .
IPOSTi	Global identification number of the rebar layer used for postprocessing. (Integer; Default = 0)
IORIi	Rebar layer orientation type, see <a href="#">Figure 9-135</a> . (Integer; Required; no Default) 1 = Thickness direction is from the 1,2,3,4 face to the 5,6,7,8 face 2 = Thickness direction is from the 1,4,8,5 face to the 2,3,7,6 face 3 = Thickness direction is from the 2,1,5,6 face to the 3,4,8,7 face

**Remarks:**

1. This entry is the property entry for CMREBAR elements and makes use of Marc’s REBAR capability for element types 23 and 146. Entries CMREBAI and PMREBAI makes use of Marc’s REBAR and INSERT capabilities and use membrane rebar elements 147 and 148.
2. See [Figure 9-134](#) and [Figure 9-135](#) for a definition of ANG and IORI.
3. Any material valid for SOL 600 may be used to define IM.
4. For each rebar layer, the user is required to define cord material identification number, cross section area of the cords, density of the cords, and an angle (defining spatial orientation of the cords).  $\alpha$  is the angle between the cord and the projection of a predefined reference axis on the rebar layer plane as shown in the [Figure 9-134](#).

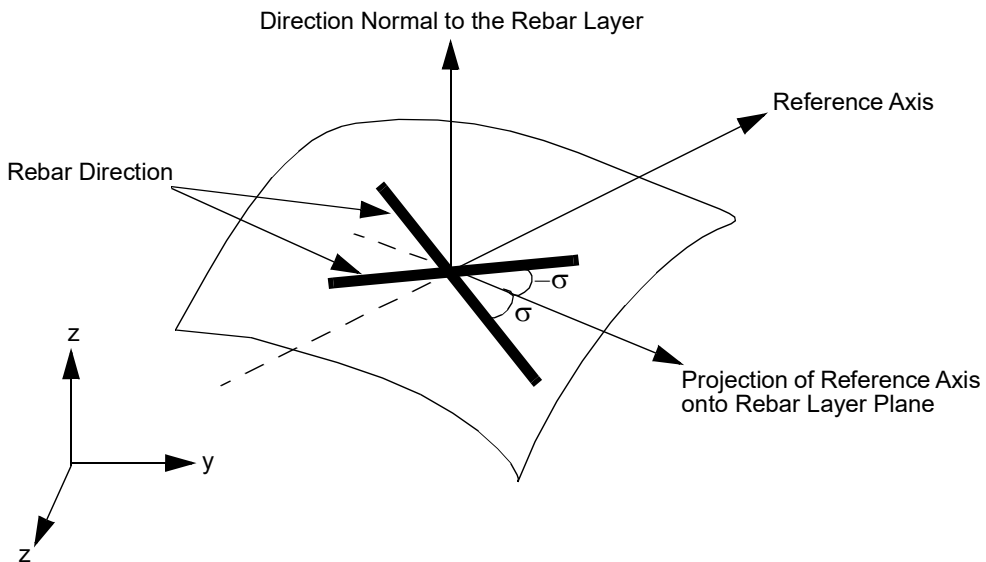


Figure 9-134 Description of rebar orientation on a single rebar layer

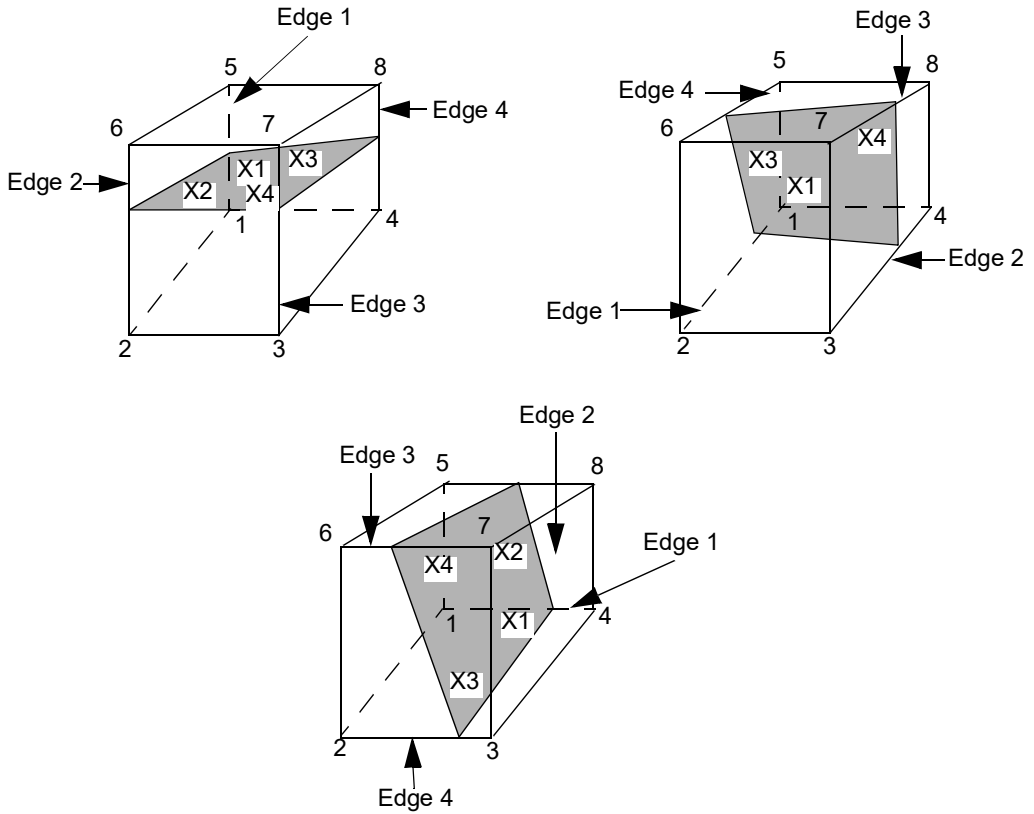


Figure 9-135 Rebar numbering and orientation type (IORI value) - only 3D supported

## POINT Geometric Point Entry

Define edge point for SELOC entries or profile PBMSECT entries.

### Format:

1	2	3	4	5	6	7	8	9	10
POINT	ID	CP	X1	X2	X3				

### Example:

POINT	12	1	1.	2.	5.				
-------	----	---	----	----	----	--	--	--	--

Describer	Meaning	Type	Default
ID	Point identification number.	Integer $\geq 0$	Required
CP	Identification number of coordinate system in which the location of point is defined.	Integer $\geq 0$	0
X1, X2, X3	Location of the point in coordinate system CP.	Real	0.0

### Remarks:

1. SELOC entries can refer to POINT entries in the residual or part superelements.
2. ID of POINTs must be unique with respect to ID of GRID entries.
3. POINT entries can be referenced on SET1/SET3 for defining arbitrary beam cross section, ABCS, via PBRSECT/PBMSECT. Note that CP and X3 must be left blank for POINT entries used for ABCS.

**POINTAX**

## Conical Shell Point

Defines the location of a point on an axisymmetric shell ring at which loads may be applied via the FORCE or MOMENT entries and at which displacements may be requested. These points are not subject to constraints via MPCAX, SPCAX, or OMITAX entries.

**Format:**

1	2	3	4	5	6	7	8	9	10
POINTAX	ID	RID	PHI						

**Example:**

POINTAX	2	3	30.0						
---------	---	---	------	--	--	--	--	--	--

Describer	Meaning
ID	Point identification number. (Unique Integer > 0)
RID	Identification number of a RINGAX entry. (Integer > 0)
PHI	Azimuthal angle in degrees. (Real)

**Remarks:**

1. This entry is allowed only if an AXIC entry is also present.
2. POINTAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

## PORFCPL

## Flow Between Two Coupling Surfaces Through a Hole

Defines an interaction between two coupling surfaces through a hole. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORFCPL	PID	SIZE		FLOW	CSID	MID			

### Example:

PORFCPL	1	SMALL		BOTH	1				
---------	---	-------	--	------	---	--	--	--	--

### Describer

### Meaning

PID	Unique PORFCPL identification number. (Integer > 0; Required)
SIZE	Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL)
	SMALL The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.
	LARGE The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH)
	BOTH In- and outflow are allowed.
	IN Only inflow allowed into the COUPLE that references this entry.
	OUT Only outflow allowed into the COUPLE that references this entry.
CSID	The ID of the COUPLE entry. This COUPLE is the one that is connected to the coupling surface that references this entry. (Integer > 0; Required)
MID	MATDEUL ID number of the transported gas. Only used when connecting to a GBAG and Euler solver uses the multi-material and SIZE=LARGE. See Remark 2. (Integer > 0)

### Remarks:

1. The PORFCPL entry can only be referenced from LEAKAGE entry.

2. For SIZE=LARGE: once gas from a GBAG flows into an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

## PORFGBG

## Flow Between Two Air Bags Through a Hole

Defines a hole in a couple and/or GBAG (sub)surface, connected to another GBAG. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORFGBG	FID	SIZE		FLOW	GBID	MID			

### Example:

PORFGBG	1	SMALL		BOTH	1				
---------	---	-------	--	------	---	--	--	--	--

### Describer

### Meaning

FID	Unique number of a PORFGBG entry. It can be referenced from either a LEAKAGE to model the flow between GBAGs, or between Eulerian air bag and a GBAG or between Eulerian air bags. (Integer > 0; Required)
SIZE	Defines the type of flow method that is used for mass leaving or entering the air bag volume. (Character; Default = SMALL)  SMALL      The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.  LARGE      The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH)  BOTH      In- and outflow are allowed.  IN      Only inflow allowed into the GBAG or the coupling surface that references this entry.  OUT      Only outflow allowed into the GBAG or the coupling surface that references this entry.
GBID	Number of a GBAG entry. This GBAG is the one that is connected to the GBAG or coupling surface that references this entry. (Integer > 0; Required)
MID	MATDEUL ID number of the GBAG gas. Only used when connecting a GBAG to an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 3. (Integer > 0)



**Remarks:**

1. The PORFGBG entry can be referenced from a LEAKAGE entry.
2. When used with Euler and SIZE=SMALL, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. For SIZE=LARGE: once gas from a GBAG enters an Eulerian domain it is treated as Eulerian material. For the single material Euler solver only one Eulerian material is present and the material number MID can be left blank. Since GBAG material is an ideal gas it is required that Eulerian material also uses an EOSGAM (ideal gas) equation of state. When using the Multi-material solver the Material number MID has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.

## PORFLOW

## Porous Flow Boundary

Defines the material properties for the in- or outflow of an Eulerian mesh through a porous area of the couple surface. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORFLOW	FID		TYPE1	VALUE1	TYPE2	VALUE2	TYPE3	VALUE3	
	TYPE4	VALUE4	-etc.-						

### Example:

PORFLOW	120		XVEL	100.0	1				

### Describer

### Meaning

FID	Unique number of a PORFLOW entry. (Integer > 0; Required)
TYPE	The properties on the flow boundary. (Character, Required)
MATERIAL	MATDEUL ID number.
XVEL	Velocity in the x-direction.
YVEL	Velocity in the y-direction.
ZVEL	Velocity in the z-direction.
PRESSURE	Pressure.
DENSITY	Density.
SIE	Specific internal energy.
FLOW	The type of flow boundary required.
METHOD	The method used for the material transport.
VALUE <sub>i</sub>	The value of the property specified in the TYPE field. (Real or Character, Required) For TYPE <sub>i</sub> set to FLOW, the value is a character entry: either IN, OUT or BOTH, indicating that the flow boundary is defined as an inflow, outflow, or possibly an in- or outflow boundary. The default is BOTH. See Remark 4. For TYPE <sub>i</sub> set to METHOD, the value is a character entry: either VELOCITY or PRESSURE, indicating that the material transport is based on the velocity method or the pressure method. The default is VELOCITY. See Remark 4.

### Remarks:

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the (SUB)SURFACE segment is intersecting.

3. The SURFACE can be only a general coupling surface (see the COUPLE entry).
4. The different methods used to calculate the material transport through a porous (sub)surface are described in General Coupling.
5. METHOD=VELOCITY is valid for all equation of state models.  
 METHOD=PRESSURE is valid for EOSGAM (ideal gas) in combination with the single material hydrodynamic Euler solver.
6. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
7. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions
8. Prescribing both pressure and velocity may lead to the instabilities.

## PORFLWT

## Time Dependent Porous Flow Boundary

Defines a time dependent flow through a porous area of the couple surface. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORFLWT	FID	TYPE							
	VELTYPE	VELOCITY	PRESTYP	PRES					
	MID	DENSTYP	DENSITY	SIETYP	SIE				

### Example:

PORFLWT	2	IN							
	TABLE	101	TABLE	102					
	91	TABLE	105	TABLE	107				

### Describer

### Meaning

FID	Unique number of a PORFLWT entry. (Integer > 0; Required)
TYPE	Type of flow boundary. (Character; Default = BOTH)
	IN                      Inflow boundary (see Remarks 2. and 3.). Only inflow is allowed. The inflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The same holds for the density and sie.
	OUT                     Only outflow is allowed. The outflow velocity and pressure can be optionally specified. If not given, the values in the adjacent Euler element will be used. The outflow boundary will always use the material mixture as present in the adjacent Euler element.
	BOTH                   Material is allowed to flow in or out. The switch between inflow and outflow is based on the direction of the velocity in the adjacent Euler element. Only pressure can be left unspecified. If not given the pressure in the adjacent Euler element will be taken.
VELTYPE	Type of velocity definition. (Character; Default = ELEMENT)
	ELEMENT            Value of Euler element.
	CONSTANT          Value is constant in time.
	TABLE                Value varies in time.
VELOCITY	Value of inflow or outflow velocity. If VELTYPE = TABLE it refers to a TABLED1. The velocity direction is normal to the coupling surface or subsurface. A positive velocity corresponds with inflow. See Remark 6. (Integer or Real)
PRESTYP	Type of pressure definition. See Remark 6. (Character)

	ELEMENT	Value of Euler element.
	CONSTANT	Value is constant in time.
	TABLE	Value varies in time.
PRES		Value of inflow or outflow pressure. If PRESTYPE = TABLE it refers to a TABLED1. (Integer or Real)
MID		MATDEUL ID of inflowing material. Input is not allowed for TYPE = OUT. When MID is specified, it is required to also define density and SIE for the inflowing material. (Integer; Default = 0)
DENSTYP		Type of density definition: (Character, Required when MID > 0.)
	ELEMENT	Value of Euler element
	CONSTANT	Value is constant in time
	TABLE	Value varies in time
DENSITY		Value of density. If DENSTYP = TABLE, it refers to a TABLED1 ID. (Integer or Real, Required when MID is given.)
SIETYPE		Type of density definition. Required when MID is given. (Integer or Real)
	ELEMENT	Value of Euler element.
	CONSTANT	Value is constant in time.
	TABLE	Value varies in time.
SIE		Value of specific internal energy. If SIETYPE = TABLE it refers to a TABLED1. Required when MID is given. (Integer or Real)

**Remarks:**

1. Reference FID by a LEAKAGE entry.
2. Any material properties not specifically defined have the same value as the element that the segment of the coupling surface is intersecting.
3. The surface can be only a general coupling surface (see the COUPLE entry).
4. Alternative methods are available to define holes and permeable sections in an air bag. See the entries: LEAKAGE, PORHOLE, PERMEAB, PORFGBG and PERMGBG.
5. In the case of material flow into a multi-material Euler mesh, the material number, the density and specific energy have to be set. On the other hand when material flows out of a multi-material Euler mesh it is assumed that each of the materials present in the outflow Euler element contributes to the out flow of mass. The materials are transported in proportion to their relative volume fractions

6. The boundary condition initiates/determines a wave in compressible material like gas and water. This can be either an outgoing or an ingoing wave. For stability it is important that the waves created are compatible with the flow type near the boundary. Relevant flow types are subsonic inflow, subsonic outflow, supersonic inflow and supersonic outflow. For example for subsonic inflow prescribing both pressure and velocity would initiate outgoing waves. Outgoing waves for an inflow boundary condition is known to be instable. However, for supersonic inflow one can specify both pressure and velocity since there are no outgoing waves at a supersonic inflow boundary.

## PORHOLE

## Holes in Air Bag Surface

Defines a hole in a COUPLE and/or GBAG surface. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORHOLE	FID	SIZE		FLOW	PENV	RHOENV	SIEENV	CP	
	MID								

### Example:

PORHOLE	301								

Describer	Meaning
PID	Unique number of a PORHOLE entry. (Integer > 0; Required)
SIZE	Defines the type of flow method that is used for mass leaving or entering the airbag volume. (Character; Default = SMALL) <ul style="list-style-type: none"> <li>SMALL      The size of the hole in the surface is the same or smaller than the size of the Euler mesh used. The velocity of the gasflow through the hole is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is the method that is used if the PORHOLE is used on a GBAG entry.</li> <li>LARGE      The size of the hole in the surface is larger than the Euler mesh used. The velocity of the gasflow through the hole is based on the velocity method for an Eulerian air bag. If the PORHOLE is used on a GBAG entry, it will default back to method used for SIZE=SMALL.</li> </ul>
FLOW	Defines the allowed directions of the flow. (Character; Default = BOTH) <ul style="list-style-type: none"> <li>BOTH      In- and outflow are allowed.</li> <li>IN          Only inflow allowed.</li> <li>OUT        Only outflow allowed.</li> </ul>
PENV	Environmental pressure. (Real > 0.0; Required)
RHOENV	Environmental density. (Real > 0.0; Required)
SIEENV	Environmental specific internal energy. (Real > 0.0; Required)
CP	Environmental specific heat at constant pressure. See Remark 4. (Real > 0.0)
MID	MATDEUL ID number of the environment gas. Only used for an Eulerian air bag that uses the multi-material Euler solver and SIZE=LARGE. See Remark 6. (Integer > 0)

**Remarks:**

1. The PORHOLE entry can be referenced from a LEAKAGE entry.
2. When used with Euler, this entry can only be used with the single material hydrodynamic Euler solver, using an EOSGAM (ideal gas) equation of state.
3. The values for the environment  $p_{env}$  (PENV),  $\rho_{env}$  (RHOENV),  $e_{env}$  (SIEENV) must be defined consistent with an ideal-gas equation of state:

$$p_{env} = (\gamma_{env} - 1)\rho_{env}e_{env}$$

The  $\gamma_{env}$  is calculated and is only used when inflow occurs. Inflow occurs when  $p_{env} > p_{inside}$ .

4. CP is only required if updating of Euler or gasbag gas constants is done, for example if hybrid inflators are defined.
5. For in and out flow of an uniform pressure air bag (GBAG), the material transport is based on the theory of one-dimensional gas flow through a small orifice, and depends on the pressure difference. This is equivalent to the PORHOLE entry.
6. When used in combination with the single material hydrodynamic Euler solver, an EOSGAM (ideal gas) equation of state is required. In that case, the material number, MID, can be left blank. When using the multi-material solver, the material number, MID, has to point to one of the Eulerian materials and the equation of state of that material has to be of type EOSGAM.



## PORHYDS

### Porous Flow Boundary with a Hydrostatic Pressure Profile

Prescribes a hydrostatic pressure profile on a porous BSURE. Used in SOL 700 only.

#### Format:

1	2	3	4	5	6	7	8	9	10
PORHYDS	PID								

#### Example:

PORHYDS	120								
---------	-----	--	--	--	--	--	--	--	--

Describer	Meaning
-----------	---------

PID	Unique number of a PORHYDS entry. (Integer > 0; Required)
-----	---

#### Remarks:

1. Reference PID by a LEAKAGE entry.
2. The surface can be only a general coupling surface (see the COUPLE entry).
3. It is required that the coupling surface refers to a HYDSTAT entry. This HYDSTAT entry will be used to prescribe a hydrostatic pressure profile on the subsurface. For example, the water level and atmospheric pressure are taken from the HYDSTYAT entry. This defines the pressure and the inflow density.
4. In contributions of the surface to the Euler elements the pressure gradient across the surface is taken into account. Therefore splitting up of the surface and creating new PORHYDS entries does not increase the accuracy of prescribed pressures. If the water level and atmospheric pressure are the same in the whole region outside the coupling surface using one PORHYDS entry is sufficient.
5. The atmospheric pressure is prescribed on those parts of the surface that are above the water level.

## PORUDS

User-defined Porosity Model Specified by a User Defined Service.

Defines a porosity model of a COUPLE surface through a user-written subroutine. Use in SOL700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PORUDS	PID	GROUP	UNAME						

### Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE mypor 'SCA.MDSolver.Obj.Uds.Dytran.Flow'
```

*In Bulk Data:*

1	2	3	4	5	6	7	8	9	10
PORUDS	12	mypor	EXPOR						

Describer	Meaning
PID	Unique porosity model ID. (Integer > 0; Required)
GROUP	The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)
UNAME	User subroutine name associated with the entry. (Character; default=EXPOR)

### Remarks:

1. Only can be used for SOL 700.
2. The porosity ID (PID) must be referenced by a LEAKAGE entry.
3. UNAME can be:

Subroutine Name	Function
EXPOR	Standard user defined flow boundary on the coupling

## PRAC2D

## CRAC2D Element Property

Defines the properties and stress evaluation techniques to be used with the CRAC2D structural element.

### Format:

1	2	3	4	5	6	7	8	9	10
PRAC2D	PID	MID	T	IPLANE	NSM	GAMMA	PHI		

### Example:

PRAC2D	108	2	0.10	0	.17	.50	180.		
--------	-----	---	------	---	-----	-----	------	--	--

### Describer

### Meaning

PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
T	Element thickness. (Real > 0.0)
IPLANE	Plane strain or plane stress option. Use 0 for plane strain; 1 for plane stress. (Integer = 0 or 1, Default = 0)
NSM	Non-structural mass per unit area. (Real $\geq$ 0.0; Default = 0)
GAMMA	Exponent used in the displacement field. See Remark 4. (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x-axis along which stress intensity factors are to be calculated. See Remark 4. (Real; Default = 180.0)

### Remarks:

1. PRAC2D is a primary property entry. Primary property entries are grouping entities for many applications in Nastran. Therefore it is highly recommended that the PRAC2D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC2D entries.
2. PRAC2D entry may refer to MAT1, MAT2, or MAT8 material property entries.
3. For plane strain analysis, only MAT1 type data should be used.
4. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

**PRAC3D****CRAC3D Element Property**

Defines the properties of the CRAC3D structural element.

**Format:**

1	2	3	4	5	6	7	8	9	10
PRAC3D	PID	MID	GAMMA	PHI					

**Example:**

PRAC3D	108	2	.50	180.					
--------	-----	---	-----	------	--	--	--	--	--

**Describer****Meaning**

PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
GAMMA	Exponent used in the displacement field. See Remark 3. (Real; Default = 0.5)
PHI	Angle (in degrees) relative to the element x axis along which stress intensity factors are to be calculated. See Remark 3. (Real; Default = 180.0)

**Remarks:**

1. PRAC3D is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PRAC3D property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PRAC3D entries.
2. Either isotropic (MAT1) or anisotropic (MAT9) material entries may be referenced.
3. Nondefault values for GAMMA and PHI have not been tested. Therefore, the default value should be used.

## PRESAX Conical Shell Pressure Load

Defines the static pressure loading on a conical shell element.

### Format:

1	2	3	4	5	6	7	8	9	10
PRESAX	SID	P	RID1	RID2	PHI1	PHI2			

### Example:

PRESAX	3	7.92	4	3	20.6	31.4			
--------	---	------	---	---	------	------	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
P	Pressure value. (Real)
RID1, RID2	Ring identification numbers. See RINGAX entry. (Integer > 0)
PHI1, PHI2	Azimuthal angles in degrees. (Real; PHI2 > PHI1)

### Remarks:

1. PRESAX is allowed only if an AXIC entry is also present.
2. Load sets must be selected with the Case Control command LOAD = SID.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.
4. For axisymmetric loading over 360 degrees, use PHI1 = 0.0 and PHI2 = 360.0.

**PRESPT** Fluid Pressure Point

Defines the location of pressure points in the fluid for recovery of pressure data.

**Format:**

1	2	3	4	5	6	7	8	9	10
PRESPT	IDF	IDP1	PHI1	IDP2	PHI2	IDP3	PHI3		

**Example:**

PRESPT	14	141	0.0			142	90.0		
--------	----	-----	-----	--	--	-----	------	--	--

<b>Describer</b>	<b>Meaning</b>
IDF	Fluid point (RINGFL entry) identification number. (Integer > 0)
IDPi	Pressure point identification number. (Integer > 0)
PHIi	Azimuthal position on fluid point referenced by IDF in fluid coordinate system. (Real)

**Remarks:**

1. PRESPT is allowed only if an AXIF entry is also present.
2. All pressure point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. The pressure points are used primarily for the identification of output data. They may also be used as points at which to measure pressure for input to control devices.
4. One, two, or three pressure points may be defined per entry.
5. Output requests for velocity and acceleration of these degrees-of-freedom will result in derivatives of pressure with respect to time.

## PRIMx

### Thermal Geometric Primitives for RC Radiation

Specifies the properties of geometric primitives to be used in radiation calculations in place of elements.

#### Format (GEOM2):

1	2	3	4	5	6	7	8	9	10
PRIM1	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	A_mesh	B_mesh				

#### Example:

PRIM1	11	101	102	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	3	4				

#### Format:

1	2	3	4	5	6	7	8	9	10
PRIM2	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	P4(1)	P4(2)	P4(3)	A_mesh	B_mesh	

#### Example:

PRIM2	12	102	103	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	1.	1.	0.	3	4	

#### Format:

1	2	3	4	5	6	7	8	9	10
PRIM3	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	A_mesh	B_mesh				

#### Example:

PRIM3	13	103	104	3	4	2			
	0.	0.	0.	1.	0.	0.			
	0.	1.	0.	3	4				

**Format:**

1	2	3	4	5	6	7	8	9	10
PRIM4	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Diam2	Angle1	Angle2		
	A_mesh	B_mesh							

**Example:**

PRIM4	14	104	105	3	4	2			
	0.	0.	0.	0.0	1.0	0.			
	0.	0.	1.	1.	0.	60.	180.		
	3	4							

**Format:**

1	2	3	4	5	6	7	8	9	10
PRIM5	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	A_mesh	B_mesh	

**Example:**

PRIM5	15	105	106	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	3	4	

**Format:**

1	2	3	4	5	6	7	8	9	10
PRIM6	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Diam2	Angle1	Angle2		
	A_mesh	B_mesh							

**Example:**

PRIM6	16	106	107	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	0.	60.	80.		
	3	4							



**Format:**

1	2	3	4	5	6	7	8	9	10
PRIM7	PRIMID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	Trunc1	Trunc2	
	A_mesh	B_mesh							

**Example:**

PRIM7	17	107	108	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	-0.5	0.5	
	3	4							

**Format:**

1	2	3	4	5	6	7	8	9	10
PRIM8	PRIMID	IVIEWF	IVIEWB	RADMID F	RADMID B	SET3ID			
	P1(1)	P1(2)	P1(3)	P2(1)	P2(2)	P2(3)			
	P3(1)	P3(2)	P3(3)	Diam1	Angle1	Angle2	Trunc1	Trunc2	
	A_mesh	B_mesh							

**Example:**

PRIM8	18	108	109	3	4	2			
	0.	0.	0.	0.	1.	0.			
	0.	0.	1.	1.	60.	180.	0.5	1.	
	3	4							

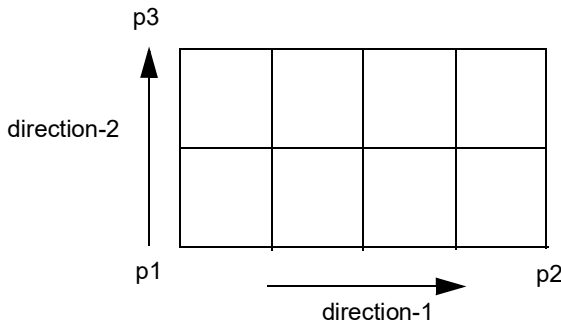
Describer	Meaning
PRIMID	Primitive identification number, unique to all other PRIMx (Integer > 0; Required)
IVIEWF	A VIEW entry identification number for the front face of surface element. (Integer > 0).
IVIEWB	A VIEW entry identification number for the back face of surface element. (Integer > 0).
RADMIDF	RADM identification number for front face of surface element. (Integer > 0).
RADMIDB	RADM identification number for back face of surface element. See Remark 2.. For default. (Integer > 0).

Describer	Meaning
SET3ID	ID of the element collection to which this primitive describes. This collection acts as the elements that will exist in the thermal model, but the collection will be absent from the radiation model. Instead, the primitive will be used to calculate radiation, and be redistributed back onto the elements. All radiation properties for the primitive will be applied to the element collection, and must be consistent across. (Integer > 0; Required)
Pi(a)	The position of point i in the a axis, as described in the correlating picture. For example P2(2) denotes the y coordinate of the second point. Position is always described in global coordinates. (Real; Required)
Diamx	Diameter x of the primitive, if applicable and as described in the correlating picture. (Real ≥ 0.0; Required)
Anglex	Angle x of the primitive in degrees, if applicable and as described in the correlating picture. (0 ≤ Real ≤ 360.0; Required)
Truncx	Truncation x of the primitive, if applicable and as described in the correlating picture. (For PRIM8, Real ≥ 0; Required). (For PRIM7, -0.5*Diam1 ≤ Real ≤ 0.5*Diam1)
A_mesh	Number of mesh spaces in parametric direction-1, as described in the correlating picture. (Integer > 0; Required)
B_mesh	Number of mesh spaces in parametric direction-2, as described in the correlating picture. (Integer > 0; Required)

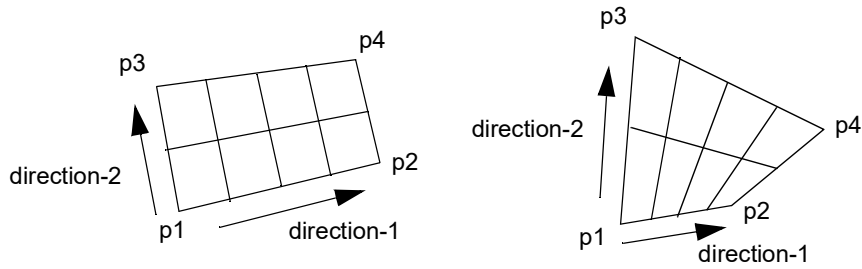
**Remarks:**

1. This entry is for RC Network solver only.
2. Set 3 should include all the elements which belong to this primitive. It will cause wrong results if only partial of the elements are included.
3. About the primitives

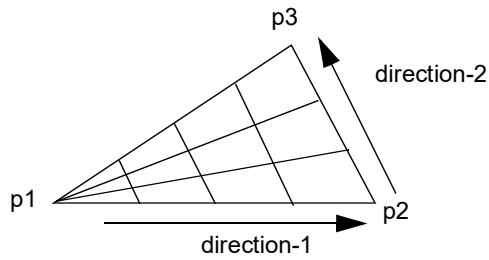
Prim1: Rectangle



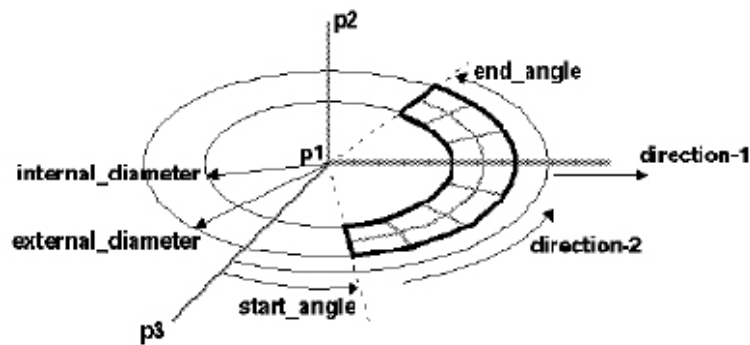
Prim2: Quad



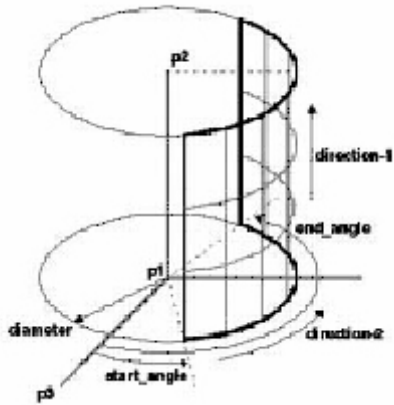
Prim3: Triangle



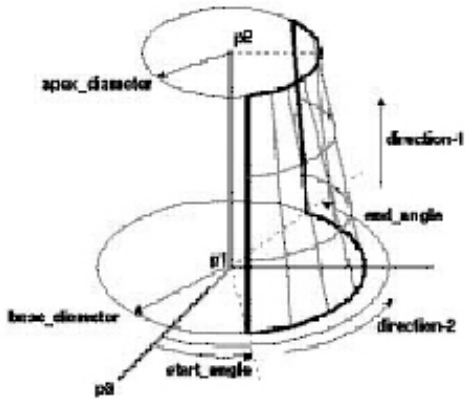
Prim4: Disc



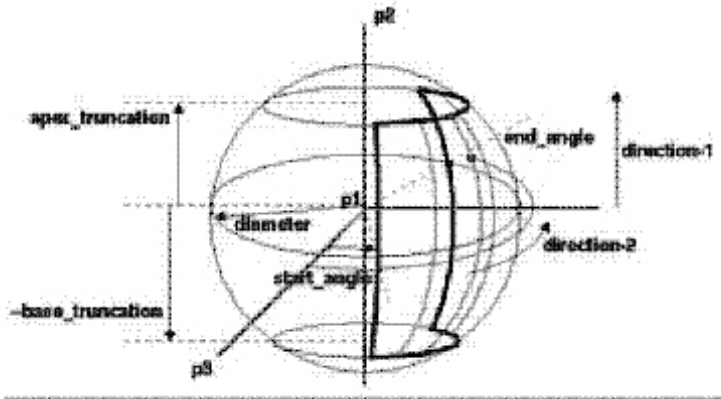
Prim5: Cylinder



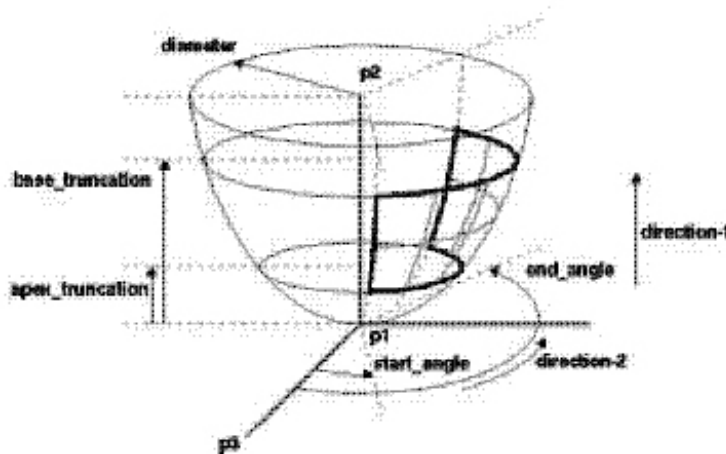
Prim6: Cone



Prim7: Sphere



Prim8: Parabolic



**PRJCON** Thermal RC Element Contact

Specifies a thermal connection between two regions of elements. The connection is automatically determined geometrically as a projection of the secondary region on to the primary, and the strength of the connection is calculated based on the properties given.

**Format: (HEAT1)**

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT1"	SET3 PRIMARY	SET3 SECNDRY	h					

**Example:**

PRJCON	1								
	HEAT1	1	2	1.2					

**Format: (HEAT2)**

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT2"	SET3 PRIMARY	SET3 SECNDRY	ID					

**Example:**

PRJCON	1								
	HEAT2	1	2	1001					

**Format: (HEAT3)**

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT3"	SET3 PRIMARY	SET3 SECNDRY	F	Emis Primary	Emis Secndry			

**Example:**

PRJCON	1								
	HEAT3	1	2	1.	0.85	0.5			

**Format: (HEAT4)**

1	2	3	4	5	6	7	8	9	10
PRJCON	BID								
	"HEAT4"	SET3 PRIMARY	SET3 SECNDRY	F	RADC ID Primary	RADC ID Secndry			



**Example:**

PRJCON	1								
	HEAT4	1	2	1.	1001	1002			

Describer	Meaning
BID	BCBODY identification number (Integer > 0; Required)
HEATx	Indicates the start of HEAT elements (Character)
SET3 Primary	ID of the primary element collection for connection (Integer > 0; Required)
SET3 Secondary	ID of the secondary element collection for connection (Integer > 0; Required)
h	Convection correlation (Real $\geq 0.0$ ; Required for HEAT1)
PCONID	PCONID of the property to be used for h value. (Integer > 0; Required for HEAT2)
F	View factor between parts ( $0.0 \leq \text{Real} \leq 1.0$ ; Required for HEAT3 & HEAT4)
Emis Primary	Emissivity of primary collection (Real $\geq 0.0$ ; Required for HEAT3)
Emis Secndry	Emissivity of secondary collection (Real $\geq 0.0$ ; Required for HEAT3)
RADC id Primary	RADMID of the material to be used for primary emissivity value. (Integer > 0; Required)
RADC id SECNDRY	RADMID of the material to be used for secondary emissivity value. (Integer > 0; Required)

**Remarks:**

1. This “HEATx” and the later parameters are for RC Network solver only.
2. For HEAT2, PCONID must refer to a PCONV1 type and not a PCONV or PCONVM.
3. RC Network Solver uses a projection method to determine the connection (not the “nearest neighbor method”. In most of the cases, the projection method is more accurate than the nearest neighbor method.

**PROD**

## Rod Property

Defines the properties of a rod element (CROD entry).

**Format:**

1	2	3	4	5	6	7	8	9	10
PROD	PID	MID	A	J	C	NSM			

**Example:**

PROD	17	23	42.6	17.92	4.2356	0.5			
------	----	----	------	-------	--------	-----	--	--	--

**Describer****Meaning**

PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 2. and 3. (Integer > 0)
A	Area of the rod. (Real)
J	Torsional constant. (Real)
C	Coefficient to determine torsional stress. (Real; Default = 0.0)
NSM	Nonstructural mass per unit length. (Real)

**Remarks:**

1. PROD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PROD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PROD entries.
2. For structural problems, MID must reference a MAT1 material entry.
3. For heat transfer problems, MID must reference a reference MAT4 or MAT5 entry.
4. The formula used to calculate torsional stress is

$$\tau = \frac{CM_{\theta}}{J}$$

where  $M_{\theta}$  is the torsional moment.



## PRODN1

## Nonlinear Property Extensions for a PROD Entry

Specifies additional nonlinear properties for elements that point to a PROD entry in SOL 400.

### Format:

1	2	3	4	5	6	7	8	9	10
PRODN1	PID	MID			ANAL				
	"C2"	BEH2	INT2	BEH2H	INT2H				

### Example:

PRODN1	22	98							
--------	----	----	--	--	--	--	--	--	--

### Describer

### Meaning

PID	Property identification number of an existing PROD entry. (Integer > 0)
MID	Material identification number. Remarks 7. (Integer ≥ 0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL='ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH).
C2	Keyword indicating that items following apply to elements with two end grids. (Character)
BEH2	Element structural behavior. See Remark 4. (Character Default ROD)
INT2	Integration scheme. See Remarks 4. and 5. (Character Default L)
BEH2H	Element heat behavior. See Remark 4. (Character Default ROD)
INT2H	Integration scheme. See Remarks 4. and 5. (Character Default L)

### Remarks:

1. The PID above must point to an existing PROD Bulk Data entry and is honored only in SOL 400.
2. MID if blank (or 0) use the MID value on the PROD entry. If > 0 it will override the MID value on the PROD.
3. The MID entry may point to the MAT1 entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

**Caution:** The MATVE, MATVP, MATEP, and MATF entries are only associated with a CROD element if the CROD element refers to a PRODN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials
MAT1
MATVE
MATVP
MATEP
MATF
MATS1

Heat Materials	
MAT4	MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.  
 If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID.  
 MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.  
 MID for heat entries must follow the uniqueness of the MAT4 and MAT5 entries.

4. BEH2/BEH2H refers to the nonlinear structural/heat behavior of the ROD element. An underlined item delineates default.

Structural/Heat Classification of Elements				
Element Structural/Heat Type	BEH2/BEH2H CODE	Integration Code	Element Type	# Nodes
Rod	ROD	<u>L</u>	ROD	2

5. Integration codes in Remark 4. are:

INT CODE	Integration Type
L	Linear

6. Any J, C, or NSM value on the PROD will be ignored.
7. The structural element damping coefficient, GE, is not supported on elements which reference PRODN1.

## PSEAM

## CSEAM Property

Defines the PSEAM property values.

### Format:

1	2	3	4	5	6	7	8	9	10
PSEAM	PID	MID	TYPE	W	T	IN			

### Example:

PSEAM	7	1		16.					
-------	---	---	--	-----	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number. (Integer > 0)
TYPE	“KEYWORD” type of Seam Weld generated. (Character; Default = LINE)
W	Width of the SEAM. See Remark 1. (Real > 0.)
T	Thickness of the SEAM. See Remark 2. (Real > 0. or blank)
IN	Integration scheme. See Remark 6. of PSOLID entry. (Integer 0 or 2; Default = 2) If IN = 0, 2x2x2 reduced shear integration with bubble functions. If IN = 2, 2x2x2 reduced shear integration.

### Remarks:

1. The length of the SEAM is the distance between GS and GE. The width  $W$  of the SEAM is measured perpendicular to the length and lies in the plane of the patches A and B (see [Figure 9-136](#)). The width is also used to find the projection of the SEAM on the two patches A and B.

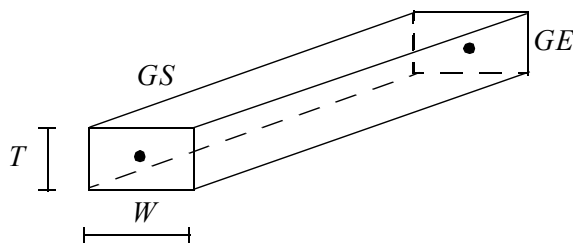


Figure 9-136 Dimensions of a CSEAM Element

2. If left blank, the thickness will be computed as  $T = (T_A + T_B)/2$  where  $T_A$  is the thickness of patch A and  $T_B$  is the thickness of patch B.

3. PSEAM is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSEAM property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSEAM entries.

## PSHEAR Shear Panel Property

Defines the properties of a shear panel (CSHEAR entry).

### Format:

1	2	3	4	5	6	7	8	9	10
PSHEAR	PID	MID	T	NSM	F1	F2			

### Example:

PSHEAR	17	23	42.6	17.92	4.236	0.5			
--------	----	----	------	-------	-------	-----	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Material identification number of a MAT1 entry. (Integer > 0)
T	Thickness of shear panel. (Real ≠ 0.0)
NSM	Nonstructural mass per unit area. (Real)
F1	Effectiveness factor for extensional stiffness along edges 1-2 and 3-4. See Remark 2. (Real ≥ 0.0; Default = 0.0)
F2	Effectiveness factor for extensional stiffness along edges 2-3 and 1-4. See Remark 2. (Real ≥ 0.0; Default = 0.0)

### Remarks:

1. PSHEAR is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSHEAR property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSHEAR entries.
2. The effective extensional area is defined by means of equivalent rods on the perimeter of the element. If  $F1 \leq 1.01$ , the areas of the rods on edges 1-2 and 3-4 are set equal to  $(F1 \cdot T \cdot PA) / (L12 + L34)$  where PA is the panel surface area-half the vector cross product area of the diagonals-and L12, L34 are the lengths of sides 1-2 and 3-4. Thus, if  $F1 = 1.0$ , the panel is fully effective for extension in the 1-2 direction. If  $F1 > 1.01$ , the areas of the rods on edges 1-2 and 3-4 are each set equal to  $0.5 \cdot F1 \cdot T^2$ .

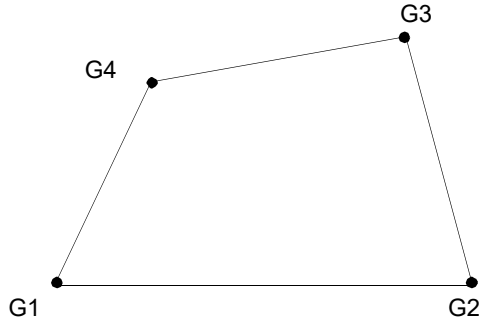


Figure 9-137 Extensional Area for Shear Panel

Thus, if  $F1 = 30$ , the effective width of skin contributed by the panel to the flanges on edges 1-2 and 3-4 is equal to  $15T$ . The significance of  $F2$  for edges 2-3 and 1-4 is similar.

3. Poisson's ratio coupling for extensional effects is ignored.
4. The parameter entry MDLPRM,SHEARP,GARVEY(default) selects the standard Garvey shear panel. MDLPRM,SHEARP,HARDER selects the Harder shear panel: See Remark 4. of the CSHEAR entry.

## PSHEARN

## Nonlinear Property Extensions for a PSHEAR Entry

Specifies nonlinear properties for elements that point to a PSHEAR entry in SOL 400.

### Format:

1	2	3	4	5	6	7	8	9	10
PSHEARN	PID	MID			ANAL				
	"C4"	BEH4	INT4						

### Example:

PSHEARN	22	18							
---------	----	----	--	--	--	--	--	--	--

### Describer

### Meaning

PID	Property identification number of an existing PSHEAR entry. (Integer > 0)
MID	Material identification number. Remark 9. (Integer ≥ 0 or blank)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. (Character: Default IS)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
BEH4	Element structural behavior. See Remark 5. (Character Default MB)
INT4	Integration scheme. See Remarks 4. and 6. (Character Default L)

### Remarks:

1. The PID must point to an existing PSHEAR Bulk Data entry and is honored only in SOL 400.
2. Only large membrane rotation is supported. Stringer effectiveness is ignored and only membrane action is considered.
3. MID if blank (or 0) use the MID value on the PSHEAR entry. If > 0 is will override the MID value on the PSHEAR.
4. The MID entry for nonlinear structures may point to the MAT entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The MATVE, MATVP, MATEP, MATF and MATSMA entries are only associated with a CSHEAR element if the CSHEAR element refers to a PSHEARN entry.

Implicit Structure Materials
MAT1
MATVE
MATVP
MATEP
MATF
MATS1
MATSMA

5. BEH4 refers to the nonlinear structural behavior of the SHEAR element. An underlined item delineates default.

Structural Classification of Elements				
Element Structural Type	BEH2/BEH2H CODE	Integration Code	Element Type	# Nodes
SHEAR	MB	<u>L</u>	SHEAR	4

6. Integration codes in Remark 4. are:

INT CODE	Integration Type
L	Linear

7. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
8. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
9. The structural element damping coefficient, GE, is not supported on elements which reference PSHEARN.



## PSHELL Shell Element Property

Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12I/T**3	MID3	TS/T	NSM	
	Z1	Z2	MID4						

### Example:

PSHELL	203	204	1.90	205	1.2	206	0.8	6.32	
	+.95	-.95							

### Describer

### Meaning

PID	Property identification number. (Integer > 0)
MID1	Material identification number for the membrane. (Integer ≥ 0 or blank)
T	Default membrane thickness for Ti on the connection entry. If T is blank then the thickness must be specified for Ti on the CQUAD4, CTRIA3, CQUAD8, and CTRIA6 entries. (Real or blank) Average thickness if TFLAG = 1 ( $T_i = T_i \cdot T$ ).
MID2	Material identification number for bending. (Integer ≥ -1 or blank)
12I/T**3	Bending moment of inertia ratio, $12I/T^3$ . Ratio of the actual bending moment inertia of the shell, $I$ , to the bending moment of inertia of a homogeneous shell, $T^3/12$ . The default value is for a homogeneous shell. (Real > 0.0; Default = 1.0)
MID3	Material identification number for transverse shear. If MID2 is blank or -1, then MID3 must be blank. (Integer > 0 or blank)
TS/T	Transverse shear thickness ratio, $T_s/T$ . Ratio of the shear thickness, ( $T_s$ ), to the membrane thickness of the shell, $T$ . The default value is for a homogeneous shell. (Real > 0.0; Default = .833333)
NSM	Nonstructural mass per unit area. (Real)
Z1, Z2	Fiber distances for stress calculations. The positive direction is determined by the right-hand rule, and the order in which the grid points are listed on the connection entry. See Remark 11. for defaults. (Real or blank)
MID4	Material identification number for membrane-bending coupling. See Remarks. (Integer > 0 or blank, must be blank unless MID1 > 0 and MID2 > 0, may not equal MID1 or MID2.)

**Remarks:**

1. PSHELL is a primary property entry. Primary properties are used to internally group entities. Therefore it is highly recommended that PSHELL entries use unique identification numbers (PIDs) with respect to all other property entries to avoid unexpected grouping results. PSHELL, PCOMP, and PCOMPG entries must have unique PIDs.
2. The structural mass is calculated from the density using the membrane thickness and membrane material properties. If MID1 is blank, then the density is obtained from the MID2 material.
3. The results of leaving an MID<sub>i</sub> field blank (or MID2 = -1) are:

MID1	No membrane or coupling stiffness
MID2	No bending, coupling, or transverse shear stiffness
MID3	No transverse shear flexibility
MID4	No bending-membrane coupling unless ZOFFS is specified on the connection entry. See Remark 6.
MID2=-1	See Remark 12.

**Note:** MID1 and MID2 must be specified if the ZOFFS field is also specified on the connection entry.

4. The continuation entry is not required.
5. The structural damping (GE on the MAT<sub>i</sub> entry) is obtained from the MID1 material. If MID1 is blank, then it is obtained from the MID2 material. If PARAM,SHLDAMP,DIFF is set or is anything other than SAME, then the structural damping  $K^4$  matrix is computed using the GE entries on the MAT<sub>i</sub> entries according to rules in the following table (Table 28). If a single PSHELL corresponds to row eight (8) of Table 28, then all PSHELLs in the model will follow the rule of row eight (8). Rows 1-7 is an attempt to maintain upward compatibility, if a user inadvertently places a SHLDAMP,DIFF in the model.

**Note:** Large values of damping associated with an MID4 entry, when using PARAM,SHLDAMP,DIFF, can cause structural instability in transient dynamics.

Table 28 SHELL Structural Damping Rules

SHELL Structural Damping Rules					
Row	MID1	MID2	MID3	MID4	$K^4$ based on
1*	v	v			GE on MID1
2	v				
3	v	-1			
4	v	v			
5		v			GE on MID2
6		v	v		GE on MID1 if  $n = m$ and $ge_i = ge_2 = \dots = ge_m$ or $m = 1$ and $ge_1 \neq 0$ or $m = 0$  where: $n$ is total number of non blank MIDi (vi) $m$ is total number of non zero $ge_i$
7	v1	v2	v3	v4	
8	v1	v2	v3	v4	Otherwise:  $GE = ge_1 \cdot \text{membrane-stiff} +$ $ge_2 \cdot \text{bending-stiff} +$ $ge_3 \cdot \text{transverse shear-stiff} +$ $ge_4 \cdot \text{bending-membrane-stiff}$
* v $\rightarrow$ MIDi values the same; vi $\rightarrow$ MIDi values different or blank $ge_i \rightarrow$ GEvalue from a MATj entry associated with MIDi					
If for row eight (8), a $ge_i = 0$ , it is replaced by $ge_i = 1.0E-8$					

PSHELL rules with extended MAT2 entries
The PSHELL has four MIDi entries: MID1-Membrane; MID2-Bending; MID3-Transverse Shear; MID4-Membran-Bending Coupling.
The Damping for shell elements is determined by the GE value associated to the material entry specified by MID1 (or MID2 if MID1 is blank).
PSHELL allows any of its MIDi fields to use any combination of MAT1, MAT2, or MAT8 material entries.
The MAT2 entry has a feature consisting of six (6) structural damping coefficients called GE11, GE12, GE13, GE22, GE23, and GE33 in a <i>second continuation entry</i> . When any one or more of these GEij entries are non-zero, the GE entry in field six (6) of the first continuation entry is ignored for that material.

PSHELL rules with extended MAT2 entries	
<p>MAT2 entries with any GEij are called <i>extended</i> MAT2 entries. The damping is computed for such a material as <math>G_{ij} = G_{ij} * GE_{ij}</math> where <math>G_{ij}</math> are the elastic material coefficients on the parent MAT2 entry. An element structural damping matrix is then generated using these scaled <math>G_{ij}</math>.</p> <p>Any PSHELL with an <i>extended</i> MAT2 in the MID1 field assumes that every MID2 and MID4 present will performs a <math>G_{ij} = G_{ij} * GE_{ij}</math> calculation to form the appropriate structural damping matrix. The MID3 calculation is of the form <math>J_{11} = J_{11} * GE_{11}</math>, <math>J_{12} = J_{12} * GE_{12}</math>, and <math>J_{22} = J_{22} * GE_{22}</math>.</p>	
1	<p>For the case where MID1 is <i>extended</i> (is a reference to a MAT2 with GEij defined) but MID2, MID3, and/or MID4 contain a MAT1, a non-<i>extended</i> MAT2, or a MAT8 entry the calculation is of the form <math>G_{ij} = G_{ij} * GE</math> where GE is the structural damping coefficient of the associated MAT1, MAT2, or MAT8 entry.</p>
2	<p>For the case where MID1 is a MAT1, a non-<i>extended</i> MAT2, or a MAT8, but any one of the MID2, MID3, or MID4 fields have <i>extended</i> MAT2 entries, the structural damping coefficient GE of the MATi entry is used to simply scale the element stiffness as <math>K4</math> matrix contribution <math>? GE * K</math>.</p>
3	<p>For case two (2) above, the user has the option of using PARAM,SHLDAMP,DIFF in which case any MID2, MID3, or MID4 containing an <i>extended</i> MAT2 entry force the use or the extended fields. HOWEVER, remember the PARAM,SHLDAMP,DIFF is global – meaning EVERY PSHELL scales each of its MIDi material by the appropriate GEij or GE to form an element structural damping matrix.</p>

6. The following should be considered when using MID4.
  - The MID4 field should be left blank if the material properties are symmetric with respect to the middle surface of the shell. If the element centerline is offset from the plane of the grid points but the material properties are symmetric, the preferred method for modeling the offset is by use of the ZOFFS field on the connection entry. Although the MID4 field may be used for this purpose, it may produce ill-conditioned stiffness matrices (negative terms on factor diagonal) if done incorrectly.
  - Only one of the options MID4 or ZOFFS should be used; if both methods are specified the effects are cumulative. Since this is probably not what the user intends, this may results in unexpected answers. Note that the mass properties are not modified to reflect the existence of the offset when the ZOFFS and MID4 methods are used. If the weight or mass properties of an offset plate are to be used in an analysis, the RBAR method must be used to represent the offset. See [Shell Elements \(CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, CQUADR\)](#) in the *MSC Nastran Reference Guide*.
  - The effects of MID4 are not considered in the calculation of differential stiffness. Therefore, it is recommended that MID4 be left blank in buckling analysis.
7. This entry is referenced by the CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, and CQUADR entries via PID.
8. For structural problems, MIDi must reference a MAT1, MAT2, or MAT8 material property entry

9. If the transverse shear material MID3 or the membrane-bending coupling term MID4 references a MAT2 entry, then G33 must be zero. If MID3 references a MAT8 entry, then G1Z and G2Z must not be zero.
10. For heat transfer problems, MIDi must reference a MAT4 or MAT5 material property entry.
11. The default for Z1 is  $-T/2$ , and for Z2 is  $+T/2$ . T is the local plate thickness defined either by T on this entry or by membrane thicknesses at connected grid points, if they are input on connection entries.
12. For plane strain analysis, set MID2=-1 and set MID1 to reference a MAT1 entry. In-plane loads applied to plain strain elements are interpreted as line-loads with a value equal to the load divided by the thickness. Thus, if a thickness of "1.0" is used, the value of the line-load equals the load value. Pressure can be approximated with multiple line loads where the pressure value equals the line-load divided by the length between the loads. For SOL 600, plain strain models must be in the basic coordinate system X-Y plane.
13. For a material nonlinear property, MID1 must reference a MATS1 entry and be the same as MID2, unless a plane strain (MID2 = -1) formulation is desired. Also, MID3 cannot reference a MATS1 entry.
14. If transverse shear flexibility is specified for a model with curved shells where the loading is dominated by twist and shell normals are turned off (e.g., PARAM,SNORM,-1), then results may be inaccurate and may diverge when the mesh is refined. PARAM,SNORM should be set for this unique model condition.
15. For 3D contact analysis, MID2 must be non-zero and for 2D contact analysis, MID1 must be non-zero.

## PSHELL1

### Properties of shell elements for SOL 700 only

Defines the properties of shell elements with variable shell thickness for SOL 700 only.

#### Format:

1	2	3	4	5	6	7	8	9	10
PSHELL1	PID	MID	FORM	QUAD	NUMB	SHFACT	REF	SPINCOR	
+	T1	T2	T3	T4	TRANSHR	SHRLCK			

#### Example:

PSHELL1	7	2	BLT	GAUSS	5	0.9			
	10.0	10.0	10.0	10.0					

Describer	Meaning
PID	Unique property number. (Integer; Required)
MID	Material number. See Remark 2. (Integer; Required)
FORM	Shell formulation. See Remark 2. (Character; Required)
	HUGHES     Hughes-Liu
	BLT             Belytschko-Lin-Tsay
	KEYHOFF     Key-Hoff
	C0-TRIA     C0 triangle
	MEMB         Membrane element (no bending)
	DUMMY        Dummy element (only applicable for FSI coupling)
QUAD	Type of quadrature. (Character; Default = GAUSS)
	GAUSS         Gauss quadrature
	LOBATTO     Lobatto quadrature
NUMB	The number of integration points through the thickness. (Integer; Default = 3 for quadrature). 1 for membrane element. Maximum value is 5.
SHFACT	Shear factor. (Real; Default = 0.83333)
REF	Reference surface. (character, default=MID)
	TOP            Reference surface is the top surface.
	MID            Reference surface is the central surface.
	BOT            Reference surface is the bottom surface.
SPINCOR	Spin correction. See Remark 5. (Character; default=NO)
	NO              No SPINCOR applied

Describer	Meaning
	YES            SPINCOR is applied
T1 toT4	Element thickness at the grid points. See Remark 4. (Real; Default = 0.0)
TRANSHR	Method of transverse-shear calculation. (Character; See Remark 6.)
	LINEAR        Linear transverse shear
	CONSTANT    Constant transverse shear
	CONAPX        Approximated constant transverse shear
SHRLCK	Shear-lock avoidance. (Character; See Remark 6.)
	AVOID         Avoid shear lockup
	NOAVOID      No avoid

**Remarks:**

1. For constant thickness shell element with three-point Gauss integration, PSHELL entry is recommended.
2. For CQUAD4 elements, the default of FORM for formulation is KEYHOFF. For CTRIA3 elements, the default of FORM for formulation is CO-TRIA.
3. Property identification number (PID) must be unique.
4. If the thickness T1 is set to blank or 0.0, the thickness of the shell must be defined on the CTRIA3 and CQUAD4 entry.
5. The options for SPINCOR are:

NO                No SPINCOR correction is applied.  
 YES                SPINCOR correction is applied.

When SPINCOR = NO, slight asymmetric forces are applied to the shell element's grid points. This approach is, in general, acceptable up to about 10° in plane shear angle.

6. The following defaults apply:

	BLT	HUGHES	KEYHOFF
TRANSHR	Not Available	Not Available	LINEAR
SHRLCK	NOAVOID	Not Available	AVOID

## PSHLN1

## Nonlinear Property Extensions for a PSHELL or PCOMP(G) Entry

Specifies additional nonlinear properties for shell elements that point to a PSHELL or PCOMP(G) entry in SOL 400.

### Format:

1	2	3	4	5	6	7	8	9	10
PSHLN1	PID	MID1	MID2		ANAL				
	"C3"	BEH3	INT3	BEH3H	INT3H				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C6"	BEH6	INT6	BEH6H	INT6H				
	"C8"	BEH8	INT8	BEH8H	INT8H				

### Example:

PSHLN1	22	98							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PSHELL or PCOMP(G) entry. (Integer > 0)
MID1	Membrane material identification number. See Remark 17. (Integer $\geq 0$ or blank)
MID2	Bending material identification number. See Remark 17. (Integer $\geq 0$ or blank)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. ANAL = 'IH' - Implicit heat analysis elements are being referred to. ANAL = 'ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C3	Keyword indicating that two items following apply to elements with three corner grids. (Character)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C6	Keyword indicating that items following apply to elements with three corner grids and three midside grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)
BEHi	Element structural behavior. See Remark 12. (Character Default: DCTN for BEH3, DCT for BEH4, and DCT for BEH8, MB for BEH6)
INTi	Integration scheme. See Remarks 11. and 13. (Character Default: LDK for INT3, L for INT4, QRI for INT8, Q for INT6)



Describer	Meaning
BEHiH	Element heat behavior. See Remark 12. (Character Default: DCT for BEH3H, BEH4H, and BEH8H, MB for BEH6H)
INTiH	Integration scheme. See Remarks 11. and 13. (Character Default: L for INT3H, L for INT4H, Q for INT8H and INT6H)

**Remarks:**

- The PID must point to an existing PSHELL, PCOMP, or PCOMPG Bulk Data entry and is honored only in SOL 400.
- The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
- MID1 if blank (or 0) use the MID1 value on the PSHELL. If > 0 it will override the MID1 value on the PSHELL. MID1 is ignored for PCOMP/PCOMPG.
- MID2:
  - If BEHi = DCT or DCTN: If blank (or 0) use the MID2 value on the PSHELL. If > 0 it will override the MID2 value on the PSHELL. If MID2 = -1 on the PSHELL entry it must be replaced with a positive value MID2 entry or the PSHELL should be replaced with a PLPLANE entry and a PSHLN2 entry should be used instead of a PSHLN1 entry.
  - If BEHi = MB: MID2 on both the PSHELL and PSHLN1 entries are ignored. MID2 is ignored for PCOMP/PCOMPG.
- The MID1 or MID2 entries, were applicable, may point to MAT1, MAT2, MAT8, MATHE and MATSMA entries. The table below shows associated nonlinear entries. The association is established through the material entries having the same values as the MID1 or MID2 entries.

Caution: The primary MATHE and the secondary MATVE, MATVP, MATEP, MATF, and MATS8 entries are only associated with a Shell (CQUAD4, CQUADR, CQUAD8, CTRIA3, CTRIAR, or CTRIA6) element if the Shell element refers to a PSHLN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structural Materials			
MAT1	MAT2	MAT8	MATHE
MATVE	<MATVE>	<MATVE>	MATVE
MATVP	MATVP		
MATEP	MATEP	MATEP	
MATF	MATF	MATF	
MATS1		MATS8	
<MATVE> refers to the ALTERNATE format for type ORTHO			

6. If MID3 is not specified on the PSHELL, BEH4 = DCTN with INT4 = LDK should be used and any CQUAD8 elements using this PSHELL should have a new PSHELL with MID3 specified. If not a fatal message will be issued. MID3 is not used for PCOMP/PCOMPG.
7. If MID2 is not specified on the PSHELL or overridden with a nonzero value on the PSHLN1, BEH4 = MB. and BEH8 = MB. If not, a fatal message will be issued.

Heat Materials	
MAT4	MAT5

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.  
 If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.  
 MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

8. NSM is not currently supported when this entry is used.
9. When this entry points to a PCOMP or a PCOMPG, special restrictions should be noted for some of the PCOMP/PCOMPG entries:
  - NSM, FT and GE are not supported. If a failure theory for any ply defined on the PCOMP/PCOMG entry is required, a MATF entry should be specified for the associated material.
  - The allowable inter-laminar bond strength SB is supported. When used in conjunction with the parabolic shear option (NLMOPTS, TSHEAR), the provided value of SB is used to calculate the ply shear bond index for thick shells using the relation - Bond index = max(inter-laminar shear stress)/SB.
  - LAM=BLANK and LAM=SYM (only for PCOMP) are supported. No smearing i.e., conversion of PCOMP/PCOMG into equivalent PSHELL is supported. Conventional integration through the thickness and across all layers is used. LAM=MEM, BEND, SMEAR and SMCORE are treated in a manner similar to LAM=BLANK, i.e., the sequence of the plies and the data given for each ply is used to carry out the conventional thickness integration.
  - SOUTi is not supported for individual plies. If STRESS output is requested for a particular shell element, then integration point stresses and elastic strains in the material coordinate system for all plies of the element are printed. If the parabolic shear option is used, then integration point values of the inter-laminar stresses and the bond index are also printed.
  - Layer Composite (PCOM/PCOMG) is not supported to CTRAI6 by PSHLN1.
10. If BEHi = MB is selected on PSHLN1, any ZOFF entry on the element connection entry will be ignored with a user warning.
11. In the following table, BEHi refers to the nonlinear structural behavior of shell elements. An underlined item delineates a default.

Implicit Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Doubly-curved thick shell	DCT	<u>L</u>	QUAD	4
		<u>QRI</u>	QUAD	8
		LRIH	QUAD	4
Doubly-curved thin shell	DCTN	<u>LDK</u>	TRIA	3
		LDK	QUAD	4
Membrane three-dimensional	MB	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
		<u>Q</u>	TRIA	6

12. In the following table, BEHiH refers to the nonlinear heat behavior of shell elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Doubly-curved thick shell	DCT	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
Membrane three-dimensional	MB	L	QUAD	4
		Q	QUAD	8
		L	TRIA	3
		<u>Q</u>	TRIA	6

13. Integration codes in Remark 11. are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass control (assumed strain)
Q	Quadratic
QRI	Quadratic Reduced Integration
LDK	Linear Discrete Kirchhoff

14. Smear shell formulation will be used in case MID1 and MID2 are different from one another or different from MID3 used on the corresponding PSHELL entry. Smear shell formulation allows membrane, bending and transverse shear properties to be defined independently, however, it is limited to linear elastic material behavior only.
15. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation if BEH=PSTRS.
16. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available if BEH=PSTRS.
17. Structural damping is supported for elements which reference PSHLN1. The structural damping coefficient, GE, is supported for elements using the MAT1 entries.

**PSHLN2****Nonlinear Property Extensions for a PLPLANE Entry**

Specifies additional nonlinear properties for plane strain, plane stress, or axisymmetric elements that point to a PLPLANE entry in SOL 400.

**Format:**

1	2	3	4	5	6	7	8	9	10
PSHLN2	PID	MID	DIRECT	T	ANAL				
	"C3"	BEH3	INT3	BEH3H	INT3H				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C6"	BEH6	INT6	BEH6H	INT6H				
	"C8"	BEH8	INT8	BEH8H	INT8H				

**Example:**

PSHLN2	22	98							
--------	----	----	--	--	--	--	--	--	--

Descriptor	Meaning
PID	Property identification number of an existing PLPLANE entry. (Integer > 0)
MID	Material identification number. Remarks 13. (Integer > 0)
DIRECT	The layer direction for BEHi=COMPS or AXCOMP. See Remark 6. (Integer = 1 or 2; Default = 1)
T	A thickness. (Real; Default = 1.0)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. ANAL = 'IH' - Implicit heat analysis elements are being referred to. ANAL = 'ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C3	Keyword indicating that two items following apply to elements with three corner grids. (Character)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C6	Keyword indicating that two items following apply to elements with three corner grids and three midside grids. (Character)
C8	Keyword indicating that two items following apply to elements with four corner grids and four midside grids. (Character)
BEHi	Element structural behavior. See Remark 7. (Character Default: PLSTRN for BEH3, BEH4, BEH6, and BEH8)
INTi	Integration scheme. See Remarks 7. and 10. (Character Default: L for INT3, INT4, Q for INT6 and INT8)

Describer	Meaning
BEHiH	Element heat behavior. See Remark 9. (Character Default: PLSTRN for BEH3H, BEH4H, BEH6H, and BEH8H.
INTiH	Integration scheme. See Remarks 9. and 10. (Character Default: L for INT3H, L for INT4H, Q for INT6H and INT8H)

**Remarks:**

1. The PID must point to an existing PLPLANE Bulk Data entry and is honored only in SOL 400. Since these are additional nonlinear properties to the PLPLANE, the PLPLANE must still have an associated MATHP.
2. It is REQUIRED to override the MID value on the PLPLANE entry.
3. The element must lie in the x-y plane of the basic system. The CID field of the PLPLANE entry is not valid for this entry.
4. The MID entry may point to MAT1, MAT3, MATHORT, MATHE, MATG or MATSMA entry and MUST be used to override the MID field on a PLPLANE entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MAT1, MAT3, MAT8, MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP, MATF, MATS1, MATS3, MATS8, and MATSORT entries are only associated with a 2D or axisymmetric (CQUAD4, CQUAD, CQUAD8, CTRIA3, or CTRIA6) or (CTRIAX or CQUADX) element if the element refers to a PSHLN2 entry.

Implicit Structural Materials							
MAT1	MAT2	MAT3	MAT8	MATORT	MATHE	MATG	MATHP
MATVE	<MATVE>	<MATVE>	<MATVE >	<MATVE>	MATVE		
MATVP	MATVP	MATVP		MATVP			
MATEP	MATEP	MATEP	MATEP	MATEP			
MATF	MATF	MATF	MATF	MATF			
MATS1		MATS3	MATS8	MATSORT			

<MATVE> refers to the ALTERNATE format for type ORTHO  
 MAT1 applicable to all BEHi codes of 7 below, except COMPS and AXCOMP.

MAT3 axisymmetric orthotropic applicable only to BEHi=AXSOLID code of 7 below.

MAT8 orthotropic applicable only to BEHi=PSTRS code of 7 below.

MATG for BEH4=COMPS or AXCOMP; INT4=L only.  
 MATG has an associated field IDMEM that points to a MAT1.

With MATHE, MATHP, and MAT1 (with MATS1 or MATEP) BEH3=IPS or IAX with INT3=L is recommended.



### Heat Materials

MAT4	MAT5
------	------

If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries. The CTRIAX6 remains a valid nonlinear heat transfer element BUT cannot be used in conjunction with this entry because it lies in an x-z plane and not an x-y plane.

- The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
- The following table describes layer orientation for BEHi=COMPS or AXCOMP.

Layer Orientation			
DIRECT	Normal to Layer edge	Layers run parallel from edge	to edge
1	Element Y direction	G1-G2	G4-G3
2	Element X direction	G1-G4	G2-G3

- In the following table, BEHi refers to the nonlinear structural behavior of 2D-solid elements. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Plane Stress	PSTRS	L	QUAD	4
		Q	QUAD	8
		QRI	QUAD	8
		LRIH	QUAD	4
		Q	TRIA	6
Plane Strain	PLSTRN	L	QUAD	4
		L	TRIA	3
		Q	QUAD	8
		QRI	QUAD	8
		LRIH	QUAD	4
		Q	TRIA	6
Plane Strain composite	COMPS	L	QUAD	4
Axisymmetric Solid	AXSOLID	L	QUAD	4
		L	TRIA	3
		LT	QUAD	4
		Q	QUAD	8
		QRI	QUAD	8
		QT	QUAD	8
		LRIH	QUAD	4
		Q	TRIA	6
Axisymmetric Composite	AXCOMP	L	QUAD	4
Incompressible Plane Strain	IPS	L	TRIA	3
Incompressible Axisymmetric	IAX	L	TRIA	3

Only BEH4 = COMPS or AXCOMP with INT4 = L may use the MATG, additionally they should not use a MAT1, MAT2, MAT3, MAT8, MATORT, or MATHE as they will suffer from hourglassing.

- Note for this entry with a BEHi=COMPS or AXCOMP, the THETA/MCID value on the element connection entry will be ignored.
- In the table below, BEHiH refers to the nonlinear heat behavior of 2D-solid elements. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
Planar Solid	PLSTRN	L	QUAD	4
		L	TRIA	3
		Q	QUAD	8
		Q	TRIA	6
		Q	TRIA	6
Axisymmetric Solid	AXSOLID	L	QUAD	4
		L	TRIA	3
		Q	QUAD	8
		Q	QUAD	8
		Q	TRIA	6



10. Integration codes in Remarks 7. and 9. are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass Control (assumed strain)
Q	Quadratic
QRI	Quadratic Reduced Integration
QT	Quadratic with Twist
LT	Linear with Twist

11. For creep material defined through MATVP, VALC=0 must be set on NLMOPTS, for explicit formulation.
12. For shape memory materials defined through MATSMA, only the thermo-mechanical model is available.
13. The structural element damping coefficient, GE, is not supported on elements which reference PSHLN2.

## PSHLPF

## Property of Perforated Shell Element

Specifies properties for perforated shell of trim component, TRMC, in porous elastic material.

### Format:

1	2	3	4	5	6	7	8	9	10
PSHLPF	PID	MID	T	SPACEG	RADIUS	TOPOLOGY	FRHO	FVIS	
	HOMG								

### Example:

PSHLPF	1	2	8.1-4	1.132-2	1.245-3	SQUARE	1.225	1.71-5	
--------	---	---	-------	---------	---------	--------	-------	--------	--

Describer	Meaning
MID	Identification number of MAT1. (Integer>0)
T	Thickness. (Real>0.0)
SPACEG	Spacing of perforation holes (Real>0.0)
RADIUS	Radius of perforation holes (Real>0.0)
TOPOLOGY	Hole pattern: SQUARE, TRIA or HEXA
FRHO	Fluid density (Real>0.0)
FVIS	Fluid viscosity (Real>0.0)
HOMG	Homogenization flag (Integer=0 or 1; default=1)

### Remarks:

1. PSHLPF should be used for PEM modelling under 'BEGIN TRMC=trmid' only.
2. Finer mesh of the holes or around the holes should be avoided. The computation of an equivalent transfer admittance allows to reduce meshing effort and CPU time.
3. With default HOMG, stiffness and density are varying with porosity. With HOMG set to 0, stiffness provided is utilized as is.

**PSLDN1****Nonlinear Property Extensions for a PSOLID Entry**

Specifies additional nonlinear properties for solid elements that point to a PSOLID entry in SOL 400.

**Format:**

1	2	3	4	5	6	7	8	9	10
PSLDN1	PID	MID	DIRECT		ANAL				
	"C4"	BEH4	INT4	BEH4H	INT4H				
	"C5"	BEH5	INT5	BEH5H	INT5H				
	"C6"	BEH6	INT6	BEH6H	INT6H				
	"C8"	BEH8	INT8	BEH8H	INT8H				
	"C10"	BEH10	INT10	BEH10H	INT10H				
	"C13"	BEH13	INT13	BEH13H	INT13H				
	"C15"	BEH15	INT15	BEH15H	INT15H				
	"C20"	BEH20	INT20	BEH20H	INT20H				

**Example:**

PSLDN1	22	55	2						
--------	----	----	---	--	--	--	--	--	--

Describer	Meaning
PID	Property identification number of an existing PSOLID entry. (Integer > 0)
MID	Material identification number. Remark 8.(Integer ≥ 0)
DIRECT	The layer direction for BEHi=SLCOMP. See Remark 5. (Integer, 1, 2, OR 3; Default = 1)
ANAL	Analysis type. ANAL = 'IS' - Implicit structural elements are being referred to. ANAL = 'IH' - Implicit heat analysis elements are being referred to. ANAL = 'ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C4	Keyword indicating that two items following apply to elements with four corner grids. (Character)
C5	Keyword indicating that two items following apply to elements with five corner grids. (Character)
C8	Keyword indicating that two items following apply to elements with eight corner grids. (Character)
C10	Keyword indicating that two items following apply to elements with four corner grids and six midside grids. (Character)
C13	Keyword indicating that two items following apply to elements with five corner grids and eight midside grids. (Character)
C15	Keyword indicating that items following apply to elements with six corner and nine midside grids. (Character)

Describer	Meaning
C20	Keyword indicating that two items following apply to elements with eight corner grids and twelve midside grids. (Character)
BEHi	Element structural behavior. See Remark 6. (Character default: SOLID for BEH4, BEH6, BEH8, BEH10, BEH15, and BEH20)
INTi	Integration scheme. See Remark 7. (Character default: L for INT4, INT6, and INT8; Q for INT10, INT15, and INT20)
BEHiH	Element heat behavior. See Remark 6. (Character Default: SOLID for BEH4H, BEH6H, BEH8H, BEH10H, BEH15, and BEH20H)
INTiH	Integration scheme. See Remark 7. (Character Default: L for INT4H, INT6H, and INT8H; Q for INT10H, INT15H, and INT20H)

**Remarks:**

1. The PID must point to an existing PSOLID Bulk Data entry and is honored only in SOL 400.
2. The MID entry may point to MAT1, MAT9, MATORT, MATHE, or MATG entries and can be used to override the MID field on a PSOLID entry. The following table shows associated nonlinear entries. The association is established through the material entries having the same values as the MID entry.

Caution: The primary MATORT, MATHE, and MATG entries and the secondary MATVE, MATVP, MATEP and MATF entries are only associated with a 3D Solid (CHEXA, CPENTA, CPYRAM and CTETRA) element if the Solid element refers to a PSLDN1 entry.

The MID entry for nonlinear heat may point to MAT4 or MAT5 entries.

Implicit Structure Materials					
MAT1	MAT9	MATORT	MATHE	MATG	MATHP
MATVE	<MATVE>	<MATVE>	MATVE		
MATVP	MATVP	MATVP			
MATEP	MATEP	MATEP			
MATF	MATF	MATF			
MATS1		MATSORT			
<MATVE> refers to the ALTERNATE format for type ORTHO MATG for BEH8=SLCOMP, INT8=L only. MATG has an associated field IDMEM that points to a MAT1 BEH4 = ISOL, INT4 = L is currently limited to MAT1 (with possible association with MATS1 or MATEP), MATHE and MATHP					

Heat Materials	
MAT4	MAT5



If heat analysis is being performed and the user wishes to override standard Nastran heat elements, the ANAL entry must be set to IH or ISH.

If ISH is specified then the MAT1 and MAT4 or MAT1 and MAT5 must have the same ID. MID for structure entries must follow the uniqueness rules of the MAT1, MAT2, MAT3, MAT8, MAT9, MATORT, MATHP, MATHE, and MATG entries.

MID for heat entries must follow the uniqueness rules of the MAT4 and MAT5 entries.

- The keyword entries may occur in any order or not at all. If a keyword entry is missing, its defaults are assumed.
- The following table describes layer orientation for BEH8=SLCOMP, INT8=L.

Layer Orientation			
DIRECT	Normal to Layer plane	Layers run parallel from face	to face
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8

- In the following table, BEHi refers to the nonlinear structural behavior of the solid element. An underlined item delineates a default.

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
SOLID	SOLID	L	HEX	8
		<u>Q</u>	HEX	20
		QRI	HEX	20
		LRIH	HEX	8
		<u>Q</u>	TET	10
		LRIH	TET	10
		L	TET	4
		<u>L</u>	PEN	6
		<u>Q</u>	PEN	15
		L	PYR	5
		<u>LRIH</u>	PYR	5
		<u>Q</u>	PYR	13
		3D Solid Gasket	SLCOMP	L

Structural Classification of Elements				
Element Structural Type	BEHi CODE	Integration Code	Element Type	# Nodes
Solid Shell	SLCOMP	ASTN*	HEX	8
Incompressible solid	ISOL	L	TET	4
Only BEH8=SLCOMP,INT=L may use a MATG, additionally it should not use a MAT1, MAT9, MATORT, or MATHE as it will suffer from hour-glassing. It is recommended that for BEH4 = ISOL, INT4 = L that NLMOPTS,LRGSTRN,2 be flagged				
*Only DIRECT=1 is allowed.				

6. In the following table, BEHiH refers to the heat behavior of the solid element. An underlined item delineates a default.

Heat Classification of Elements				
Element Heat Type	BEHiH CODE	Integration Code	Element Type	# Nodes
SOLID	SOLID	<u>L</u>	HEX	8
		<u>Q</u>	HEX	20
		<u>Q</u>	TET	10
		<u>L</u>	TET	4
		<u>L</u>	PEN	6
		<u>Q</u>	PEN	15
		<u>L</u>	PYR	5
		<u>Q</u>	PYR	13

7. Integration codes in Remark 5 are:

INT CODE	Integration Type
L	Linear
LRIH	Linear Reduced Integration Hourglass control (assumed strain)
ASTN	Assumed STraiN enhanced formulation solid shell
Q	Quadratic
QRI	Quadratic Reduced Integration

8. Structural damping is supported for elements which reference PSLDN1.
- The structural damping coefficient, GE, is supported for elements using the MAT1 or MAT9 entries.
  - Extended damping coefficients GEij are supported for elements using the MAT9 entries.

## PSOLID Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, CPYRAM and CTETRA entries).

### Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN		

### Example:

PSOLID	2	100	6	TWO	GRID	REDUCED	PORO		
--------	---	-----	---	-----	------	---------	------	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, MAT10, MATHP, MATPE1, or MATHE entry. (Integer > 0)
CORDM	Identification number of the material coordinate system. See Remarks 3. and 4. (Integer; Default = 0, which is the basic coordinate system; see Remark 3.)
IN	Integration network. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
STRESS	Location selection for stress output. See Remarks 9. and 10. (Integer; Character, or blank)
ISOP	Integration scheme. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
FCTN	Fluid element flag. (Character: “FFLUID” indicates a fluid element with frequency dependent rigid absorber properties, “PFLUID” indicates a fluid element, “SMECH” indicates a structural element, “PORO” indicates an element of poroelastic medium, see Remark 12. , “PSLDSHL” indicates an element of solid shell of poroelastic medium, see Remark 13; Default = “SMECH.”)

### Remarks:

1. PSOLID is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PSOLID property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PSOLID, PCOMPLS entries.
2. Isotropic (MAT1 or MAT4), anisotropic (MAT5 or MAT9), or fluid (MAT10) material properties may be referenced. If FCTN = “PFLUID” or “FFLUID”, then MID must reference a MAT10 entry. PFLUID and FFLUID are not available for SOL 600 and SOL 700.

3. See the CHEXA, CPENTA, or CTETRA entry for the definition of the element coordinate systems. The material coordinate system (CORDM) may be the basic system (0 or blank), any defined system (Integer > 0), or the standard internal coordinate system of the element based on eigenvalue techniques to insure non bias in the element formulation designated as element coordinate system (-1) or the element system (-2). The default value for CORDM is zero unless it is overridden by the NASTRAN statement with the CORDM keyword. When MAT9 entry is used, the coordinate system used effects the stiffness calculation of the element.
4. For the CPYRAM entry the element coordinate system is the same as the basic coordinate system. CORDM values of 0, -1, -2 or blank are treated to be identical to the basic coordinate system.
5. Advanced nonlinear solid elements identified through an additional PSLDN1 entry, or SPROPMAP > -1 in NLMOPTS, or IDAMP>0 in NLSTEP, and so on, in SOL 400 do not support: IN, ISOP, and FCTN. Also, in this case, CORDM=-1 is only supported for CHEXA elements. Fatal error is issued if CORDM < 0 is used for CPENTA and CTETRA.

**The following Remarks, DO NOT APPLY TO SOL 600 or SOL 700.**

6. For CHEXA and CPENTA elements with no midside nodes, reduced shear integration with bubble functions (ISOP = blank or "REDUCED" and IN = blank or "BUBBLE") is the default. This is recommended because it minimizes shear locking and Poisson's ratio locking and does not cause modes of deformation that lead to no strain energy. The effects of using nondefault values are as follows:
  - a. IN = "THREE" or 3 produces an overly stiff element.
  - b. If IN = "TWO" or 2 and the element has midside nodes, modes of deformation may occur that lead to no strain energy.
  - c. Standard isoparametric integration (ISOP = "FULL" or 1 and IN = "TWO" or 2; or "THREE" or 3) produces an element overly stiff in shear. This type of integration is more suited to nonstructural problems.
  - d. In SOL105 the default reduced integration scheme often produces spurious modes in which case it is recommended that ISOP="FULL" and IN="THREE" be used.
7. IN = "BUBBLE" is not allowed for CTETRA elements or for CHEXA and CPENTA elements with midside nodes.
8. If you use IN = "BUBBLE" for CTETRA elements, NASTRAN internally switch to IN=2 if you have 4-noded CTETRA element and IN=3 greater than 4 nodes.
9. Stress output may be requested at the Gauss points (STRESS = "GAUSS" or 1) of linear CHEXA and CPENTA elements (i.e. elements with no midside nodes). Gauss point output is available for the linear and quadratic CTETRA and CPYRAM elements (i.e. elements with or without midside nodes).
10. The following tables indicate the allowed options and combination of options. If a combination not found in the table is used, then a warning message will be issued and default values will be assigned for all options.
11. The gauss point locations for the solid elements are documented in [Nonlinear Analysis, 535](#) in the *MSC Nastran Reference Guide*.



12. The entries with FCTN="PORO" must select MATPE1 identification number in MID field.

Table 9-35 CHEXA Entry Options

CHEXA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability				
					SOLs 106, 129, 400	SOL 600	SOL 700		
8 Node	2x2x2 Reduced Shear with BubbleFunction (default)	BUBBLE or Blank or 0 (Default)	Blank or GRID or GAUSS or 1	Blank or REDUCED (Default*)	Yes	Yes**	Yes		
	2x2x2 Reduced Shear Only	TWO or 2							
	2x2x2 Standard Isoparametric							FULL or 1	Yes
	3x3x3 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	Yes	Yes		
	3x3x3 Standard Isoparametric								FULL or 1
9--20 Node	2x2x2 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes	No		
	2x2x2 Standard Isoparametric								FULL or 1
	3x3x3 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)				No	No
	3 x3x3 Standard Isoparametric								
*REDUCED is the default only for structural elements (FCTN="SMECH").									
** Requires PARAM,MRALIAS									

Table 9-36 CPENTA Entry Options

CPENTA	Integration	IN	STRESS (Default: GRID)	ISOP (Default: See Remarks 5 and 7.)	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
6 Node	2x3 Reduced Shear with Bubble Function (Default)	Blank or 0 or BUBBLE (Default)	GAUSS or 1 or Blank or GRID	Blank or REDUCED (Default*)	Yes	Yes**	Yes
	2x3 Reduced Shear Only	TWO or 2					
	2x3 Standard Isoparametric						
	3x7 Reduced Shear Only	THREE or 3	Blank or GRID	Blank or REDUCED	No	No	No
	3x7 Standard Isoparametric			FULL or 1		No	No
7-15 Node	2x3 Reduced Shear Only	TWO or 2	Blank or GRID	Blank or REDUCED	No	Yes**	No
	2x3 Standard Isoparametric			FULL or 1		Yes	No
	3x7 Reduced Shear Only (default)	Blank or THREE or 3 (Default)		Blank or REDUCED (Default*)		No	No
	3x7 Standard Isoparametric			FULL or 1		No	No
** Requires PARAM,MRALIAS							
*REDUCED is the default only for structural elements (FCTN="SMECH").							

Table 9-37 CTETRA Entry Options

CTETRA	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOLs 106, 129, 400	SOL 600	SOL 700
4 Node	1-Point Standard Isoparametric (Default)	Blank or TWO or 2 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes	Yes**	Yes
	5-Point Standard Isoparametric	THREE or 3	Blank or GRID		No	Yes	No
5-10 Node	5-Point Standard Isoparametric	Blank or THREE or 3 (Default)	GAUSS or 1 or Blank or GRID	Blank or FULL	Yes	Yes	No
**Requires PARAM,MRALIAS							

Table 9-38 CPYRAM Entry Options

CPYRAM	Integration	IN	STRESS (Default: GRID)	ISOP	Nonlinear Capability		
					SOL 400	SOL 600	SOL 700
5 Node	5-Point Standard Isoparametric (Default)	Not applicable	GAUSS or 1 or Blank or GRID	Not applicable	Yes	No	No
	5-Point Standard Isoparametric	Not applicable	Blank or GRID		Yes	No	No
13 Node	8-Point Standard Isoparametric	Not applicable	GAUSS or 1 or Blank or GRID	Not applicable	Yes	No	No

- PSOLID entry with FCTN=PSLDSHL must select MAT1 identification number in MID field and should be used for trim component under 'BEGIN TRMC=trmid' only.

## PSPRMAT

## Spring and damper elements for translation and rotation - SOL 700

Defines linear and nonlinear spring element property for translation. Used in SOL 700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PSPRMAT	PID								
	K	FAILMTF	FAILMCF	LOAD	UNLOAD				

### Example:

PSPRMAT	1								
	20.0E3								

Descriptor	Meaning
PID	Property ID. PID is referenced on the CSPR entry. (Integer > 0; required.)
K	Elastic stiffness (force/displacement). (Real > 0; See Remarks 1. and 2.)
FAILMTF	Tensile failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)
FAILMCF	Compressive failure force. See Remarks 1. and 2. (Real > 0; default=Not failure)
LOAD	Number of a TABLED1 entry defining the variation of force (y-value) with displacement (x-value) during loading. (Integer > 0; See Remarks 1., 3. and 4.)
UNLOAD	Number of a TABLExx entry defining the variation of force (y-value) with displacement (x-value) during unloading. (Integer > 0; See Remarks 3. and 4.)

### Remarks:

1. Only one set of K, FAILMTF and FAILMCF or LOAD and UNLOAD is allowed. If K is defined, LOAD and UNLOAD are not allowed. If LOAD is defined, K, FAILMTF and FAILMCF are not allowed.
2. For linear spring, K is required and FAILMTF and FAILMCF are optional.
3. For nonlinear spring, LOAD is required and UNLOAD is optional. If UNLOAD is not defined, unloading path is identical as LOAD.
4. Input for loading and unloading must be consistent. Both curves must be either completed defined or have only positive values (start from (0.,0.)). When only positive values are defined, the curves are automatically mirrored.

## PSSHL Properties for Solid Shell (CSSHL) Elements in SOL 600

Defines the properties for Solid Shell (CSSHL) elements in SOL 600.

### Format:

1	2	3	4	5	6	7	8	9	10
PSSHL	PID	MID	IT	SF	MCID				

### Example:

PSSHL	11	33		.8333					
-------	----	----	--	-------	--	--	--	--	--

Describer	Meaning
PID	Property identification number. (Integer > 0; Required)
MID	Identification of a MAT <sub>xxx</sub> entry. All MAT entries available in SOL 600 can be specified except for hyperelastic materials. (Integer > 0)
IT	Transition thickness - Enter only if a solid shell is attached to a standard shell (such as CQUAD4), in which case TT is the thickness of the standard shell. (Real; Default = 0.0)
SF	Transverse shear factor - Leave blank if transverse shear is not to be considered. (Real or blank, if entered SF must range between 0.0 and 1.0)
CORDM	Identification number of a material coordinate system. See Remark 3. (Integer; Default = 0)

### Remarks:

1. PSSHL entries should have unique identification numbers with respect to all other property entries.
2. MID may reference isotropic, orthotropic or anisotropic materials with or with plasticity, however hyperelastic and foam materials are not available.
3. See the CHEXA entry for the definition of the element coordinate system. The material coordinate system (CORDM) may be the basic system (0 or blank) or any defined system (Integer > 0), or the element coordinate system (-1). The default value for CORDM is zero.

**PTUBE**

## Tube Property

Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).

**Format:**

1	2	3	4	5	6	7	8	9	10
PTUBE	PID	MID	OD	T	NSM	OD2			

**Example:**

PTUBE	2	6	6.29	0.25					
-------	---	---	------	------	--	--	--	--	--

**Describer****Meaning**

PID	Property identification number. (Integer > 0)
MID	Material identification number. See Remarks 3. and 4. (Integer > 0)
OD	Outside diameter of tube. (Real > 0.0)
T	Thickness of tube. (Real; $T \leq OD/2.0$ )
NSM	Nonstructural mass per unit length. (Real)
OD2	Diameter of tube at second grid point (G2) on CTUBE entry. (Real; Default = OD)

**Remarks:**

1. If T is zero, a solid circular rod is assumed.
2. PTUBE is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PTUBE property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PTUBE entries.
3. For structural problems, MID must reference a MAT1 material entry.
4. For heat transfer problems, MID must reference a MAT4 or MAT5 material entry.
5. Tapered OD tubes with constant wall thickness are available for heat transfer only. The effective diameter is given by:

$$D_{effective} = T + \frac{D_2 - D_1}{\log_e \left( \frac{D_2 - T}{D_1 - T} \right)}$$

where:

$$D_1 = \text{OD}$$

$$D_2 = \begin{cases} \text{OD2 if OD2} \neq 0 \\ \text{OD if OD2} = 0 \text{ or blank} \end{cases}$$

## **PVISC** Viscous Damping Element Property

Defines properties of a one-dimensional viscous damping element (CVISC entry).

### **Format:**

1	2	3	4	5	6	7	8	9	10
PVISC	PID1	CE1	CR1		PID2	CE2	CR2		

### **Example:**

PVISC	3	6.2	3.94						
-------	---	-----	------	--	--	--	--	--	--

<b>Describer</b>	<b>Meaning</b>
PID <sub>i</sub>	Property identification number. (Integer > 0)
CE1, CE2	Viscous damping values for extension in units of force per unit velocity. (Real)
CR1, CR2	Viscous damping values for rotation in units of moment per unit velocity. (Real)

### **Remarks:**

1. Viscous properties are material independent; in particular, they are temperature independent.
2. One or two viscous element properties may be defined on a single entry.
3. PVISC is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PVISC property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PVISC entries.



## PVISC1

## Nonlinear Damper Property- SOL 700

Defines the properties of a nonlinear damper where the damping constant varies with the velocity. Used in SOL700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
PVISC1	PID	TABLE							

### Example:

PVISC1	8	236							
--------	---	-----	--	--	--	--	--	--	--

Field	Content
PID	Unique property number. (Integer > 0; required)
TABLE	TABLED1 ID defining the variation of the force (y-value) with velocity (x-value). (Integer > 0; required)

### Remarks:

1. This entry defines the properties of a nonlinear damper. Use the PVISC entry to define linear dampers.
2. The values in the table are interpolated to get the force for a particular velocity.

## PWELD Connector Element Property

Defines the property of connector (CWELD) elements.

### Format:

1	2	3	4	5	6	7	8	9	10
PWELD	PID	MID	D			MSET		TYPE	
	LDMIN	LDMAX							

### Example:

PWELD	100	3	1.0						
-------	-----	---	-----	--	--	--	--	--	--

Describer	Meaning	Type	Default
PID	Property identification number.	Integer > 0	Required
MID	Material identification number. See Remark 1.	Integer > 0	Required
D	Diameter of the connector. See Remark 1.	Real > 0	Required
MSET	Active ONLY for "PARAM,OLDWELD,YES". Flag to eliminate m-set degrees-of-freedom (DOFs). The MSET parameter has no effect in a nonlinear SOL 400 analysis.  =OFF m-set DOFs are eliminated, constraints are incorporated in the stiffness, see Remark 2. =ON m-set DOFs are not eliminated, constraints are generated.	Character	OFF
TYPE	Character string indicating the type of connection, see Remarks 3. and 4.  =blank general connector ="SPOT" spot weld connector	Character	Blank
LDMIN	Active ONLY for "PARAM,OLDWELD,YES". Smallest ratio of length to diameter for stiffness calculation, see Remark 4.	Real or blank	0.2
LDMAX	Active ONLY for "PARAM,OLDWELD,YES". Largest ratio of length to diameter for stiffness calculation, see Remark 4.	Real or blank	5.0

**Remarks:**

1. The material MID, the diameter D, and the length are used to calculate the stiffness of the connector in 6 directions. MID can only refer to the MAT1 Bulk Data entry. The length is the distance of GA to GB (see [Figure 9-138](#)).

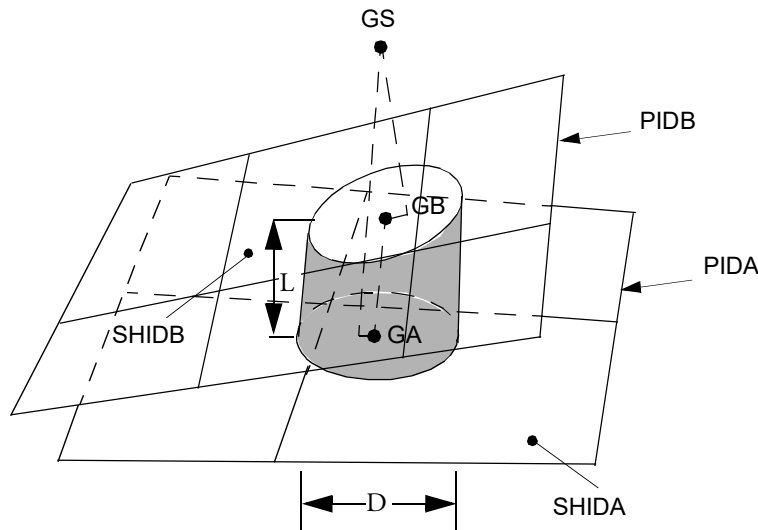


Figure 9-138 Length and Diameter of the CWELD Connector

2. This remark is now valid only if "PARAM,OLDWELD,YES" is present. The parameter MSET is active only for the formats ELEMID and GRIDID (see [CWELD, 1685](#) for the format descriptions). MSET = "OFF" incorporates constraints at the element stiffness matrix level avoiding explicit m-set constraint equations. For the formats PARTPAT and ELPAT, constraints are always eliminated on the element level. MSET = "ON" generates explicit m-set constraints. For example, if a patch-to-patch connection is specified with the formats "GRIDID" or "ELEMID" on the CWELD entry, and MSET=ON is specified, 2x6 explicit constraints are generated that connect the 6 degrees-of-freedom of GA to the translational degrees-of-freedom of grid points GA<sub>i</sub> and the 6 degrees-of-freedom of GB to GB<sub>i</sub>. The 2x6 degrees-of-freedom of GA and GB are put into the m-set. The constraints are labeled "RWELD". The identification numbers of the generated RWELD constraint elements start with an offset of 100,001,001 by default. The offset number can be changed with PARAM, OSWELM. For MSET = "OFF" or blank, the 2x6 constraint equations are built into the stiffness matrix of the CWELD element, thereby condensing the 2x6 degrees-of-freedom of GA and GB.
3. TYPE = "SPOT" is good for the formats PARTPAT, ELPAT, or ELEMID with patch to patch connections. For point to patch connections or another formats of patch to patch connections, TYPE is always set to blank.

If "PARAM,OLDWELD,YES" is present, the effective length for the stiffness of the CWELD element is set to  $L_e = 1/2 \cdot (t_A + t_B)$  regardless of the distance GA to GB.  $t_A$  and  $t_B$  are the shell thicknesses of shell A and B, respectively. The effective length is used to avoid excessively stiff or soft connections due to mesh irregularities.

For the new CWELD, the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance GA to GB is equal to  $L = 1/2 \cdot (t_A + t_B)$ .

4. If TYPE=blank and "PARAM,OLDWELD,YES" is present, the effective length  $L_e$  of the CWELD is equal to the true length L, the distance of GA to GB, as long as the ratio of the length L to diameter D is in the range  $L_{DMIN} \leq L/D \leq L_{DMAX}$ . If L is below the range, the effective length is set to  $L_e = L_{DMIN} \cdot D$  and if L is above the range, the effective length is set to  $L_e = L_{DMAX} \cdot D$ .

For the new CWELD with TYPE=blank, LDMIN and LDMAX are ignored. If the distance of GA to GB (patch to patch connections) or GS to GA (point to patch connections) is less than  $1.0e-4$ , the locations of GA/GB (if GA/GB is not specified by the user or the SWLDPRM parameter MOVGAB is equal to 1) will be adjusted so that the distance is equal to  $1.0e-4$ .

5. PWELD is a primary property entry. Primary property entries are grouping entities for many applications in MSC Nastran. Therefore it is highly recommended that the PWELD property entries have unique identification numbers with respect to all other property entries else unexpected grouping results may occur. There must be uniqueness between PWELD entries.
6. The new CWELD will not contribute to MASS by default even if it's associated MATi entry has a non-zero density. To react to a non-zero density "SWLDPARM,WMASS,1" is required. If mass is computed, the PARAM,COUPMASS effects the mass calculation.

## Entries Q-S

### QBDY1

### Boundary Heat Flux Load for CHBDYj Elements, Form 1

Defines a uniform heat flux into CHBDYj elements.

#### Format:

1	2	3	4	5	6	7	8	9	10
QBDY1	SID	Q0	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	EID8	-etc.-						

#### Example:

QBDY1	109	1.-5	721						
-------	-----	------	-----	--	--	--	--	--	--

#### Alternate Format and Example:

QBDY1	SID	Q0	EID1	“THRU”	EID2				
QBDY1	109	1.-5	725	THRU	735				

Describer	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Heat flux into element. (Real)
EIDi	CHBDYj element identification numbers. (Integer ≠ 0 or “THRU”. For “THRU” option EID2 > EID1.)

#### Remarks:

1. QBDY1 entries must be selected with the Case Control command LOAD = SID in order to be used in static analysis. The total power into an element is given by the equation:

$$P_{in} = (\text{Effective area}) \cdot Q0$$

2. QBDY1 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. The total power into an element is given by the equation:

$$P_{in}(t) = (\text{Effective area}) \cdot Q0 \cdot F(t - \tau)$$

where the function of time  $F(t - \tau)$  is specified on a TLOADi entry.

3. The sign convention for Q0 is positive for heat input.
4. RC network solver does not support QBDY1 for thermal analysis.

## QBDY2

## Boundary Heat Flux Load for CHBDYj Elements, Form 2

Defines grid point heat flux into CHBDYj elements.

### Format:

1	2	3	4	5	6	7	8	9	10
QBDY2	SID	EID	Q01	Q02	Q03	Q04	Q05	Q06	
	Q07	Q08							

### Example:

QBDY2	109	721	1.-5	1.-5	2.-5	2.-5			
-------	-----	-----	------	------	------	------	--	--	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
EID	Identification number of an CHBDYj element. (Integer > 0)
Q0i	Heat flux at the i-th grid point on the referenced CHBDYj element. (Real or blank)

### Remarks:

1. QBDY2 entries must be selected with the Case Control command LOAD=SID in order to be used in static analysis. The total power into each point  $i$  on an element is given by

$$P_i = AREA_i \cdot Q0i$$

2. QBDY2 entries must be referenced on a TLOADi Bulk Data entry through the EXCITEID specification for use in transient analysis. All connected grid points will have the same time function but may have individual delays. The total power into each point  $i$  on an element is given by

$$P_i(t) = AREA_i \cdot Q0i \cdot F(t - \tau_i)$$

where  $F(t - \tau_i)$  is a function of time specified on a TLOADi entry.

3. The sign convention for Q0i is positive for heat flux input to the element.
4. RC network solver does not support QBDY2 for thermal analysis.

## QBDY3 Boundary Heat Flux Load for a Surface

Defines a uniform heat flux load for a boundary surface.

### Format:

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

### Example:

QBDY3	2	20.0	10	1	THRU	50	BY	2	
-------	---	------	----	---	------	----	----	---	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real)
CNTRLND	Control point for thermal flux load. (Integer ≥ 0; Default = 0)
EIDi	CHBDYj element identification numbers. (Integer ≠ 0 or “THRU” or “BY”)

### Remarks:

1. QBDY3 entries must be selected in Case Control (LOAD = SID) to be used in steady state. The total power into a surface is given by the equation:
  - if  $CNTRLND \leq 0$  then  $P_{in} = (\text{Effective area}) \cdot Q0$
  - if  $CNTRLND > 0$  then  $P_{in} = (\text{Effective area}) \cdot Q0 \cdot u_{CNTRLND}$

where  $u_{CNTRLND}$  is the temperature of the control point and is used as a load multiplier.
2. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi multiplies the general load, with  $\tau$  specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
3. The CNTRLND multiplier cannot be used with any higher-order elements.
4. When using “THRU” or “BY”, all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.
5. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

## QHBDY Boundary Heat Flux Load

Defines a uniform heat flux into a set of grid points.

### Format:

1	2	3	4	5	6	7	8	9	10
QHBDY	SID	FLAG	Q0	AF	G1	G2	G3	G4	
	G5	G6	G7	G8					

### Example:

QHBDY	2	AREA4	20.0		101	102	104	103	
-------	---	-------	------	--	-----	-----	-----	-----	--

Describer	Meaning
SID	Load set identification number. (Integer > 0)
FLAG	Type of face involved (must be one of the following: "POINT", "LINE", "REV", "AREA3", "AREA4", "AREA6", "AREA8")
Q0	Magnitude of thermal flux into face. Q0 is positive for heat into the surface. (Real)
AF	Area factor depends on type. (Real > 0.0 or blank)
Gi	Grid point identification of connected grid points. (Integer > 0 or blank)

### Remarks:

- The continuation entry is optional.
- For use in steady state analysis, the load set is selected in the Case Control Section (LOAD = SID).
- In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
- The heat flux applied to the area is transformed to loads on the points. These points need not correspond to an HBDY surface element.
- The flux is applied to each point  $i$  by the equation
 
$$P_i = AREA_i \cdot Q0$$
- The number of connected points for the types are 1 (POINT), 2 (LINE, REV), 3 (AREA3), 4 (AREA4), 4-6 (AREA6), 5-8 (AREA8).



7. The area factor *AF* is used to determine the effective area for the POINT and LINE types. It equals the area and effective width, respectively. For axisymmetric elements, the magnitude of the area factors should account for an integration over the entire circumference (i.e.,  $2\pi$  radians). Note that the REV option can be used to apply the heat flux along the edges of axisymmetric elements. *AF* is not used for the other types, which have their area defined implicitly and must be left blank.
8. The type of face (FLAG) defines a surface in the same manner as the CHBDYi Bulk Data entry. For physical descriptions of the geometry involved, see the CHBDYG discussion.

## QSET Generalized Degree-of-Freedom

Defines generalized degrees-of-freedom (q-set) to be used for dynamic reduction or component mode synthesis.

### Format:

1	2	3	4	5	6	7	8	9	10
QSET	ID1	C1	ID2	C2	ID3	C3	ID4	C4	

### Example:

QSET	15	123456	1	3	9	2	105	6	
------	----	--------	---	---	---	---	-----	---	--

Describer	Meaning
ID <sub>i</sub>	Grid or scalar point identification number. (Integer > 0)
C <sub>i</sub>	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)

### Remarks:

- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
- Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
- When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPC<sub>i</sub> or MPC entries) will be placed in the omitted set (o-set).
- If Modules are present then this entry may only be specified in the main Bulk Data section.
- The SUPORT entry is not allowed in SOLs 106 and 129. SOL 400 does not support ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, SUPORT<sub>i</sub>, and QSET<sub>i</sub> except in the following situations:
  - Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - Linear perturbation with:
    - EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - Superelements defined with BEGIN SUPER may contain ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, and QSET<sub>i</sub> entries.

## QSET1 Generalized Degree-of-Freedom (Alternate Form of QSET Entry)

Defines generalized degrees-of-freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.

### Format:

1	2	3	4	5	6	7	8	9	10
QSET1	C	ID1	ID2	ID3	ID4	ID5	ID6	ID7	
	ID8	ID9	-etc.-						

### Example:

QSET1	123456	1	7	9	22	105	6	22	
	52	53							

### Alternate Format and Example:

QSET1	C	ID1	“THRU”	ID2					
QSET1	0	101	THRU	110					

Describer	Meaning
C	Component number. (Integer zero or blank for scalar points or any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.)
IDi	Grid or scalar point identification number. (Integer > 0; For THRU option, ID1 < ID2.)

### Remarks:

1. Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
2. Degrees-of-freedom specified on QSET and QSET1 entries are automatically placed in the a-set.
3. When ASET, ASET1, QSET, and/or QSET1 entries are present, all degrees-of-freedom not otherwise constrained (e.g., SPCi or MPC entries) will be placed in the omitted set (o-set).
4. If Modules are present then this entry may only be specified in the main Bulk Data section.
5. The SUPORT entry is not allowed in SOLs 106 and 129. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:



- i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
- ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
- c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.

## QVECT

## Thermal Vector Flux Load

Defines thermal vector flux from a distant source into a face of one or more CHBDYi boundary condition surface elements.

### Format:

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc.-						

### Example:

QVECT	10	20.0	1000.0		1.0	1.0	1.0	101	
	20	21	22	23					

Describer	Meaning
SID	Load set identification number. (Integer > 0)
Q0	Magnitude of thermal flux vector into face. (Real or blank)
TSOUR	Temperature of the radiant source. (Real or blank)
CE	Coordinate system identification number for thermal vector flux. See Remark 9. (Integer ≥ -1 or blank)
Ei	Vector components (direction cosines in coordinate system CE) of the thermal vector flux. (Real; Default = 0.0)
TIDi	TABLEDi entry identification numbers defining the components as a function of time. (Integer > 0)
CNTRLND	Control point. (Integer ≥ 0; Default = 0)
EIDi	Element identification number of a CHBDYE, CHBDYG, or CHBDYP entry. (Integer > 0) Key word "THRU" can be used to assist the listing with ascending order of EIDi

### Remarks:

1. The continuation entry is required.
2. If the coordinate system CE is not rectangular, then the thermal vector flux is in different directions for different CHBDYi elements. The direction of the thermal vector flux over an element is aligned to be in the direction of the flux vector at the geometric center of the element. The geometric center is measured using the grid points and includes any DISLIN specification on the VIEW entry for TYPE=LINE CHBDYi elements. The flux is presumed to be uniform over the face of each element; i.e., the source is relatively distant.

3. For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID). The total power into an element is given by:

- If CNTRLND = 0 then,  $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0$ .
- If CNTRLND > 0 then,  $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0 \cdot u_{CNTRLND}$ .

where

$\alpha$  = face absorptivity (supplied from a RADM statement).

A = face area as determined from a CHBDYi surface element.

$\vec{e}$  = vector of direction cosines E1, E2, E3.

$\vec{n}$  = face normal vector. See CHBDYi entries.

$\vec{e} \cdot \vec{n}$  = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

$u_{cntrlnd}$  = temperature value of the control point used as a load multiplier.

4. If the absorptivity is constant, its value is supplied by the ABSORP field on the RADM entry. If the absorptivity is not a constant, the thermal flux is assumed to have a wavelength distribution of a black body at the temperature TSOUR.

- For a temperature-dependent absorptivity, the element temperature is used to determine  $\alpha$ .
- For a wavelength-dependent absorptivity, the integration of the flux times  $\alpha$  is computed for each wavelength band. The sum of the integrated thermal fluxes over all the wavelength bands is Q0. The wave bands are specified with the RADBND entry.
- The user has the responsibility of enforcing Kirchhoff's laws.

5. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  provides any required time delay.  $F(t - \tau)$  is a function of time specified on the TLOADi entry. The value of  $F(t - \tau)$  is calculated for each loaded grid point. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.

The total power into an element is given by:

- If CNTRLND = 0 then,  $P_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot Q0 \cdot F(t - \tau)$ .
- If CNTRLND > 0 then,

$$P_{in} = -\alpha A(\vec{e}(t) \cdot \vec{n}) \cdot F(t - \tau) \cdot Q0 \cdot u_{CNTRLND}$$

6. If the referenced face is of TYPE = ELCYL, the power input is an exact integration over the area exposed to the thermal flux vector.
7. If the referenced face is of TYPE = REV, the thermal flux vector must be parallel to the axis of symmetry if an axisymmetric boundary condition is to be maintained.

8. When applied to a surface element associated with a radiation enclosure cavity, any incident energy that is not absorbed ( $\alpha < 1.0$ ) is lost from the system and is not accounted for in a reflective sense ( $\alpha + \rho = 1.0$ ).
9. If a heat flux normal to the surface is desired, set CE to -1. This allows a nondirectional temperature dependent heat load on the CHBDYi. the RADMT scale factor times Q0 equals to the total power. Remember that the absorptivity must fall between 0.0 and 1.0, (see the RADMT Bulk Data entry).
10. For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.

## QVOL Volume Heat Addition

Defines a rate of volumetric heat addition in a conduction element.

### Format:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

### Example:

QVOL	5	10.0	101	10	12	11	9		
------	---	------	-----	----	----	----	---	--	--

Describer	Meaning
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0; Default = 0)
EIDi	A list of heat conduction elements. (Integer > 0 or “THRU” or “BY”)

### Remarks:

- EIDi has material properties (MAT4) that include HGEN, the element material property for heat generation, which may be temperature dependent. This association is made through the element EID. If HGEN is temperature dependent, it is based on the average element temperature.
- QVOL provides either the constant volumetric heat generation rate or the load multiplier. QVOL is positive for heat generation. For steady-state analysis, the total power into an element is
  - If CNTRLND = 0, then  $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL}$ .
  - If CNTRLND > 0, then  $P_{in} = \text{volume} \cdot \text{HGEN} \cdot \text{QVOL} \cdot u_{\text{CNTRLND}}$ .

where  $u_{\text{CNTRLND}}$  is the temperature multiplier.

- For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID).
- In transient analysis SID is referenced by a TLOADi Bulk Data entry. A function of time  $F[t - \tau]$  defined on the TLOADi entry multiplies the general load where  $\tau$  specifies time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
- For “THRU” or “BY”, all intermediate referenced heat conduction elements must exist.
- The CNTRLND multiplier cannot be used with any higher-order elements.
- For RC network solver in thermal analysis, CNTRLND can only be used as the ID in CONTROLT as thermostats controller.



8. QVOL is not supported for CQUADX and CTRIAX axisymmetric elements. It is supported for the CTRIAX6 element.

## RADBC

## Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

### Format:

1	2	3	4	5	6	7	8	9	10
RADBC	NODAMB	FAMB	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	EID7	-etc.-						

### Example:

RADBC	5	1.0	101	10					
-------	---	-----	-----	----	--	--	--	--	--

Describer	Meaning
NODAMB	Ambient point for radiation exchange. (Integer > 0)
FAMB	Radiation view factor between the face and the ambient point. (Real ≥ 0.0)
CNTRLND	Control point for radiation boundary condition. (Integer ≥ 0; Default = 0)
EIDi	CHBDYi element identification number. (Integer > 0) Key words "THRU" and "BY" can be used to assist the listing with ascending order of EIDi

### Remarks:

- The basic exchange relationship is:
  - if CNTRLND = 0, then  $q = \sigma \cdot \text{FAMB} \cdot \epsilon_e \cdot (T_e^A - T_{amb}^A)$
  - if CNTRLND > 0, then
 
$$q = \sigma \cdot \text{FAMB} \cdot u_{\text{CNTRLND}} \cdot \epsilon_e \cdot (T_e^A - T_{amb}^A)$$
- NODAMB is treated as a black body with its own ambient temperature for radiation exchange between the surface element and space. No surface element that is a member of a radiation enclosure cavity may also have a radiation boundary condition applied to it.
- Two PARAM entries are required when stipulating radiation heat transfer:
  - ABS defines the absolute temperature scale; this value is added internally to any specified temperature given in the problem. Upon solution completion, this value is subtracted internally from the solution vector.
  - SIGMA ( $\sigma$ ) is the Stefan-Boltzmann constant.
- RADBC allows for surface radiation to space in the absence of any cavity behavior. The emissivity is supplied from a RADM entry.
- When using "THRU" or "BY", all intermediate referenced CHBDYi surface elements must exist.

## RADBND

## Radiation Wavelength Band Definition

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

### Format:

<b>1</b>	<b>2</b>	<b>3</b>	<b>4</b>	<b>5</b>	<b>6</b>	<b>7</b>	<b>8</b>	<b>9</b>	<b>10</b>
RADBND	NUMBER	PLANCK2	LAMBDA1	LAMBDA2	LAMBDA3	LAMBDA4	LAMBDA5	LAMBDA6	
	LAMBDA7	etc.							

### Example:

RADBND	6	14388.0	1.0	2.0	4.0	8.0	12.0		
--------	---	---------	-----	-----	-----	-----	------	--	--

Describer	Meaning
NUMBER	Number of radiation wave bands. See Remarks. (Integer > 1)
PLANCK2	Planck's second radiation constant. See Remarks. (Real > 0.0)
LAMBDAi	Highest wavelength of the i-th wave band. See Remarks. (Real ≥ 0.0)

### Remarks:

1. Only one RADBND entry may be specified in the Bulk Data Section and must always be used in conjunction with the RADM entry.
2. PLANCK2 has the units of wavelength times temperature. The same units of length must be used for LAMBDAi as for PLANCK2. The units of temperature must be the same as those used for the radiating surfaces. For example: 25898.μm °R or 14388.μm °K.
3. The first wavelength band extends from 0 to LAMBDA1 and the last band extends from LAMBDA<sub>n</sub> to infinity, where n = NUMBER – 1.
4. Discontinuous segments in the emissivity versus wavelength piecewise linear curve must be treated as a wavelength band of zero width.
5. LAMBDAi must be specified in ascending order, and all LAMBDAi fields where i is greater than or equal to NUMBER must be blank.
6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RADC** Thermal Radiative Coating Properties - SOL400 - RC Network solver

Defines the radiative properties of advanced materials such as coatings and multilayer insulation, commonly used in the aerospace market.

**Format: (COAT)**

1	2	3	4	5	6	7	8	9	10
RADC	RADMID	Emis	Absorptivity	IR Spec	UV Spec				
	"COAT"	IR Traspa	IR Transluc	UV Transpa	UV Transluc	IR Refrac Ind	UV Refrac Ind		

**Example:**

RADC	101	1.	1.	0.					
	COAT	0.	0.	0.	0.	1.	1.		

**Format: (MLI)**

1	2	3	4	5	6	7	8	9	10
RADC	RADMID	Emis	Absorptivity	IR Spec	UV Spec				
	"MLI"	Estar							

**Example:**

RADC	102	1.	1.	0.					
	MLI	0.02							

Describer	Meaning
RADMID	Radiation material identification number (Integer > 0; Required)
COAT	Identification that a coating type material has been started (Character)
Emis	Emissivity (Real $\geq$ 0.0; Default 1.0)
Absorptivity	Absorptivity (Real $\geq$ 0.0; Default 1.0)
IR Spec	IR Specularity (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
UV Spec	UV Specularity (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
IR Transpa	IR Transparency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
IR Transluc	IR Translucency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
UV Transpa	UV Transparency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
UV Transluc	UV Translucency (0.0 $\leq$ Real $\leq$ 1.0; Default 0.0)
IR Refrac Ind	IR Refraction Index (Real $\geq$ 1.0; Default 1.0)
UV Refrac Ind	UV Refraction Index (Real $\geq$ 1.0; Default 1.0)
MLI	Identification that an MLI type material has been started (Character)
Estar	MLI E-star parameter (Real; Default 0.0)



**Remarks:**

1. This entry is for RC Network solver only.
2. Estar is defined as the effective emissivity from the wall to MLI outer surface. The general value is around 0.01 to 0.03, the typical value is 0.02.
3. Emis is usually for the IR waveband, and Absorptivity is for the UV waveband.

## RADCAV

## Radiation Cavity Identification

Identifies the characteristics of each radiant enclosure.

### Format:

1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPCH	NFECI	RMAX	NCOMP	
	SET11	SET12	SET21	SET22	SET31	SET32	etc.		

### Example:

RADCAV	1	1					.99		
	3	5	4	5	7	5			

Describer	Meaning
ICAVITY	Unique cavity identification number associated with enclosure radiation. (Integer > 0)
ELEAMB	CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.)
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character = "YES" or "NO"; Default = "YES")
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. (0.0 ≤ Real ≤ 1.0; Default = 0.0)
PRTPCH	Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer = 0, 1, 2, 3, 4, or 5; Default = blank):

Hemi-Cube & VIEW3D		
Value	Printout in .f06 file	Printout in .pch file
Blank	No	Yes
0	Full Print	Yes
1	No	Yes
2	Full Print	No
3	No	No
4	Summary Print	Yes
5	Summary Print	No

NFECI	Controls whether finite difference or contour integration methods are to be used in the calculation of view factors in the absence of a VIEW3D Bulk Data entry. (Character = "FD" or "CONT"; See Remark 4. for default.)
RMAX	Subelement area factor. See Remark 5. (Real $\geq 0.0$ ; Default = 0.1)
NCOMP	Total number of computational element for one-half ring. See Remark 8. (Default = 32)
SETij	Set identification pairs for the calculation of global view factors. Up to 30 pairs may be specified (i = 1 to 2 and j = 1 to 30). (Integer > 0)

**Remarks:**

1. For the surfaces of an incomplete enclosure (view factors add up to less than 1.0), a complete enclosure may be achieved (SUM = 1.0) by specifying an ambient element, ELEAMB. When multiple cavities are defined, each cavity must have a unique ambient element if ambient elements are desired. No elements can be shared between cavities.
2. Third-body shadowing is ignored in the cavity if SHADOW = "NO". In particular, if it is known a priori that there is no third-body shadowing, SHADOW = NO overrides KSHD and KBSHD fields on the VIEW Bulk Data entry as well as reduces the calculation time immensely.
3. The view factors for a complete enclosure may add up to slightly more than 1.0 due to calculation inaccuracies. SCALE can be used to adjust all the view factors proportionately to acquire a summation equal to the value specified for SCALE. If SCALE is left blank or set to 0.0, no scaling is performed.
4. If the VIEW3D Bulk Data entry is not specified, the view factors are calculated using finite difference and contour integration methods. If NFECI = "FD", then all view factors are calculated using the finite difference technique. NFECI = "CONT" invokes contour integration for all view factor calculations. If NFECI is blank, the program selects a method to use between any two particular elements based on RMAX.
5. The comparison value for RMAX is equal to  $A_s/d_{rs}^2$  where  $A_s$  is the area of a subelement and  $d_{rs}$  is the distance between two subelements r and s for which view factors are being computed. When NFECI is blank, the program selects the contour integral method only if  $A_s/d_{rs}^2 > RMAX$ .
6. When a number of elements are grouped together and considered as a conglomerate surface, view factors can be calculated between these groups. These are referred to as global view factors. The SET1 Bulk Data entry is used to define the conglomerate. When using this feature, negative EIDs are not allowed.
7. If a RADLST and RADMTX entry exists for this cavity ID, new view factors are not computed and the existing RADLST and RADMTX are used in the thermal analysis.
8. The VIEW3D Bulk Data entry must be specified for the calculation of axisymmetric view factors. The process relies on the internal construction of a semi-circle of computational elements. NCOMP specifies the number of such elements desired.
9. For SOL 600, fields 4-8 of the primary entry and all continuation lines are ignored.

10. For SOL 600 field 9 of the primary entry (NCOMP) is used to indicate if the cavity is open or closed according to the following designation:
  - 0 Cavity is closed
  - 1 Cavity is closed an view factors are scaled such that they sum to exactly 1.0
  - 2 Cavity is open
11. For RC network solver in thermal analysis, RADCAV is only used to specify radiation space node, the ELEAMB must be a POINT type CHBDYP element. The SHADOW, SCALE, PRTPCH, NFECI, RMAX, NCOMP, SET11, SET12, SET21, SET22, SET31 and SET32 are ignored.



**RADCOL****Radiation Collective Entity - SOL400 - RC Network solver**

Specifies a collection of boundary elements to be used as a single face in the radiation calculation. This will decrease computation time at the small cost of accuracy. Computational savings and accuracy are dependent on the coarseness of the collection versus the constituents. View factors of the collection are redistributed across the elements for calculation of the radiative energy transfer.

**Format:**

1	2	3	4	5	6	7	8	9	10
RADCOL	RADCOLID	IVIEWF	IVIEWB	RADMIDF	RADMIDB	SET3ID			

**Example:**

RADCOL	101	5	6	2	3	7			
--------	-----	---	---	---	---	---	--	--	--

Describer	Meaning
RADCOLID	Radiation Collection identification number. (Integer > 0; Required)
IVIEWF	A VIEW entry identification number for the front face. (Integer ≥ 0; Default 0)
IVIEWB	A VIEW entry identification number for the back face. (Integer ≥ 0; Default 0)
RADMIDF	RADM identification number for the front face. (Integer ≥ 0; Default 0)
RADMIDB	RADM identification number for the back face. (Integer ≥ 0; Default 0)
SET3 idn	ID of the element collection to be considered a super element. (Integer > 0)

**Remarks:**

1. This entry is for RC Network solver only.
2. IVIEWF/IVIEWB will default to 0 if left blank. This would indicate that the corresponding front and/or back do not participate in the radiation.
3. If an IVIEWF is specified, there must also be a RADMIDF for surface material properties. If a IVIEWB is specified, there must also be a RADMIDB for surface material properties.

**RADCT** Thermal Radiative Functionally-Dependent Properties - SOL400 - RC Network solver

Specifies table references for RADC properties that are functionally dependent.

**Format:**

1	2	3	4	5	6	7	8	9	10
RADCT	RADMID	Emis f(T)	Abs f(T)						

**Example:**

RADCT	11	101	102						
-------	----	-----	-----	--	--	--	--	--	--

Describer	Meaning
RADMID	Radiation material identification number (Integer > 0; Required)
Emis f(T)	Emissivity TABLEMj table ID (Integer ≥ 0; Default.0)
Absorptivity f(T)	Absorptivity TABLEDj table ID (Integer ≥ 0; Default 0)

**Remarks:**

1. This entry is for RC Network solver only.
2. RADCT always comes with the RADC entry. They have the same RADMID so that they can be referenced together.
3. Currently, RC Network solver does not support this entry for enclosure radiation.

**RADLST**

## Listing of Enclosure Radiation Faces

Identifies the individual CHBDYi surface elements that comprise the entire radiation enclosure.

**Format:**

1	2	3	4	5	6	7	8	9	10
RADLST	ICAVITY	MTXTYP	EID1	EID2	EID3	EID4	EID5	EID6	
	EID7	-etc.-							

**Example:**

RADLST	2	1	2	3	4	5	6	7	
--------	---	---	---	---	---	---	---	---	--

Describer	Meaning
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation faces. (Integer > 0)
MTXTYP	Type of radiation exchange matrix used for this cavity. (Integer $\leq 4$ and $\neq 0$ ; Default = 1 for an enclosure without an ambient element. Default = 4 for an enclosure with an ambient element as specified on the RADCAV entry.) <ul style="list-style-type: none"> <li>1 Symmetric view factor matrix [F] and nonconservative radiation matrix [R].</li> <li>2 Symmetric exchange factor matrix [<math>\mathfrak{F}</math>] and conservative radiation matrix [R].</li> <li>3 Unsymmetric exchange factor matrix [<math>\mathfrak{F}</math>] and conservative radiation matrix [R].</li> <li>4 Symmetric view factor matrix [F] and conservative radiation matrix [R].</li> <li>-n The first n CHBDYi elements may lose energy to space but the remainder may not. Symmetric exchange factor matrix [F] and nonconservative radiation matrix [R].</li> </ul>
EIDi	Identification numbers for the CHBDYi elements in this radiation cavity. (Integer $\neq 0$ or "THRU")

**Remarks:**

1. A radiation EIDi list isolates those CHBDYi surface element faces that are to communicate in a radiation enclosure. View-factor calculation and RADMTX formation for an enclosure is performed only for (or among) those faces identified within the same RADCAV.
2. A radiation exchange matrix (RADMTX) can only reference one radiative face list (RADLST). The companion RADCAV, RADLST, and RADMTX must share a unique ICAVITY.
3. For each EIDi, the appropriate CHBDYi element is located, and the proper RADM entry ID field found.
4. If the radiation exchange matrix or any radiation boundary conditions are available from an external source, the RADMTX must be user generated.

5. Multiple RADLST entries may be specified.
6. If any RADLST entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADLST entry changes or additions are not recommended on restart.
7. RC network solver does not support RADLST for thermal analysis.

## RADM

## Radiation Boundary Material Property

Defines the radiation properties of a boundary element for heat transfer analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

### Example:

RADM	11		.45	.33	.29	.20	.17	.13	
------	----	--	-----	-----	-----	-----	-----	-----	--

Describer	Meaning
RADMID	Material identification number. (Integer > 0)
ABSORP	Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2. ( $0.0 \leq \text{Real} \leq 1.0$ )
EMIS <sub>i</sub>	Surface emissivity at wavelength LAMBDA <sub>i</sub> or the temperature function curve multiplier if EMIS <sub>i</sub> is variable (See the RADBND entry.) ( $0.0 \leq \text{Real} \leq 1.0$ )

### Remarks:

1. The RADM entry is directly referenced only by one of the CHBDYE, CHBDYG, or CHBDYP type surface element entries.
2. For radiation enclosure problems, ABSORP is set equal to emissivity. For QVECT loads, absorptivity is specified by ABSORP.
3. If there is more than one EMIS<sub>i</sub>, then:
  - There must be a RADBND entry.
  - The number of EMIS<sub>i</sub> may not exceed the number of LAMBDA<sub>i</sub> on the RADBND entry.
  - The emissivity values are given for a wavelength specified by the corresponding LAMBDA<sub>i</sub> on the RADBND entry. Within each discrete wavelength band, the emissivity is assumed to be constant.
  - At any specific wavelength and surface temperature, the absorptivity is exactly equal to the emissivity.
4. To perform any radiation heat transfer exchange, the user must furnish PARAM entries for:
  - TABS to define the absolute temperature scale.
  - SIGMA ( $\sigma$ ) to define the Stefan-Boltzmann constant in appropriate units.
5. RC network solver only supports RADMID, ABSORP and EMIS1 for thermal analysis, other fields are ignored.

## RADMT

## Radiation Boundary Material Property Temperature Dependence

Specifies table references for temperature dependent RADM entry radiation boundary properties.

### Format:

1	2	3	4	5	6	7	8	9	10
RADMT	RADMID	T(A)	$T(\epsilon_1)$	$T(\epsilon_2)$	$T(\epsilon_3)$	$T(\epsilon_4)$	$T(\epsilon_5)$	$T(\epsilon_6)$	
	$T(\epsilon_7)$	-etc.-							

### Example:

RADMT	11		1	2	3	4	5	6	
-------	----	--	---	---	---	---	---	---	--

Describer	Meaning
RADMID	Material identification number. (Integer > 0)
T(A)	TABLEMj identifier for surface absorptivity. (Integer $\geq 0$ or blank)
$T(\epsilon_j)$	TABLEMj identifiers for surface emissivity. (Integer $\geq 0$ or blank)

### Remarks:

1. The basic quantities on the RADM entry of the same RADMID are always multiplied by the corresponding tabular function.
2. Tables T(A) and  $T(\epsilon_j)$  have an upper bound that is less than or equal to one and a lower bound that is greater than or equal to zero.
3. The TABLEMj enforces the element temperature as the independent variable. Blank or zero fields means there is no temperature dependence of the referenced property on the RADM entry.
4. RC network solver only supports RADMID, T(A) and  $T(\epsilon_1)$  for thermal analysis, other fields are ignored.

## RADMTX Radiation Exchange Matrix

Provides the  $F_{ji} = A_j f_{ji}$  exchange factors for all the faces of a radiation enclosure specified in the corresponding RADLST entry.

### Format:

1	2	3	4	5	6	7	8	9	10
RADMTX	ICAVITY	INDEX	$F_{i,j}$	$F_{i+1,j}$	$F_{i+2,j}$	$F_{i+3,j}$	$F_{i+4,j}$	$F_{i+5,j}$	
	$F_{i+6,j}$	-etc.-							

### Example:

RADMTX	2	1	0.0	0.1	0.2	0.2	0.3	0.2	
--------	---	---	-----	-----	-----	-----	-----	-----	--

Descriptor	Meaning
ICAVITY	Unique cavity identification number that links a radiation exchange matrix with its listing of enclosure radiation surface elements. (Integer > 0)
INDEX	Column number in the matrix. (Integer > 0)
$F_{k,j}$	If symmetric, the matrix values start on the diagonal ( $i = j$ ) and continue down the column ( $k = i + 1, i + 2, \text{etc.}$ ). If unsymmetric, the values start in row ( $i = 1$ ). $i$ refers to EID <sub>i</sub> on the RADLST entry. (Real $\geq 0$ )

### Remarks:

1. If the matrix is symmetric, only the lower triangle is input, and  $i = j = \text{INDEX}$ . If the matrix is unsymmetric,  $i = 1$ , and  $j = \text{INDEX}$ .
2. Only one ICAVITY may be referenced for those faces that are to be included in a unique radiation matrix.
3. Coefficients are listed by column with the number of columns equal to the number of entries in the RADLST.
4. All faces involved in any radiation enclosure must be defined with an CHBDY<sub>i</sub> element.
5. If any RADMTX entry is changed or added on restart then a complete re-analysis may be performed. Therefore, RADMTX entry changes or additions are not recommended on restart.
6. Set NASTRAN SYSTEM (87) = 3 is a new option in MSC Nastran 2005 that prevents radiation energy from being lost to space.
7. RC network solver does not support RADMTX for thermal analysis.

**RADSET** Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
RADSET	ICAVITY1	ICAVITY2	ICAVITY3	ICAVITY4	ICAVITY5	ICAVITY6	ICAVITY7	ICAVITY8	
	ICAVITY9	-etc.-							

**Example:**

RADSET	1	2	3	4					
--------	---	---	---	---	--	--	--	--	--

Describer	Meaning
ICAVITYi	Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0)

**Remark:**

1. For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.



## RANDPS Power Spectral Density Specification

Defines load set power spectral density factors for use in random analysis having the frequency dependent form

$$S_{jk}(F) = (X + iY)G(F)$$

### Format:

1	2	3	4	5	6	7	8	9	10
RANDPS	SID	J	K	X	Y	TID			

### Example:

RANDPS	5	3	7	2.0	2.5	4			
--------	---	---	---	-----	-----	---	--	--	--

Descriptor	Meaning
SID	Random analysis set identification number. See Remarks 1. and 9. (Integer > 0)
J	Subcase identification number of the excited load set. See Remarks 6. through 9. (Integer > 0)
K	Subcase identification number of the applied load set. See Remarks 6. through 9. (Integer ≥ 0; K ≥ J)
X, Y	Components of the complex number. See Remarks 2. and 7. (Real)
TID	Identification number of a TABRNDi entry that defines G(F). See Remarks 3. and 7. (Integer > 0 or Blank)

### Remarks:

1. Set identification numbers must be selected with the Case Control command (RANDOM = SID) or through use of FTGLOAD when doing random vibration fatigue analysis.
2. For auto spectral density, J = K, X must be greater than zero and Y must be equal to zero.
3. For TID = Blank, G(F) = 1.0.
4. RANDPS may only reference subcases included within a single loop (a change in direct matrix input is not allowed).
5. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
6. In the case of cyclic symmetry Solution Sequence 118, J and K must refer to the coded subcase IDs.
7. In superelement analysis, J and K must refer to the superelement subcases. For example, if superelement 10 has SUBCASEs 1 and 2 and superelement 20 has SUBCASEs 3 and 4, then a separate RANDPS entry is required for each superelement, even though X, Y, and TID may be identical.
8. For uncoupled PSDF (no J < K entries) only one J = K entry is allowed for unique value of J. For coupled PSDF (some J < K entries) any number of entries are allowed.

9. J and K must reference valid subcase IDs. In superelement analysis, J and K must reference valid subcase IDs that pertain to the applicable superelement. If this requirement is not met, the program issues an appropriate user warning message and ignores the associated RANDOM=SID command.
10. If Modules are present then this entry may only be specified in the main Bulk Data section.

## RANDT1 Autocorrelation Function Time Lag

Defines time lag constants for use in random analysis autocorrelation function calculation.

### Format:

1	2	3	4	5	6	7	8	9	10
RANDT1	SID	N	T0	TMAX					

### Example:

RANDT1	5	10	3.2	9.6					
--------	---	----	-----	-----	--	--	--	--	--

Describer	Meaning
SID	Random analysis set identification number. (Integer > 0)
N	Number of time lag intervals. (Integer > 0)
T0	Starting time lag. (Real ≥ 0.0)
TMAX	Maximum time lag. (Real > T0)

### Remarks:

1. Time lag sets must be selected with the Case Control command RANDOM = SID.
2. At least one RANDPS entry must be present with the same set identification number.
3. The time lags defined on this entry are given by

$$T_i = T0 + \frac{TMAX - T0}{N}(i - 1), i = 1, N + 2$$

4. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of random analysis.
5. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RBAR**

## Rigid Bar

Defines a rigid bar with six degrees-of-freedom at each end.

**Format:**

1	2	3	4	5	6	7	8	9	10
RBAR	EID	GA	GB	CNA	CNB	CMA	CMB	ALPHA	
	TREF								

**Example:**

RBAR	5	1	2	123456				6.5-6	
------	---	---	---	--------	--	--	--	-------	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification number of connection points. (Integer > 0)
CNA, CNB	Component numbers of independent degrees-of-freedom in the global coordinate system for the element at grid points GA and GB. See Remark 3. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
CMA, CMB	Component numbers of dependent degrees-of-freedom in the global coordinate system assigned by the element at grid points GA and GB. See Remarks 4. and 5. (Integers 1 through 6 with no embedded blanks, or zero or blank.)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create the Lagrange multiplier degrees-of-freedom internally in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- For the linear method, the total number of components in CNA and CNB must equal six; for example, CNA = 1236, CNB = 34. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA = 123456 or CNB = 123456 is allowed. If both CNA and CNB are blank, then CNA = 123456. For this method, RBAR1 gives the simpler input format.

4. If both CMA and CMB are zero or blank, all of the degrees-of-freedom not in CNA and CNB will be made dependent. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected in the RIGID Case Control command. However, the rules regarding the m-set described below apply to both methods.
5. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RBAR, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.
12. When there is large rotation, CMA and CMB must have all “456” if any rotational degrees of freedom is used.

**RBAR1**

## Rigid Bar

Alternative format for RBAR.

**Format:**

1	2	3	4	5	6	7	8	9	10
RBAR1	EID	GA	GB	CB	ALPHA	TREF			

**Example:**

RBAR1	5	1	2	123	6.5-6				
-------	---	---	---	-----	-------	--	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB, which are constrained to move as the rigid bar. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.)
ALPHA	Thermal expansion coefficient. See Remark 8. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
- RBAR1 is a preferred input format to define the Lagrange method for a rigid bar.
- When CB = "123456" or blank, the grid point GB is constrained to move with GA as a rigid bar. For default CB = "123456". Any number of degrees-of-freedom at grid point GB can be released not to move with the rigid body.
- For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA and GB will be computed even if these two points have different global coordinate systems.
- For the Lagrange method, the thermal expansion effect will be computed for the rigid bar element if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

7. Element identification numbers should be unique with respect to all other element identification numbers.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

## RBAX3D

## Axisymmetric Harmonic Element Rigid Connection

Defines a rigid connection between a point on an axisymmetric harmonic element (CQUADX or CTRIAX) and a point on the axis of symmetry of that element.

### Format:

1	2	3	4	5	6	7	8	9	10
RBAX3D	EID	G3D	GAX						

### Example:

RBAX3D	100	10	20						
--------	-----	----	----	--	--	--	--	--	--

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000). See Remark 1..
G3D	Identification number of a grid point on the axis of symmetry (Integer > 100,000,000). See Remarks 2. and 4. through 7.
GAX	Identification number of a grid point referenced on an axisymmetric harmonic element (CQUADX or CTRIAX). (0 < Integer < 100,000,000). See Remarks 3. through 7.

### Remarks:

1. The EID should be unique with respect to all other elements.
2. The G3D point may not be referenced on an axisymmetric harmonic element. If it is, the program terminates the execution with an appropriate user fatal message.
3. The GAX point must be referenced on an axisymmetric harmonic element. If it is not, the program terminates the execution with an appropriate user fatal message. Further, the axisymmetric harmonic element (or elements) that reference GAX must have harmonic index 1 specified on the associated PAXSYMH entry. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. The G3D point must be on the axis of symmetry. The GAX point need not be on the axis of symmetry.
5. The G3D and GAX points must both have the same axial coordinate along the axis of symmetry.
6. The G3D point is regarded as the independent point and the GAX point is regarded as the dependent point. The dependent degrees of freedom of GAX will be selected from the axisymmetric harmonic degrees of freedom.



7. If the GAX point is on the axis of symmetry, the connection is made only to the G3D displacement components perpendicular to the axis. However, if the GAX point is not on the axis of symmetry, then connection is made not only to the G3D displacement components perpendicular to the axis, but also to the slopes. The tolerance to determine if the GAX point is on the axis of symmetry is  $1.0E-6$ .

## RBE1 Rigid Body Element, Form 1

Defines a rigid body connected to an arbitrary number of grid points.

### Format:

1	2	3	4	5	6	7	8	9	10
RBE1	EID	GN1	CN1	GN2	CN2	GN3	CN3		
		GN4	CN4	GN5	CN5	GN6	CN6		
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	-etc.-	ALPHA	TREF			

### Example:

RBE1	59	59	123456						
	UM	61	246	6.5-6					

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GNi	Grid points at which independent degrees-of-freedom for the element are assigned. (Integer > 0)
CNi	Independent degrees-of-freedom in the global coordinate system for the rigid element at grid point(s) GNi. See Remark 1. (Integers 1 through 6 with no embedded blanks.)
“UM”	Indicates the start of the dependent degrees-of-freedom. (Character)
GMj	Grid points at which dependent degrees-of-freedom are assigned. (Integer > 0)
CMj	Dependent degrees-of-freedom in the global coordinate system at grid point(s) GMj. (Integers 1 through 6 with no embedded blanks.)
ALPHA	Thermal expansion coefficient. See Remark 13. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CMj.

3. For the linear method, the total number of components in CN1 to CN6 must equal six; for example, CN1 = 123, CN2 = 3, CN3 = 2, CN4 = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. The first continuation entry is not required if there are fewer than four GN points. For the Lagrange method, the total number of components must also be six. In addition, CN1 must be 123456, and CN2 through CN6 must be blank.
4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be member of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding to m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. A degree-of-freedom cannot be both independent and dependent for the same element. However, both independent and dependent components can exist at the same grid point.
7. Element identification numbers should be unique with respect to all other element identification numbers.
8. RBE1, among other eligible rigid element types, can be selected via MPC and SET3.
9. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
10. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
11. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
13. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The average temperature of the connected grid points is used as the temperature of the rigid body element. In this case, for the Lagrange method, PARAM, BAILOUT, -1 is activated within the solver.

## RBE2

## Rigid Body Element, Form 2

Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points.

### Format:

1	2	3	4	5	6	7	8	9	10
RBE2	EID	GN	CM	GM1	GM2	GM3	GM4	GM5	
	GM6	GM7	GM8	-etc.-	ALPHA	TREF			

### Example:

RBE2	9	8	12	10	12	14	15	16	
	20	6.5-6							

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GN	Identification number of grid point to which all six independent degrees-of-freedom for the element are assigned. (Integer > 0)
CM	Component numbers of the dependent degrees-of-freedom in the global coordinate system at grid points GMi. (Integers 1 through 6 with no embedded blanks.) See Remark 12.
GMi	Grid point identification numbers at which dependent degrees-of-freedom are assigned. (Integer > 0)
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom which is obtained by CM multiplied with the number of dependent grid points.
- For the linear method, the dependent degrees-of-freedom indicated by CM will be made members of the m-set at all grid points. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
- Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

5. Element identification numbers should be unique with respect to all other element identification numbers.
6. RBE2, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The average temperature of the connected grid points is used as the temperature of the rigid body element. In this case, for the Lagrange method, PARAM, BAILOUT, -1 is activated within the solver.
12. It can be any combination in 1, 2, 3, but must have all "456" if any rotation DOF is used when there is large rotation.

## RBE2GS

## Internally Generate an RBE2 Element

Defines an RBE2 connecting the two closest grids to GS.

### Format:

1	2	3	4	5	6	7	8	9	10
RBE2GS	EID	GS	TYPE		TREF	R	CM	ALPHA	
	XS	YS	ZS	GNi	GMj				

### Example:

RBE2GS	3	17				1.3		6.5-6	
				end/	56	99			

### Alternate Formats and Examples:

RBE2GS	EID	GS	TYPE			R	CM	ALPHA	
	XS	YS	ZS	GNi	THRU	GNj	end/		
	GMk	THRU	GM/						

RBE2GS	15	35				-.66			
				88	THRU	107	end/	15	
	76	88							

RBE2GS	25					.66			
	5.173	0.0	19.3185	88	99	108	end/		
	15	THRU	88						

RBE2GS	35	28				-.66			
				56	THRU	102	end/	19	
	21	THRU	200						

RBE2GS	45	16				-.66			
				56	THRU	102	1129		
	1146	THRU	1200	end/					

Descriptor	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GS	Search POINT or GRID point. (Integer ≥ 0 or blank)
TYPE	Connectivity: (Character)
	blank                      Search the complete model. (Default)

Describer	Meaning
	NMFLIP The independent and dependent DOF's are interchanged.
	IIRBE2 The grids chosen will be the independent GN's of the two closest existing RBE2 elements.
	NMIIRBE2 Same as 'IIRBE2' except the independent and dependent grids are interchanged.
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).
R	Radius. (Real > 0, or < 0)
CM	Component number of dependent degrees-of-freedom for grid GM. (Integers 1 through 6 with no embedded blanks. Blank defaults to 123456)
ALPHA	Thermal expansion coefficient. (Real or blank)
XS, YS, ZS	Location of search point if GS is blank. Used only if GS=0 or blank. (Real or blank)
GNi	List of grids to be excluded from candidate grids for GN. If a GNi list is given it must end with an <code>end/</code> . (Integer > 0 or blank or "THRU")
GMj	List of grids to be excluded from candidate grids for GM. If no GNi list is given and a GMj list is present, then GNi must have an <code>end/</code> entry. (Integer > 0 or blank or "THRU")

1. Element ID numbers must be unique with respect to all other element ID numbers.
2. This entry will internally define a RBE2 element with the same ID of the RBE2GS entry. The grid assigned to GN will always be the independent grid. GM will be the dependent grid. If GS is a 'POINT' entry (or GS is blank and XYZ is specified), the first two grids that fall within the search radius  $|R|$  about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid GN the next closest will be the dependent grid GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj (and if TYPE = 'NMFLIP', then GN and GM will be reversed. If GS is 'GRID' entry, and is part of the physical model, i.e., it has physical structural elements attached to it, it will become GN the independent grid for the RBE2 to be generated. If it is in the GNi exclusion list, the next closest grid will be chosen. the closest grid within the search radius about  $|R|$  GN will be chosen as GM, the dependent grid. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with our without use of the exclusion lists for GNi and GMj) and if TYPE = 'NMFLIP', then GN and GM will be reversed. If GS is a 'GRID' entry, and is determined not to be part of the physical model, the first two grids that fall within the search radius  $|R|$  about GS (or XYZ) will be chosen as GN and GM. The closest to the search location will be the independent grid, GN, the next closest will be the dependent grid, GM. Any grids contained in a GNi list will be excluded from the GN search and any grids contained in a GMj list will be excluded from the GM search. After GN and GM have been determined (with or without use of the exclusion lists for GNi and GMj) and if TYPE='NMFLIP', then GN and GM

will be reversed. The GS grid will remain on the GEOM1 table for post-processing viewing purposes, but will not be part of the Nastran solution g-set. If TYPE='IIRBE2', the two grids chosen for the RBE2 will be the independent GN's of existing RBE2 elements whose independent grids lie within the search radius. If TYPE='NMIIRB2', then GN and GM will be reversed. If no existing RBE2 elements have independent grids within the search radius or if only one existing RBE2 has an independent grid within the search radius, then a fatal message will be issued. POINT and GRID entries must be unique with respect to all other POINT and GRID entries. If TYPE='IIRB2', it is recommended that PARAM,AUTOMSET,YES be used in the analysis run.

3. If  $R < 0$ , the two located grids GN and GM will be made coincident to the GS (or XYZ) location.
4. If CM is 123456 and GM is touching only solid elements, CM will be internally changed to 123. For solid elements, the grids GN and GM should always be coincident.
5. When Module GP4 is run, checks are made to insure that the selected grids, GN and GM, do not violate existing constraint sets. If a violation occurs a fatal message will be issued for the offending grids. These grids can be excluded from further search inclusion by specifying them on the GNi or GMj list. PARAM,AUTOMSET,YES will often prevent any violation.
6. If GS=0 or blank and XS, YX, ZS is not specified or if both GS and XS, YS, ZS are specified a fatal error will occur.
7. The end of a grid exclusion lists is indicated by the existence of "end/" in the field following the last entry in the list. In the "thru" option, not all grids in the range need exist.
8. For superelement or part superelement connection the independent grid assigned to GN will be exterior to the superelement. The dependent grid GM will be an interior grid to the superelement. If the resulting RBE2 element, connects two different superelements, the element will be assigned to the superelement with the lower ID and the grid attached to the superelement with the higher ID will be moved to the superelement with the lower ID.
9. If the RBE2GS is listed on a SEELT entry, it will be placed as the SEELT directs. If say grid G1 lies in another superelement and G2 lies in the SEELT superelement, G1 will be moved to the SEELT defined superelement.
10. "THRU" should not be the first nonblank field of a continuation line. Blank fields are allowed in the exclusion lists for readability.
11. Use PARAM,SEP1XOVR,16 to print the grids found by each RBE2GS entry.
12. RBE2GS is not supported in the presence of part (BEGIN SUPER) superelements.
13. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The average temperature of the connected grid points is used as the temperature of the rigid body element. In this case, for the Lagrange method, PARAM, BAILOUT, -1 is activated within the solver.



## RBE3 Interpolation Constraint Element

Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points.

### Format:

1	2	3	4	5	6	7	8	9	10
RBE3	EID		REFGRID	REFC	WT1	C1	G1,1	G1,2	
	G1,3	WT2	C2	G2,1	G2,2	-etc.-	WT3	C3	
	G3,1	G3,2	-etc.-	WT4	C4	G4,1	G4,2	-etc.-	
	“UM”	GM1	CM1	GM2	CM2	GM3	CM3		
		GM4	CM4	GM5	CM5	-etc.-			
	“ALPHA”	ALPHA	TREF						

### Example:

RBE3	14		100	1234	1.0	123	1	3	
	5	4.7	1	2	4	6	5.2	2	
	7	8	9	5.1	1	15	16		
	UM	100	14	5	3	7	2		
	ALPHA	6.5-6							

Descriptor	Meaning
EID	Element identification number. Unique with respect to all elements. (0 < Integer < 100,000,000)
REFGRID	Reference grid point identification number. (Integer > 0)
REFC	Component numbers at the reference grid point. (Any of the integers 1 through 6 with no embedded blanks.)
WTi	Weighting factor for components of motion on the following entry at grid points Gi,j. (Real)
Ci	Component numbers with weighting factor WTi at grid points Gi,j. (Any of the integers 1 through 6 with no embedded blanks.)
Gi,j	Grid points with components Ci that have weighting factor WTi in the averaging equations. (Integer > 0)
“UM”	Indicates the start of the degrees-of-freedom belonging to the dependent degrees-of-freedom. The default action is to assign only the components in REFC to the dependent degrees-of-freedom. (Character)
GMi	Identification numbers of grid points with degrees-of-freedom in the m-set. (Integer > 0)

Describer	Meaning
CMi	Component numbers of GMi to be assigned to the m-set. (Any of the Integers 1 through 6 with no embedded blanks.)
“ALPHA”	Indicates that the next number is the coefficient of thermal expansion. (Character)
ALPHA	Thermal expansion coefficient. See Remark 14. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, the REFC must be “123”, “456”, or “123456”. No other combination is allowed.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points. The number of Lagrange multiplier degrees-of-freedom is equal to the number of degrees-of-freedom given by REFC.
- For the linear method, the dependent degrees-of-freedom indicated by REFC will be made members of the m-set. For Lagrange rigid element, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
- We recommend that for most applications only the translation components 123 be used for Ci. An exception is the case where the Gi,j are colinear. A rotation component may then be added to one grid point to stabilize its associated rigid body mode for the element.
- Blank spaces may be left at the end of a Gi,j sequence.
- For the Lagrange method, the default for “UM” must be used. For the linear method, the default for “UM” should be used except in cases where the user wishes to include some or all REFC components in displacement sets exclusive from the m-set. If the default is not used for “UM”:
  - The total number of components in the m-set (i.e., the total number of dependent degrees-of-freedom defined by the element) must be equal to the number of components in REFC (four components in the example).
  - The components specified after “UM” must be a subset of the components specified under REFC and (Gi,j, Ci).
  - The m-rows by m-columns partition [Rm,m] of the global stiffness containing the dependent degrees-of-freedom of multipoint constraints and rigid elements must be nonsingular. PARAM,CHECKOUT in SOLs 101 through 200 may be used to check for this condition. For some complex constraints, forming a well-conditioned [Rm,m] is difficult for a user to perform manually and PARAM,AUTOMSET,YES will automatically determine a well-conditioned partition by employing a rectangular decomposition of the [Rm,g] matrix.
- Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.

9. RBE3, among other eligible rigid element types, can be selected via MPC and SET3.
10. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
11. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
12. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets, 1103](#) for a list of these entries.
13. The formulation for the RBE3 element was changed in Version 70.7. This change allowed the element to give consistent answers that are not dependent upon the units of the model. Only models that connected rotation degrees-of-freedom for Ci were affected. Note that these models are ignoring the recommendation in Remark 5. The formulation prior to Version 70.7 may be obtained by setting SYSTEM(310)=1.
14. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The average temperature of the connected grid points is used as the temperature of the rigid body element. In this case, for the Lagrange method, PARAM, BAILOUT, -1 is activated within the solver.

**RBE3U** Defines Method to Distribute Applied Loads to a Surface in SOL 600

**Format:**

1	2	3	4	5	6	7	8	9	10
RBE3U	EID	IOPT	GREF	G1	G2	G3	G4	G5	
	G6	G7	G8	G9	etc.				

**Alternate Format:**

1	2	3	4	5	6	7	8	9	10
RBE3U	EID	IOPT	GREF	G1	THRU	G3	BY	G4	
	G5	THRU	G6	BY	G7				
	G8	THRU	G9	BY	G10	etc.			

**Examples:**

RBE3U	250	1	1000	100	THRU	120			
-------	-----	---	------	-----	------	-----	--	--	--

Describer	Meaning
EID	Identification number, unique among all elements. (Integer; no Default; Required)
IOPT	Option of how to distribute the load. (Integer; Default = 1) 1 Equal loads values will be applied to all grid points on the surface. 2 Loads will be distributed according to how close each grid point on the surface is to GREF.
Gi	List of grids to which forces at GREF will be distributed. (Integer; Default = 123)

**Remarks:**

1. The sum of the loads on all grid points is the same as the applied to GREF in each of the CREF directions. If IOPT=1 the force on each grid point will be the total loads divided by the number of applicable grid points. If IOPT=2 the load of each grid point will be weighted according to its distance from GREF, however the sum of all loads will be the same as that applied to GREF. The load is removed from GREF after distribution.
2. RBE3U is only used to distributed applied loads. No actual rigid elements are included in the analysis. If rigid elements are desired, use RBE2 or RBE3.
3. Only FORCE and MOMENT will be distributed using this entry. All other types of forces such as FORCE1, MOMENT1, TEMP, etc. applied to GREF will not be distributed.

## RBJOINT

### Defines a Joint Between Two Rigid Bodies

Defines a joint between two rigid bodies. Used in SOL 700 only.

#### Format:

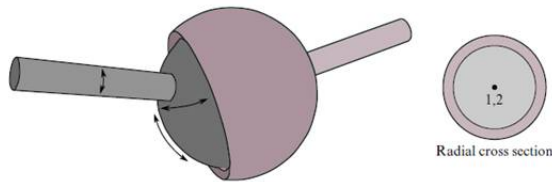
1	2	3	4	5	6	7	8	9	10
RBJOINT	ID	TYPE	N1	N2	N3	N4	N5	N6	
		RPS							

#### Examples:

##### Example 1 - Cylindrical Joint

RBJOINT	1	CYLIND	101	201	103	204			
---------	---	--------	-----	-----	-----	-----	--	--	--

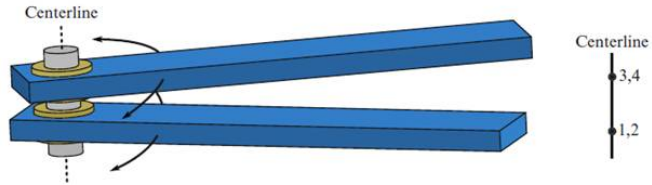
Describer	Meaning
ID	RBJOIN identification number. (Integer > 0; Required)
TYPE	Type of RBJOIN. (Character; Required) Types available are (node numbers in the images correspond to N1, N2, etc.): TYPE=SPHER(spherical), The relative motion of the rigid bodies are constrained so that nodes which are initially coincident remain coincident. In the below figure the socket's node is not interior to the socket-SOL700 does not require that a rigid body's nodes be interior to the body.



Spherical joint

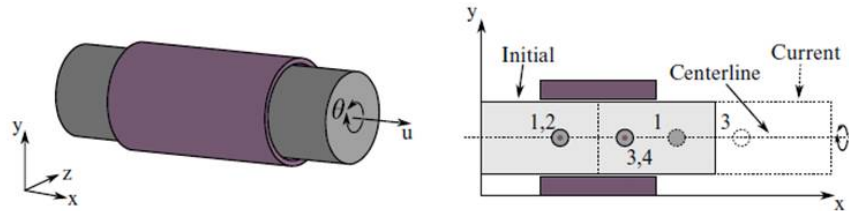
Describer	Meaning
-----------	---------

TYPE=REVOLUTE, Both nodal pairs (1,2) and (3,4) are constrained to remain coincident. Consequently, the relative motion of these rigid bodies is restricted to rotations about the line segment formed by the two pairs of coincident nodes. This segment is labeled the "centerline".



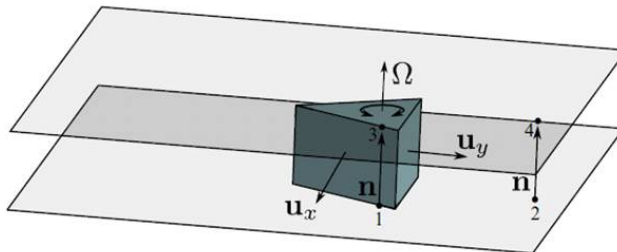
Revolute joint

TYPE=CYLIND (cylindrical), This joint is derived from the rotational joint by relaxing the constraints along the centerline. This joint admits relative rotation and translation along the centerline.



Cylindrical joint

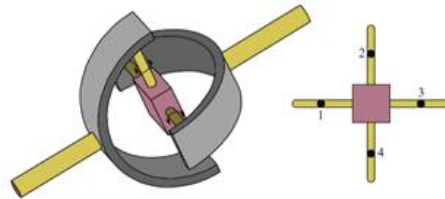
TYPE=PLANAR, This joint is derived from the rotational joint by relaxing the constraints normal to the centerline. Relatively displacements along the direction of the centerline are excluded.



Planar joint

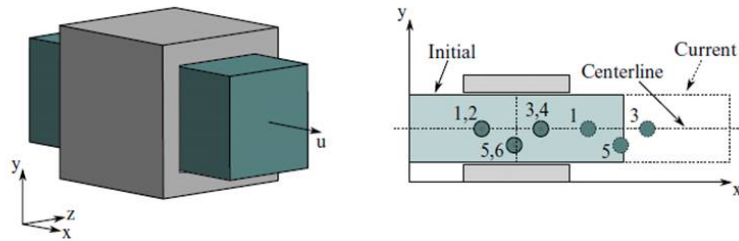
Describer	Meaning
-----------	---------

TYPE=UNIVERS (universal joint), In contrast with the preceding joints, nodal pairs (1,2) and (3,4) are not initially coincident. Rather, the segments formed by (1,3) and (2,4) must be orthogonal; and they serve as axes about which the two bodies may undergo relative rotation. The universal joint excludes all other relative motion and the axes remain orthogonal at all time.



Universal joint

TYPE=TRANSL (translational), This is a cylindrical joint with a third pair of off-centerline nodes which restrict rotation. Aside from translation along the centerline the two rigid bodies are stuck together.



Translational joint

- N1 GRID id of Node 1, in rigid body A. Define for all joint types
- N2 GRID id of Node 2, in rigid body B. Define for all joint types
- N3 GRID id of Node 3, in rigid body A. Define for all joint types except SPHER.
- N4 GRID id of Node 4, in rigid body B. Define for all joint types except SPHER.
- N5 GRID id of Node 5, in rigid body A. Define only for joint types TRANSL.
- N6 GRID id of Node 6, in rigid body B. Define only for joint types TRANSL.
- RPS Relative penalty stiffness (Default = 1.0)

**RCONN**

## Rigid Connection

Defines a rigid connection between the different parts of Lagrangian meshes (tied surfaces). Used in SOL 700 only.

**Format:**

1	2	3	4	5	6	7	8	9	10
RCONN	CID	STYPE	MTYPE	SID	MID	OPTION			
	CLSGAP	GAPDIS	GAPDISV						

**Example:**

RCONN	7	GRID	SURF	3	7	NORMAL			
-------	---	------	------	---	---	--------	--	--	--

Field	Content
CID	Unique rigid-connection number (Integer; Required)
STYPE	Type of entity used to define the secondary surface (Character; Default = SURF)
	<p>SURF                    The faces of the elements are used for the secondary surface. SID refers to BSURF ID. See Remark 1.</p> <p>GRID                    Grid points will be tied to the primary surface. SID then refers to a SET or BCGRID ID containing the list of grid points to be used. See Remarks 2. and 3.</p>
MTYPE	Type of entity used to define the primary surface (Character; Default = SURF)
	<p>SURF                    The faces of the elements are used for the primary surface. MID refers to BSURF ID.</p>
SID	Secondary BSURF ID or SET1 ID containing the list of grid points (Integer; Required)
MID	Primary BSURF ID (Integer; Required)
OPTION	Only used if discrete grid points are tied to a surface (STYPE is equal to GRID). (Character; Default = NORMAL)
	<p>NORMAL                The grid points are tied to the primary surface. See Remark 2.</p> <p>SHELL                  The grid points are attached to the edge of shell or beam elements, which are tied to the shell surface. See Remark 3.</p>
CLSGAP	Switch to automatically close any gaps that are present between the primary-secondary surface (Character; Default = NO)
	<p>YES                      Gaps are automatically closed</p> <p>NO                        Gaps are not closed. See Remark 2.</p>



Field	Content
GAPDIS	Defines the tolerance used in the search for a primary face. If the distance between a secondary point and a primary face falls within this tolerance, the primary face is accepted. If not, the search for a correct primary face continues (Character; Default = DISTANCE)  DISTANCE      The tolerance has the length as specified at GAPDISV
GAPDISV	The value of the gap tolerance or a factor to calculate this tolerance depending on the value of GAPDIS (Real; Default = 1.0E20)

### Remarks

1. The RCONN entry can be used to define three types of connection:
  - a. Two Surfaces Tied Together.
  - b. Define secondary and primary segments representing the two surfaces to be tied together. There should not be a gap between the two sets of segments. The two surfaces are tied together during the analysis.
  - c. Grid Points Tied to a Surface.
  - d. If STYPE is set to GRID and OPTION is set to NORMAL, the secondary entities comprise discrete grid points that are tied to the primary surface during the analysis. The grid points must lie on the surface.
  - e. Shell Edge Tied to a Shell Surface.
  - f. If STYPE is set to GRID and OPTION is set to SHELL, the edges of shell or beams elements can be tied to the faces of other shells. The grid points attached to the edge of the shells/beams must be selected as the secondary grid points. The shell surface to which they are tied must be selected as the primary surface. The two sets will then be tied together throughout the analysis. All degrees of freedom will be coupled.
2. The CLSGAP entry enables you to define two different meshes that are not coincident over the primary/secondary interface. If the option is set to YES, the secondary surface becomes coincident (according to projections) with the primary surface.
3. The search method of the contact algorithm is used to find the closest primary face. The tolerance defined with the GAPDIS/GAPDISV fields is similar to the monitoring distance defined on the CONTACT entry with the MONDIS/ MONDISV fields.
4. The use of the gap closing CLSGAP can cause an element to collapse. This may happen if the GAPDISV tolerance is set to a value greater than the length of the side of an element.
5. When a solid and a shell mesh are tied together, the rotational degrees of freedom of the shell grid points are not coupled.
6. When OPTION=SHELL and CLSGAP=NO, the time step scale factor will be set to 0.4. This can be overwritten by:  
PARAM\*, STEPFCRCONN, xxx

## RCPARM

## Parameters for RC Heat Transfer Analysis Control

Defines a set of parameters for nonlinear resistance-capacitance based analysis.

### Format:

1	2	3	4	5	6	7	8	9	10
RCPARM	ID	SOLVER	DRLXCA	ARLXCA	BALENG	NLOOP	DAMPD	GRVCON	
	TIMEND	OUTPUT	CSGFAC	DTIMEI					

### Example:

RCPARM	1	SNDUFR	1.0-2	1.0-2	1.0-4	2000	0.7	9.81	
	1800.	60.	1.2	0.0					

Describer	Meaning
ID	Identification number. See Remark 2. (Integer > 0; Required)
Solver	Equivalent Sinda Solution name. See Remarks 3. and 7. (String; Required) Possible values: RCNS "SNSOR", "STDSTL" RCNT "SNFRDL", "FWDBKL", "SNADE", "SNDUFR", "ATSDUF", "ATSF BK", "SNTSM", "SNTSM3", "SNTSM1", "SNTSM4", Maintained for legacy, but not shown in GUI "SNSOR1", "SNSORA", "SNSOR1A", "SNHOSD", "SNSOSS", "SNHOSS", "SCROUT", "SNDSDNR", "TRSPMA"
DRLXCA	Diffusion node convergence criterion (Real $\geq$ 0.0; Default 1.0e-3 degrees) (SimX: Default 1.0e-4 for transient)
ARLXCA	Arithmetic node convergence criterion (Real $\geq$ 0.0; Default 1.0e-3 degrees) (SimX: Default 1.0e-4 for transient)
BALENG	Allowable system energy imbalance (Real $\geq$ 0.0; Default 0.0 energy/time)
NLOOP	Number of iterations allowable (Integer $\geq$ 0; Default 5000 loop) (SimX: Default 50 for transient)
DAMPD	Damping constant (Real $\geq$ 0.0; Default 0.0 non dimensional)
GRVCON	Gravitation constant (Real $\geq$ 0.0; Default 9.81 length/time <sup>2</sup> )
TIMEND	Problem end time (Real; Default 3600.0 time)
OUTPUT	Output interval (Real $\geq$ 0.0; Default 60.0 time)
CSGFAC	Time step control factor (Real $\geq$ 0.0; Default 1.0 non dimensional)
DTIMEI	Time step (Real $\geq$ 0.0; Default 0.0 time)

**Remarks:**

1. This entry is for RC Network solver only.
2. ID can be referenced by the case control command. Different cases may reference different RCPARM entries. It is possible to have multiple RCPARM entries inside one BDF file.
3. RCNS stands for RC Network Steady-state (static), RCNT stands for RC Network Transient
4. The default values are for Nastran only. The modeler (Patran or SimXpert) will have their default values. Some are related with the model units. The default values will not be all 0.0.
5. Default solvers and result checking SNSOR and SNDUFR are the default steady and transient solvers. They are good for most of the cases. Users are encouraged to check the results by the following methods.

Obtain nearly identical results with a different solver

Obtain nearly identical results with tighter convergence

Obtain nearly identical results with a smaller time step (transient)

Obtain nearly identical results with more rays or different random seeds (external radiation solvers)

6. About non-default solvers:

For non-spacecraft, non-radiation problems:

Solid type model – SNSOR, SNTSM1

For spacecraft/radiation models:

SNSOR (with user-specified DAMPD if necessary)

FWDBKL if thermo-stats are present

ATSDUF, SNTSM1 for most other cases

7. About the time step
  - a. The default computed time step (DTIMEU) = CSGMIN\* CSGFAC. CSGMIN can be checked in the .sot file. If CSGFAC is not specified, it is internally set to 1.0. The DTIMEU in the .sot file has been truncated by the OUTPUT point.
  - b. In a normal sized model, CSGMIN usually is small enough for the time step which will assure a convergent transient run.
  - c. CSGFAC is used to adjust the time step. It is recommended to determine the best CSGFAC to the model while maintaining acceptable temperature errors.
  - d. If  $OUTPUT < CSGFAC * CSGMIN$  or  $OUTPUT < DTIMEI$ , then OUTPUT becomes the time step. All the OUTPUT points are automatically required to be calculated.
  - e. DTIMEI is the forced time step which will ignore any other factors. Sometimes it may lead to inaccurate answer if it is too large. DTIMEI does not affect the automatic time step solvers.
  - f. If the model size is very small, CSGMIN may be too big for the time step. A small CSGFAC or DTIMEI should be used to adjust the time step.
  - g.  $CSGFAC * CSGMIN$  or DTIMEI should be small enough to “catch” any details in time fields, temperature fields or orbital flux arrays.

8. For more details about these parameters, please reference *MSC SINDA User's Guide and Library Reference*.

**RCROSS****Cross-Power Spectral Density and Cross-Correlation Functions Output**

Defines a pair of response quantities for computing the cross-power spectral density and cross-correlation functions in random analysis.

**Format:**

1	2	3	4	5	6	7	8	9	10
RCROSS	SID	RTYPE1	ID1	COMP1	RTYPE2	ID2	COMP2	CURID	

**Example:**

RCROSS	10	DISP	100	3	STRESS	200	10	2	
--------	----	------	-----	---	--------	-----	----	---	--

Describer	Meaning
SID	Case Control RCROSS identification number for cross-power spectral density function and cross-correlation function. (Integer > 0)
RTYPEi	Type of response quantity. At least one field must be selected. See Remark 2. (Character or blank)
IDI	Element, grid or scalar point identification number. (Integer > 0)
COMPi	Component code (item) identification number. See Remark 3. (Integer > 0)
CURID	Curve identification number. See Remark 4. (Integer ≥ 0 or blank)

**Remarks:**

1. This entry is required for computing the cross-power spectral density function and cross-correlation function. SID must be selected with the Case Control command (RCROSS = SID). Fields RTYPE1, ID1, and COMP1 represent the first response quantity, and fields RTYPE2, ID2, and COMP2 the second in the correlation.
2. The keywords for field RTYPEi are listed as follows:

Keyword	Meaning
DISP	Displacement Vector
VELO	Velocity Vector
ACCEL	Acceleration Vector
OLOAD	Applied Load Vector
SPCF	Single-point Constraint Force Vector

Keyword	Meaning
MPCF	Multi-point Constraint Force Vector
STRESS	Element Stress
STRAIN	Element Strain
FORCE	Element Force

If anyone of RTYPE1 or RTYPE2 is blank, then the default is the one same as the other field.

- For elements, the item code COMP<sub>i</sub> represents a component of the element stress, strain, and force and is described in Tables [Element Stress-Strain Item Codes](#) and [Element Force Item Codes Part 1](#). For an item having both a real and imaginary part, the code of the real part must be selected. This is required for computing both the cross-power spectral density function and cross-correlation function.

For grid point, the item code is one of 1, 2, 3, 4, 5, and 6, which represent the mnemonics T1, T2, T3, R1, R2, and R3, respectively. For scalar point, always use 1.

- Field CURID is optional. It is for the user's convenience to identify the output by using a single index.

Defines the reference rotor for non-reference rotors in residual (SE 0). Applicable only for Complex Eigenvalue analysis (SOL 107 / 110 / 200 / 400 (ANALYSIS=DCEIG or MCEIG))

## REFROT

Defines the reference rotor for non-reference rotors in residual (SE 0). Applicable only for Complex Eigenvalue analysis (SOL 107 / 110 / 200 / 400 (ANALYSIS=DCEIG or MCEIG))

Specifies the reference rotor for non-reference rotors.

### Format:

1	2	3	4	5	6	7	8	9	10
REFROT	RID	REFROTID	NFROTID1	NFROTID2	NFROTID3	NFROTID4	NFROTID5	NFROTID6	
	NFROTID7	NFROTID8	-etc.-						

or

1	2	3	4	5	6	7	8	9	10
REFROT	RID	REFROTID	NFROTID1	THRU	NFROTIDn	BY	INC		

### Example:

REFROT	100	10	20	35	1	2	60	88	
	99	4001	100						

or

REFROT	100	10	20	THRU	100	BY	10		
--------	-----	----	----	------	-----	----	----	--	--

Describer	Meaning
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required; no Default). See Remark 1.
REFROTID	Specifies the reference rotor ID for all non-reference rotors entered in NFROTIDn field. (Integer > 0; Required; no Default)
NFROTIDi	Specifies the non-reference rotor ID. (Integer > 0; Required; no Default)
THRU	Specifies a range of identification numbers. (Optional)
BY	Specifies an increment for a THRU specification (Optional)
INC	ID number increment. (Integer)

### Remarks:

- Any REFROT bulk data entries with RID will be invoked for any subcase having a RGYRO=RID case control command. Multiple REFROT entries with same RID are allowed.
- REFROTID and NFROTID fields assume the rotor IDs are in the residual structure (SE 0).
- Any rotor whose id is used as REFROTID (in 3<sup>rd</sup> field) should have a valid RGYRO entry. REFROTID should match the RGYRO entry's REFROTR (4<sup>th</sup> field) entry. The 9<sup>th</sup> field in that RGYRO call should be 0 (meaning REFROTR rotor should be in the residual structure).

4. The same NFROTIDi (non-reference rotor id) should not be used on more than one REFROT entry. If a NFROTIDi is defined more than once, a User Fatal message will be issued. This is to make sure that each non-reference rotor can have only one reference rotor.
5. NFROTIDi (non-reference rotor ID) should not be used on any valid RGYRO definition.
6. All rotors in the residual structure (SE 0) should have either an individual RGYRO definition for it be reference rotor (or) use a REFROT entry to identify the reference rotor. If more than one RGYRO entry is used and a rotor does not have either one of these required valid definitions, a User Fatal Message will be issued. Alternately use only one RGYRO entry, which will be used as the reference rotor for all remaining rotors in the model.



Define the reference rotor for non-reference rotors in any superelement. Applicable only for Complex Eigenvalue analysis (SOL 107 / 110 / 200 / 400 (ANALYSIS=DCEIG or MCEIG))

**REFRTSE** Define the reference rotor for non-reference rotors in any superelement. Applicable only for Complex Eigenvalue analysis (SOL 107 / 110 / 200 / 400 (ANALYSIS=DCEIG or MCEIG))

Specifies the reference rotor for non-reference rotors in superelements.

**Format:**

1	2	3	4	5	6	7	8	9	10
REFRTSE	RID	REFROTID	REFROTSE	NFROTID1	NFROTSE1	NFROTID2	NFROTSE2	NFROTID3	
	NFROTSE2	NFROTID4	NFROTSE4						

**Example:**

REFRTSE	100	10	20	35	1	2	0	88	
	10	4001	100						

Describer	Meaning
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required; no Default). See Remark 1.
REFROTID	Specifies the reference rotor ID for all non-reference rotors entered in NFROTIDn field. (Integer > 0; Required; no Default)
REFROTSE	Specifies the reference rotor REFROTID Superelement ID (Integer; Default=0)
NFROTIDi	Specifies the non-reference rotor ID. (Integer > 0; Required; no Default)
NFROTSEi	Specifies the non-reference rotor Superelement ID (Integer; Default=0)

**Remarks:**

- REFRTSE bulk data entries with RID will be invoked for any subcase having RGYRO=RID. Multiple REFRTSE entries with the same RID are allowed.
- The rotor ID used as REFROTID (in 3<sup>rd</sup> field) and its superelement ID used as REFROTSE (in 4<sup>th</sup> field) should have a valid RGYRO entry. Meaning, REFROTID field should match RGYRO's REFROTR (4<sup>th</sup> field) and REFROTSE entry should match with same RGYRO's ROTRSEID (9<sup>th</sup> field).
- The same NFROTIDi and NFROTSEi combination (non-reference rotor in any superelement) should not be used in more than one REFRTSE definition with the same RID. If defined, User Fatal message will be issued. This is to make sure that non-reference rotors can have only one reference rotor.
- NFROTIDi and NFROTSEi combination (non-reference rotor id) should not be used on a valid RGYRO definition.

5. All rotors defined in any superelement should have either individual RGYRO definition for it if it is a reference rotor (or) use REFRTSE definition to identify its reference rotor. If a rotor does not have either one of these required valid definitions and there is more than one RGYRO entry with the RID, a User Fatal Message will be issued. Alternately use only one RGYRO entry, which will be used as the reference rotor for all remaining rotors in the model.

## RELEASE Superelement Boundary Grid Point Release

Defines degrees-of-freedom for superelement exterior grid points that are not connected to the superelement.

### Format:

1	2	3	4	5	6	7	8	9	10
RELEASE	SEID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	-etc.-						

### Example:

RELEASE	15	456	3	7	11	2	156	9	
	152	162							

### Alternate Formats and Examples:

RELEASE	SEID	C	G1	"THRU"	G2				
RELEASE	6	2	15	THRU	127				

(ALL must be in FIELD 4 and no continuation is allowed)

RELEASE	SEID	C	"ALL"						
RELEASE	127	156	ALL						

Describer	Meaning
SEID	Superelement identification number. (Integer > 0)
C	Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)
Gi	Grid point identification numbers. (Integer > 0; "THRU", or "ALL"; For THRU option, G1 < G2.)

### Remarks:

1. A grid point referenced on this entry must be an exterior grid point of the superelement referenced on the entry.
2. In the first alternate format, all grid points in the sequence G1 through G2 are not required to be exterior grid points. Such grid points will collectively produce a warning message but will otherwise be ignored.
3. If the "ALL" option is used, all exterior grid points are released for the referenced degrees-of-freedom.
4. The RELEASE entry is applicable to only the superelement solution sequences (SOLs 101 through 200). It may not reference the residual structure (SEID = 0).
5. This entry is not supported for partitioned superelements.

## RELEX

## External Definition of a Rigid Ellipsoid- SOL 700

Defines a rigid ellipsoid whose properties and motion are defined by either ATB. Used in SOL700 only.

### Format:

1	2	3	4	5	6	7	8	9	10
RELEX	NAME	PROG					etc		

### Example:

RELEX	HEAD	ATB							
-------	------	-----	--	--	--	--	--	--	--

Field	Content
NAME	<p>This name is used within the input file to define the interactions between the external ellipsoid and grid points and rigid bodies. This name is also used in the output requests.</p> <p>When coupled to ATB, the name must correspond to the name of the ATB segment. (Character; required)</p>
PROG	<p>Name of the external program. (Character; required)</p> <p>ATB SOL700 runs coupled with ATB</p>

**RESTART**

## Restart Data for Marc Executed from SOL 600

Specifies writing or reading of restart data for Nonlinear Analysis when Marc is executed from SOL 600. If this Bulk Data entry is found in the job stream, the type of “restart” specified by KIND and KTYPE will be performed. Only one RESTART entry is allowed.

**Format:**

1	2	3	4	5	6	7	8	9	10
RESTART	ID	KIND	NINC	NBEGIN					
	NAME		TSTEP	ENDTIME	NSTEPS	NDCYC	STEPMAX	PERCENT	

**Example(s):**

RESTART	101	1	1						
---------	-----	---	---	--	--	--	--	--	--

The above example writes a restart file. The original run named is abcde.dat.

RESTART	201	3	1	15					
	abcde								

The above example restarts the original abcde.dat run. The name of the restart run must not be abcde.dat in this case.

RESTART	151	2		11					
	my_first	_run							

The above example reads the restart file and prints out results not printed in the original run.

Descriptor	Meaning
ID	Identification number of the restart entry -- Not presently used (Integer)
KIND (2,1)	Type of restart (Integer > 0; required field; no Default) 1 = Write a restart file 2 = Restart a previous analysis (Read an existing restart file) 3 = Restart a previous analysis and write new data on restart file. 11 = Only write restart file for the last converged increment of the run. 12 = Read a restart file written with KIND=11 13 = Read a restart file written with KIND=11 and write the last increment or time step of the present run on that file as well.
NBEGIN (2,3)	The “time” increment at which the restart run begins (used only if KIND=2 or 3). (Integer > 0. See Remarks 3., 4.)
NINC (2,2)	Number of increments between writing of restart data. (Integer > 0; Default = 1)

Descriptor	Meaning
IPRINT (2,8)	Set this field to 1 if the restart data is to be printed (All data from increment INCBEG to LAST will be printed if IPRINT=1). Use this option if printing on a previous run was suppressed but now is desired. (Integer $\geq 0$ ; Default = 0)
LAST (2,9)	The last time increment is printed if IPRINT=1, otherwise this field is ignored. (Integer $> 0$ , Default = 0. See Remarks 3., 4.)
NAME	Name of input file for the original MSC Nastran run without extension. NAME is limited to 16 characters and may not contain imbedded blanks. If the small field format is used, NAME may span fields 2 and 3 of the continuation entry. If the large field is used, NAME should be coded in the 2nd field. NAME is required for a restart run, see Remarks 3., 7. If NAME exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large field fixed or free-format.
TSTEP [2.1]	Time step size after restart -- For dynamic problems only. (Real $\geq 0.0$ or blank; if the value is $\leq 0.0$ the original step size is used)
ENDTIME [2,2]	Ending time for this restart run (Real $\geq 0.0$ or blank, if the value is $\leq 0.0$ the original end time is used)
NSTEPS [2,3]	Total number of time steps for the restart run plus the original run. (Integer $\geq 1$ ; Default = 1)
NDCYC [2,6]	Desired number of cycles if AUTO INCREMENT options were specified on the original run. (Integer $\geq 0$ ; if the value is $\leq 0$ the original is used)
STEPMAX [2,7]	Maximum step size if AUTO INCREMENT options were specified on the original run. (Real $\geq 0.0$ ; if the value is $\leq 0$ the original is used)
PERCENT [2,8]	Percentage of total load to be applied. (Real $\geq 0.0$ ; if the value is $\leq 0$ the original is used)

**Remarks:**

1. RESTART is available only when Marc is executed from within MSC Nastran Implicit Nonlinear (SOL 600).
2. There should only be one RESTART entry in the bulk data. If more than one exists, the first one will be used.
3. (i,j) Indicate the field in Marc's RESTART model definition options. [i,j] Indicate the field in Marc's REAUTO model definition options.
4. The jid.marc.t16 and jid.marc.t08 files must be saved from the first run when a restart run is executed. Both original and restart runs must be located in the same directory. File extensions are .t08 and t16.
5. A restart run may not have the same input file name (jid) as that of the original run. In other words, if the input file for the original run is named abcd.dat, the input file for a restart run may not be named abcd.dat.

6. For static analyses, normally each load case has a total time of 1.0. The first case goes from 0.0 to 1.0, the second from 1.0 to 2.0, etc. If the first run has two static load cases and terminates at 1.6, it is in the middle of the second load case. The original run should be examined to determine which increment (before 1.6) to begin the restart run.
7. In the second example above, the original run was named `my_first_run.dat` (or `first_run.bdf`, etc). This is a small field example. There are 8 characters in the 2nd field of the continuation line and 4 characters in the third field. The name can start anywhere within the two fields. There must not be any blank spaces in the name. If the name exceeds 8 characters, the continuation line must be coded in small-field fixed format or in large-field fixed or free format. NAME is limited to a maximum of 16 characters.
8. Both original and restart run names (jid's) should use only lower case letters except on computer systems that are not case sensitive. "NAME" (continue line, fields 2-3) will be converted automatically to lower case.
9. The restart run must use the same values of Marc's DIST LOAD parameter as the original run or it may fail. Be sure to use parameters MARCDIS2, MARCDIS3 and MARCDIS4 to set these values to be exactly the same as the original run (examine `jid.marc.dat` from the original run to determine these values before submitting the new run).
10. The RSF options creates a complete database which is necessary for restarts. When RSF=0 is specified, the same file is overwritten after each interval. When RSF=1 is specified, a new restart file is created after each interval, thus a "family" of files is created numbered sequentially `jid.dytr.d3dump10`, `jid.dytr.d3dump02`, etc. These files can take significant disk space but are important if a model might need to be modified prior to the end time.
11. It is not necessary to enter continuation lines if not needed for the particular job to be run.
12. Restarts are not available for models with bolt loading using entries MBOLT, MBOLTUS or BOLT.

**RFORCE**

## Rotational Force

Defines a static loading condition due to an angular velocity and/or acceleration.

**Format:**

1	2	3	4	5	6	7	8	9	10
RFORCE	SID	G	CID	A	R1	R2	R3	METHOD	
	RACC	MB	IDRF						

**Example:**

RFORCE	2	5		-6.4	0.0	0.0	1.0	2	
	1.0								

Describer	Meaning
SID	Load set identification number. (Integer > 0)
G	Grid point identification number through which the rotation vector acts. (Integer ≥ 0)
CID	Coordinate system defining the components of the rotation vector. See Remark 16. (Integer ≥ 0; Default = 0)
A	Scale factor of the angular velocity in revolutions per unit time. (Real)
R1, R2, R3	Rectangular components of rotation vector $\vec{R}$ . The vector defined will pass through point G. (Real; $R1^2 + R2^2 + R3^2 > 0.0$ unless A and RACC are both zero)
METHOD	Method used to compute centrifugal forces due to angular velocity, see Remark 20.. For angular acceleration, see Remark 13. (Integer = 1 or 2; Default = 1)
RACC	Scale factor of the angular acceleration in revolutions per unit time squared. (Real; Default = 0.0)
MB	Indicates whether the CID coordinate system is defined in the main Bulk Data Section (MB = -1) or the partitioned superelement Bulk Data Section (MB = 0). Coordinate systems referenced in the main Bulk Data Section are considered stationary with respect to the assembly basic coordinate system. See Remark 15. (Integer; Default = 0)
IDRF (SOL 600 only)	ID indicating to which portion of the structure this particular RFORCE entry applies. It is possible to have multiple RFORCE entries in the same subcase for SOL 600 to represent different portions of the structure with different rotational accelerations. IDRF corresponds to a SET3 entry specifying the elements with this acceleration. A BRKSQL entry may also be specified with a matching IDRF entry. (Integer; Default = 0)



**Remarks:**

1. The forces that are created with the RFORCE entry act on the structure as follows: the forces that are defined with the RFORCE entry for a constant angular velocity (A), act in the positive radial direction. These forces represent the inertia forces on the structure due to a constant angular velocity. The forces that are defined with the RFORCE entry for a constant angular acceleration (RACC), act in the same direction as the angular acceleration. These forces would be opposite to the inertia forces on the structure due to a constant angular acceleration. In Figure 9-139, the force vector at grid point Gi is given by

$$\{\vec{F}\}_i = [m]_i[\vec{\omega} \times (\vec{\omega} \times (\vec{r}_i - \vec{r}_a)) + \vec{\alpha} \times (\vec{r}_i - \vec{r}_a)] \quad (9-29)$$

where:

$$\begin{aligned} \text{angular velocity} &= \vec{\omega} = 2\pi A \cdot \vec{R} \text{ (radians/unit time)} \\ \text{angular acceleration} &= \vec{\alpha} = 2\pi RACC \cdot \vec{R} \text{ (radians/unit time}^2\text{)} \\ [m]_i &= 3 \times 3 \text{ translational mass matrix at grid point Gi} \end{aligned}$$

**Note:** The equation for  $\vec{F}_i$  will have additional terms if the mass is offset or  $I_{23}^i, I_{13}^i$  terms exist relative to the rotation axes and METHOD = 1 is selected.

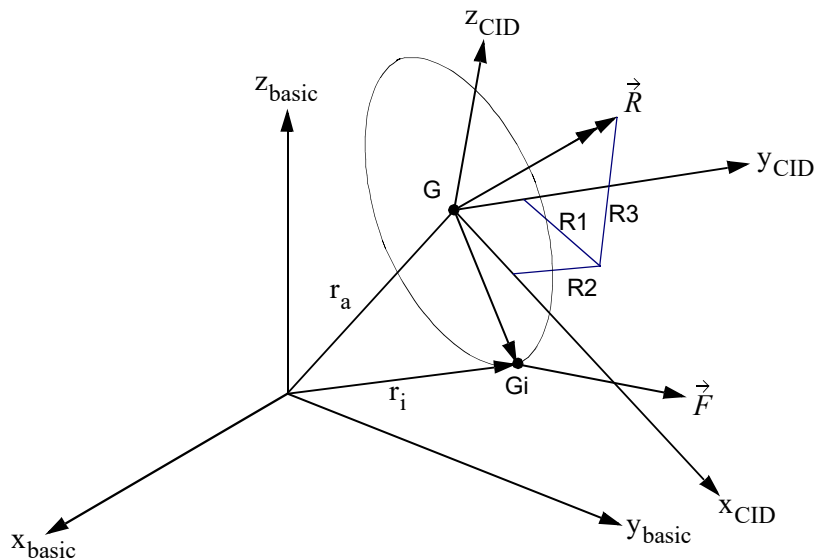


Figure 9-139 RFORCE Vector at Grid Point Gi

2. In the static solution sequences, SID must be selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
3.  $G = 0$  signifies that the rotation vector acts through the origin of the basic coordinate system.
4.  $CID = 0$  (Default) signifies that the rotation vector is defined in the basic coordinate system.
5. The load vector generated by this entry can be printed with an OLOAD command in the Case Control Section.
6.  $METHOD = 1$  yields correct results only when there is no coupling in the mass matrix. This occurs when the lumped mass option is used with or without the ZOFFS option (see the CQUAD4 entry for a description of ZOFFS).  $METHOD = 2$  yields correct results for lumped or consistent mass matrix only if the ZOFFS option is not used. The acceleration terms due to the mass offset ( $X1$ ,  $X2$ ,  $X3$ ) on the CONM2 entry are not computed with  $METHOD = 2$ . All the possible combinations of mass matrices and offset and the correct method to be used are shown below.

	No Offset	Offset
Lumped	METHOD = 1 or METHOD = 2	METHOD = 1
Coupled	METHOD = 2	Neither

7. In cyclic symmetry analyses, the T3 axis of the basic coordinate system must be coincident with the axis of symmetry. In the DIH type of cyclic symmetry, the T1 axis also must be parallel to side 1 of segment 1R of the model.
8. For superelement analysis, G should reference a residual structure point that is exterior to all superelements when loading Superelement 0 and a separate G must be interior to each superelement when loading an upstream superelement else centrifugal loads will not be generated for that superelement. However, in cyclic analysis, User Fatal Message 4347 will be issued.
9. In a geometric nonlinear static analysis (SOL 106 and 400 when PARAM LDGISP is set to +1), this type of loading is a follower force type of loading. However, the orientation of coordinate system CID is not updated.
10. In nonlinear static solutions when there is more than one increment (INC) specified on the NLPARM entry for a given subcase, the load vector resulting from the RFORCE input (and not the angular velocity vector) is scaled linearly. This means that loading by increments in the angular velocity can only be achieved by having subcases where the RFORCE loading is applied in a single increment.
11. The continuation entry is optional.
12. Forces due to angular acceleration (RACC) are computed with  $METHOD = 2$  even if  $METHOD = 1$  is specified.
13. Loads derived from this entry do not include effects due to mass specified for scalar points.

14. The follower force effects due to loads from this entry are included in the stiffness in all linear solution sequences that calculate a differential stiffness. The solution sequences are SOLs 103, 105, 107 to 112, 115 and 116 (see also the parameter [FOLLOWK](#) (Ch. 6)). In addition, follower force effects are included in the force balance in the nonlinear solution sequences, SOLs 106, 129, 153, 159 and 400, if geometric nonlinear effects are turned on with `PARAM, LGDISP, 1`. The follower force stiffness is included in the nonlinear static solution sequences (SOLs 106, 153 and 400) and general nonlinear solution sequence (SOL 400) for both `ANALYSIS=NLSTAT` and `NLTRAN` but not in the nonlinear transient dynamic solution sequences (SOLs 129 and 159).
15. The coordinate systems in the main Bulk Data Section are defined relative to the assembly basic coordinate system which is fixed. This feature is useful when a superelement defined by a partitioned Bulk Data Section is rotated or mirrored and the gravity load is more conveniently defined in terms of coordinates which are fixed.
16. If `CID` is not a rectangular coordinate system, RFORCE will treat it as if it were and unexpected answers may result.
17. Follower force stiffness (`param,followk,yes`) is supported for method 2 only.
18. Multiple RFORCE entries with different `SID`'s may be used in SOL 600 in the same subcase or SOL 400 in the same loadcase (`STEP` and/or `SUBCASE`). They should be combined using the `LOAD` entry similar to the way `FORCE` or `PLOAD4` with different `ID`'s are combined.
19. Fields `CID`, `METHOD`, `RACC`, `MB` and `IDRF` will be ignored for SOL 700.
20. For Axisymmetric Harmonic elements, (elements selected with `PAXSYMH` entry), `METHOD=2` is required. A selection of `METHOD=1` will automatically be overridden and be replaced by `METHOD=2` for all elements of the model.
21. Note that the `LOAD` entry scaling for RFORCE uses the square root of the absolute  $S \times S_i$  in rotor dynamics. Thus it is recommended that the `LOAD` entry not be used with RFORCE in rotor dynamics with a value other than  $S=1.0$  and  $S_i=1.0$  value. See Remark 6. of the `LOAD` entry.
22. If Modules are present then this entry may only be specified in the main Bulk Data section.

## RGYRO

### Rotordynamic Analysis Parameters

Specifies synchronous or asynchronous analysis, reference rotor, and rotation speed of the reference rotor.

#### Format:

1	2	3	4	5	6	7	8	9	10
RGYRO	RID	SYNCFLG	REFROTR	SPDUNIT	SPDLOW	SPDHIGH	SPEED	ROTRSEID	
	WR3WRL	WR4WRL	WRHWRL						

#### Example:

1	2	3	4	5	6	7	8	9	10
RGYRO	100	SYNC	1	RPM	1000.0	5000.0		10	

Describer	Meaning
RID	Identification number of RGYRO entry. Selected by Case Control command, RGYRO. (Required; no Default). See Remarks 1. and 4.
SYNCFLG	Specifies whether the analysis is synchronous or asynchronous analysis. Required input for frequency response and complex modes analyses. Not required for static analyses. (Character: 'SYNC', 'ASYNC', or blank). See Remarks 4. through 4.
REFROTR	Specifies the reference rotor ID for the analysis. (Integer > 0; Required; no Default)
SPDUNIT	Specifies whether the entries SPDLOW, SPDHIGH, and SPEED are given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/unit time). (Character: 'RPM' or 'FREQ'; no Default). See Remark 4.
SPDLOW	Specifies the low speed for synchronous analysis. See Remark 4. (Real; Default = 0.0)
SPDHIGH	Specifies the high speed for synchronous analysis. See Remark 4. (Real; Default = 99999.0)
SPEED	Specifies reference rotor speed for asynchronous analysis. Also required for static analyses. See Remark 4. (Default = 0)
ROTRSEID	Identification number of the superelement in which the rotor specified in the REFROTR field is defined. (Integer >= 0, Default = 0).
WR3WRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field for asynchronous complex eigenvalue analysis. (Real, Default = 0.)
WR4WRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries for asynchronous complex eigenvalue analysis. (Real, Default = 0.)
WRHWRL	Specifies whirl frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card for asynchronous complex eigenvalue analysis. (Real, Default = 0.)

**Remarks:**

1. Multiple RGYRO entries with the same RID value are allowed. All RGYROs with same RID will be analyzed for either SYNC (or) ASYNC option only.
2. For multiple RGYRO with same RID entries, reference rotors ID specified in REFROTR (4<sup>th</sup>) field are treated as individual rotors. Non-reference rotors (rotor ID not defined in RGYRO) will need REFROT (or) REFRTSE bulk data card to identify its reference rotor.
3. If single RGYRO entry with RID is used, then all rotors in the models will reference the rotor id specified in REFROTR (4<sup>th</sup>) field.
4. The required information on the RGYRO entries varies for different analyses. Values for the RID and SPDUNIT fields are always required. Values for SPDLOW, SPDHIGH and SPEED are analysis dependent as shown in the table below:

Solution Sequence	Type of Analysis	PARAM, GYROAVG	Required Entry	COMMENT
Frequency	SYNC	0	None	--
Response	SYNC	-1	SPDLOW, SPDHIGH	a, b
	ASYNC	0	SPEED	--
	ASYNC	-1	SPEED	b
Complex Modes	SYNC	--	SPDLOW, SPDHIGH	a, b
	ASYNC	--	SPEED	b, Remark 4.
Static Analysis	--	--	SPEED	Remark 3.

- a. The relative rotor speeds will be treated as linearly dependent on the reference rotor speed ( $\Omega = A0 + A1\Omega_{\text{reference}}$ ). The scale factors A0 and A1 will be determined by a least-mean-square fit of the relative rotor speeds input on the RSPINR entries between SPDLOW and SPDHIGH of the reference rotor. If SPDLOW or SPDHIGH are outside the range specified on the RSPINR entry, the values will be extrapolated from the RSPINR entry values.
  - b. PARAM, WR3 and PARAM, WR4 are required for rotor damping.
5. For static analysis, the SYNCFLG field must be left blank or must have the ASYNC option specified in it. Otherwise, the program terminates the execution with an appropriate fatal message.
6. When there is a Case Control request for Campbell diagram, the selected RGYRO Bulk Data entry must have the ASYNC option specified in its SYNCFLG field. Otherwise, the program terminates the execution with an appropriate fatal message.
7. In the presence of GR field, the rotor damping matrix (BR) in complex eigenvalue analysis is given by:

$$BR = KR1.GR1/[\Omega1(\Omega_{\text{ref}}) - WR3WRL]$$

where,

KR is the rotor stiffness matrix.

GR is the structural damping parameters specified in the RSPINR entry

$\Omega_1$  ( $\Omega_{ref}$ ) is the specified rotor speed.

8. If non-zero WR3WRL is specified in RGYRO, then PARAM,WR3 and WR3R (in RSPINR) cannot be specified. Appropriate FATAL message will be issued, in case these entries are specified.
9. WR3WRL/ WR3WRL/ WRHWRL affect both the rotor damping terms and the circulation terms. For the fixed reference frame, the circulation terms are generated from the rotating structure. For the rotating reference frame, the circulation terms are generated from the stator.

## RINGAX Conical Shell Ring

Defines a ring for conical shell problems.

### Format:

1	2	3	4	5	6	7	8	9	10
RINGAX	ID		R	Z			PS		

### Example:

RINGAX	3		2.0	-10.0			162		
--------	---	--	-----	-------	--	--	-----	--	--

Describer	Meaning
ID	Ring identification number. See Remark 6. (Integer > 0)
R	Ring radius. (Real > 0.0)
Z	Ring axial location. (Real)
PS	Permanent single-point constraints. (Any unique combination of the Integers 1 through 6 with no embedded blanks.)

### Remarks:

1. RINGAX is allowed only if an AXIC entry is also present.
2. The number of degrees-of-freedom defined is  $(6 - \text{NPS}) \cdot H$  where  $H$  is the harmonic count and NPS is the number of digits in field 8. (See [AXIC](#)).
3. RINGAX identification numbers must be unique with respect to all other POINTAX, RINGAX, and SECTAX identification numbers.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.
5. Constraints may be necessary to avoid matrix singularities. The CONEAX element has no stiffness for rotation about the normal. In addition, there is no stiffness for rotation about V (see [Figure 9-140](#)) when transverse shear flexibility is not included.

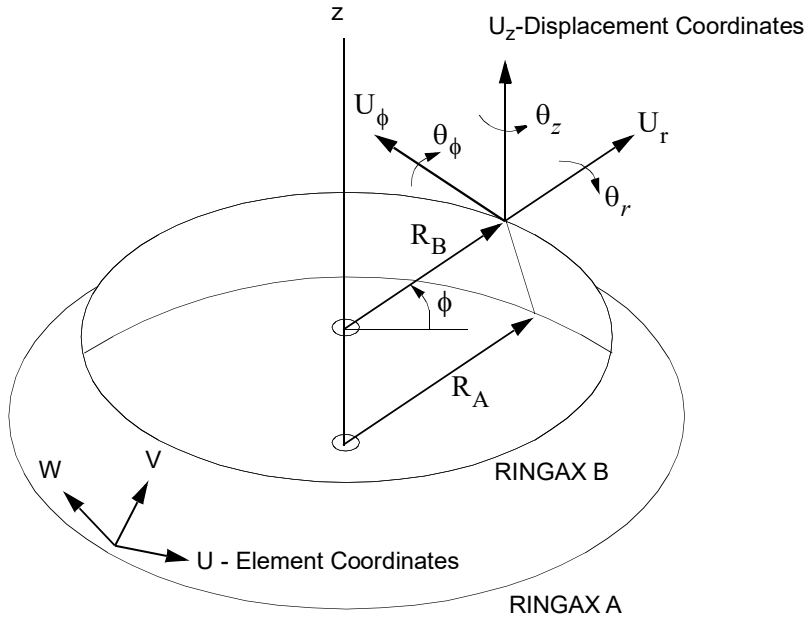


Figure 9-140 RINGAX Coordinate System

6. In order to reference this entry on a SET Case Control command, the ID must be modified by  $ID(n) = ID + 1000000 \cdot n$  where  $n$  is the harmonic number plus one and  $ID(n)$  is the value specified on the SET entry.



## RINGFL

## Axisymmetric Fluid Point

Defines a circle (fluid point) in an axisymmetric fluid model.

### Format:

1	2	3	4	5	6	7	8	9	10
RINGFL	IDFA	XA1	XA2	XA3	IDFB	XB1	XB2	XB3	

### Example:

RINGFL	3	1.0		30.0					
--------	---	-----	--	------	--	--	--	--	--

Describer	Meaning
IDFA, IDFB	Unique identification number of the fluid points. (0 < Integer < 500000)
XAi, XBi	Coordinates of the point defined in the coordinate system specified on the AXIF entry. (Real; XA1 and XB1 > 0.0)

### Remarks:

1. RINGFL is allowed only if an AXIF entry is also present.
2. All fluid point identification numbers must be unique with respect to other scalar, structural, and fluid points.
3. X1, X2, X3 are (r,  $\phi$ , z) for a cylindrical coordinate system and ( $\rho$ ,  $\theta$ ,  $\phi$ ) for a spherical coordinate system.  $\theta$  is in degrees. The value of  $\phi$  must be blank or zero.
4. One or two fluid points may be defined per entry.

**RJOINT**

## Rigid Joint

Defines a rigid joint element connecting two coinciding grid points.

**Format:**

1	2	3	4	5	6	7	8	9	10
RJOINT	EID	GA	GB	CB					

**Example:**

RJOINT	5	1	2	12345					
--------	---	---	---	-------	--	--	--	--	--

Describer	Meaning
EID	Element identification number. (Integer > 0)
GA, GB	Grid point identification numbers. (Integer > 0)
CB	Component numbers in the global coordinate system at GB. These degrees-of-freedom are constrained to move with the same degrees-of-freedom at GA. See Remarks 4. and 5. (Integers 1 through 6 with no embedded or blank.)

**Remarks:**

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 12 displacement degrees-of-freedom given by grid points GA and GB. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom given by CB.
- The length between grid points GA and GB must be zero.
- When CB = "123456" or blank, the grid point GB is constrained to move with GA and the two grid points moves as a single point. For default CB = "123456".
- If any degree-of-freedom is released on CB, RJOINT becomes a mechanical joint element. For example, CB = "12345", then RJOINT becomes a hinge. CB = "1234", then RJOINT becomes a universal joint. And CB = "123", RJOINT becomes a spherical joint.
- For the Lagrange method, the theory for the RJOINT is formulated such that a consistent mechanical joint is created even if the user requests different global coordinate systems at grid points GA and GB.
- Thermal expansion effect is not applicable for the RJOINT element, since the distance between grid points GA and GB is zero.
- Element identification numbers should be unique with respect to all other element identification numbers.

## RLOAD1 Frequency Response Dynamic Excitation, Form 1

Defines a frequency-dependent dynamic load of the form

$$\{P(f)\} = \{A\}[C(f) + iD(f)]e^{i\{\theta - 2\pi f\tau\}}$$

for use in frequency response problems.

### Format:

1	2	3	4	5	6	7	8	9	10
RLOAD1	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TC/RC	TD/RD	TYPE		

### Example:

RLOAD1	5	3	2.0	10	1				
--------	---	---	-----	----	---	--	--	--	--

Describer	Meaning
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines $\{A\}$ . See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay $\tau$ . See Remark 2. (Integer > 0 or blank)
DELAYR	Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle $\theta$ . (See Remark 2. (Integer > 0 or blank)
DPHASER	Value of phase angle $\theta$ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TC	Set identification number of the TABLEDi entry that gives $C(f)$ . See Remark 2. (Integer > 0 or blank)
RC	Value of C to be used for all frequencies. See Remark 2.. (Real or blank)
TD	Set identification number of the TABLEDi entry that gives $D(f)$ . See Remark 2. (Integer > 0 or blank)
RD	Value of D to be used for all frequencies. See Remark 2.. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Default = 0)

**Remarks:**

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.
2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, TC/RC, or TD/RD fields are blank or zero, the corresponding  $\tau$ ,  $\theta$ ,  $C(f)$  or  $D(f)$  will be zero. Either TC/RC or TD/RD may be blank or zero, but not both.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control  
EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.
- There is a LOADSET request in Case Control  
The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control  
EXCITEID will reference SPCD entries.
- There is a LOADSET request in Case Control  
The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

**RLOAD2**

Frequency Response Dynamic Excitation, Form 2

Defines a frequency-dependent dynamic excitation of the form.

$$\{P(f)\} = \{A\} \cdot B(f)e^{i\{\phi(f) + \theta - 2\pi f\tau\}}$$

for use in frequency response problems.

**Format:**

1	2	3	4	5	6	7	8	9	10
RLOAD2	SID	EXCITEID	DELAYI/ DELAYR	DPHASEI/ DPHASER	TB/RB	TP/RP	TYPE		

**Example:**

RLOAD2	5	3	15	5.0	7				
--------	---	---	----	-----	---	--	--	--	--

Describer	Meaning
SID	Set identification number. See Remarks 1. and 3. (Integer > 0)
EXCITEID	Identification number of a static or thermal load set or a DAREA or FBALOAD (in FRF Based Assembly or FBA process) or SPCD entry set that defines {A}. See Remarks 4. and 5. (Integer > 0)
DELAYI	Identification number of DELAY or FBADLAY (in FRF Based Assembly or FBA process) Bulk Data entry that defines time delay τ. See Remark 2. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
DPHASEI	Identification number DPHASE or FBAPHAS (in FRF Based Assembly or FBA process) Bulk Data entry that defines phase angle θ. (See Remark 2. (Integer > 0 or blank)
DPHASER	Value of phase angle θ (in degrees) that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 2. (Real or blank)
TB	Set identification number of the TABLEDi entry that gives B(f). (Integer > 0)
RB	Value of B to be used for all frequencies. (Real, non-zero)
TP	Set identification number of the TABLEDi entry that gives φ(f) in degrees. (Integer ≥ 0)
RP	Value of φ to be used for all frequencies. See Remark 2. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 4. and 5. (Integer, character or blank; Defaults = 0)

**Remarks:**

- Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.



2. If any of DELAYI/DELAYR, DPHASEI/DPHASER, or TP/RP fields are blank or zero, the corresponding  $\tau$ ,  $\theta$ , or  $\phi(f)$  will be zero.
3. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
4. The type of the dynamic excitation is specified by TYPE (field 8) according to the following table:

TYPE	TYPE of Dynamic Excitation
0, L, LO, LOA or LOAD	Applied load (force or moment) (Default)
1, D, DI, DIS or DISP	Enforced displacement using SPC/SPCD data
2, V, VE, VEL or VELO	Enforced velocity using SPC/SPCD data
3, A, AC, ACC or ACCE	Enforced acceleration SPC/SPCD data

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 108, 111, 146, 200, and 400. For other solution sequences such as SOL118 (Cyclic Frequency Response) the large mass method must still be used.

5. TYPE (field 8) also determines the manner in which EXCITEID (field 3) is used by the program as described below:

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control  
EXCITEID may reference DAREA, FBALOAD (in FRF Based Assembly or FBA process), static and thermal load set entries.
- There is a LOADSET request in Case Control  
The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control  
EXCITEID will reference SPCD entries.
- There is a LOADSET request in Case Control  
The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID.

6. If Modules are present then this entry may only be specified in the main Bulk Data section.

Define kinks and offsets to be applied on a rotor, pull the rotor on to its bearing and then carry out rotordynamic analysis (SOL 400 only).

**ROT BENT** Define kinks and offsets to be applied on a rotor, pull the rotor on to its bearing and then carry out rotordynamic analysis (SOL 400 only).

Specifies rotor initial deformation due to kink and offset.

**Format:**

1	2	3	4	5	6	7	8	9	10
ROT BENT	Bent ID	Rotor ID	Load ID						
	Unbcord	x1	x2	x3					
	Kink	Station	Kink angle	Phase angle					
	Offset	Station	Offset value	Phase angle					
	Brgdpr	Rotor Grid ID	Stator Grid ID						

**Example:**

1	2	3	4	5	6	7	8	9	10
ROT BENT	101	10	99						
	Unbcord	1.0	0.0	0.0					
	Kink	30.0	0.06	135.0					
	Brgdpr	1	111						
		51	151						
		81	181						

ROT BENT	200	10	1						
	Offset	3.82	0.001	30.0					
		15.54	0.002	130.0					
		28.45	0.003	230.0					
		38.45	0.004	330.0					
	Kink	10.35	0.06	45.0					
		22.02	0.12	145.0					
		33.45	0.18	245.0					
		48.45	0.24	345.0					
	Brgdpr	11	111						
		17	117						
		35	135						
	Unbcord	0.0	-1.0	2.0					

Describer	Meaning
Bent ID	Identification number of ROTBENT. (Integer > 0, required, no default).
Rotor ID	Rotor identification number for which the kinks and offsets are defined. See Remark 2., 3. (Integer > 0, required, no default).
Load ID	Load identification number, to be selected by LOAD case control in the first STEP. See Remark 4. (Integer > 0, required, no default).
Unbcord	Components of the vector to determine the X-Y plane of the rotor system for Kinks and Offsets. See Remark 6. (Real, required, no default).
Kink	Flag indicating that the next 3 fields are for kink definition. See Remark 7. (Character, optional)
Station	Location on rotor from ROTOR ID 1 <sup>st</sup> grid along the rotor axis, from where the Kink starts. (Real, required if kink is specified, no default).
Kink angle	Kink angle based on Unbcord defined rotor Z-axis. See Remark 8. (Real, DEG, required if kink is specified, no default).
Phase angle	Phase angle about the rotor X-axis, based on Unbcord defined rotor X-Y plane. (Real, DEG, required if kink is specified, no default).
Offset	Flag indicating that the next 3 fields are for offset definition. See Remark 7. (Character, optional)
Station	Location on rotor from ROTOR ID 1st grid along the rotor axis, from where the Offset starts. (Real, required if offset is specified, no default).
Offset value	Offset value on the axis of the rotor. (Real, required if offset is specified, no default).
Phase angle	Phase angle based on Unbcord defined rotor X-Y plane. (Real, DEG, required if offset is specified, no default). A phase angle of 0.0 results in an offset in the positive Rotor Y-axis.
Brgdpr	Flag indicating that the next 2 fields are for paired grids. See Remark 12. (Character, required).
GRID IDi	Paired grids ID to indicate paired connection between rotor and support after initial deformation. (Integer > 0, required, no default). First grid should be part of ROTOR & second grid should be part of Stator.

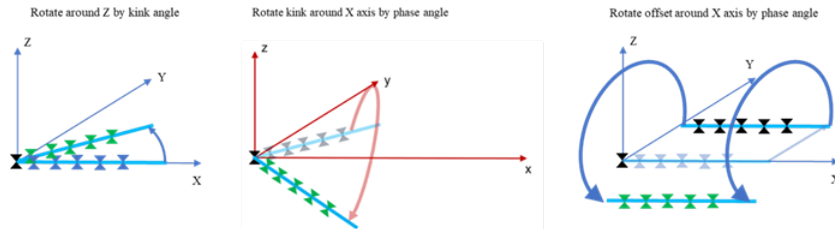
**Remarks:**

1. ROTBENT is supported only in SOL 400 analysis.
2. One ROTBENT entry with unique BENT ID value is allowed. ROTBENT analysis is supported for only one SUBCASE definition, having 2 STEP analyses. STEP1 analysis should be NLSTATIC/NLSTAT. STEP 2 analysis runs rotordynamics analysis (complex eigenvalue, frequency response).
3. ROTBENT only support 1D element rotor defined by ROTOR entry.
4. ROTBENT uses Load ID to perform STEP 1 NLSTATIC analysis.

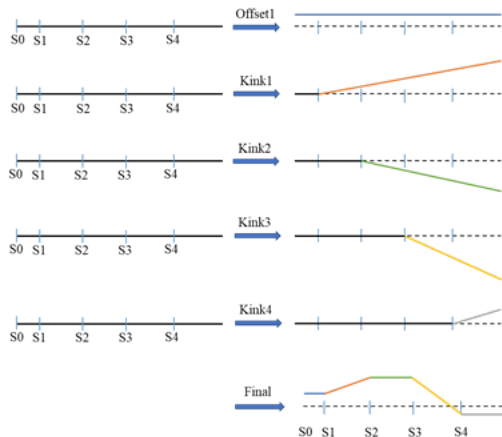


Define kinks and offsets to be applied on a rotor, pull the rotor on to its bearing and then carry out rotordynamic analysis (SOL 400 only).

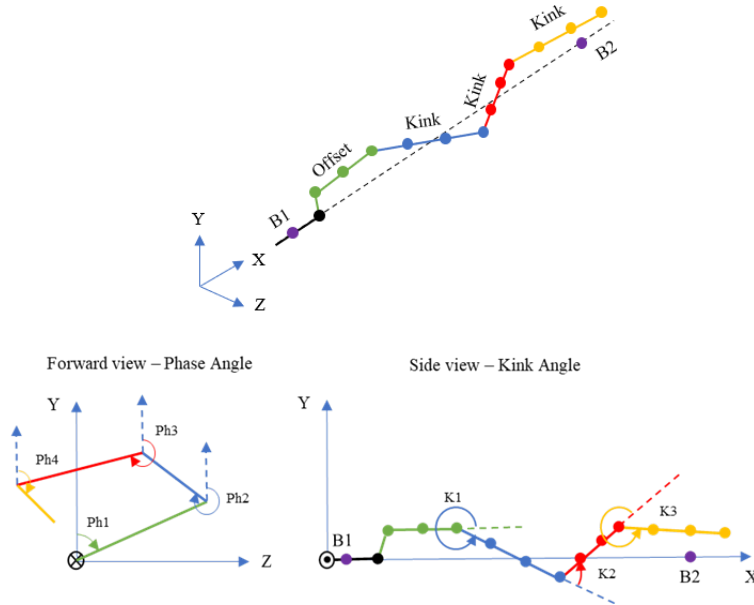
5. In a ROTBENT bulk data card, the keywords (Kink, Offset, or Brgdpr) can only be defined once. Multiple rows for each keyword are allowed, except for Unbcord.
6. ROTBENT can take only one set of 'Unbcord' input, with three REAL values. The first 2 grids in AXIS option of ROTOR entry determine the X-axis of the Rotor system, with origin of this system to be at the first grid entry in AXIS option. Unbcord entries will form the vector used to determine the X-Y plane of Rotor system, defined using the CD of the first grid on AXIS option of ROTOR entry linked to the Rotor ID.
7. A rectangular coordinate (rotor system) formed in remark 6., is used to determine the phase angle, which starts from the X-Y plane based on UNBCORD, and the kink angle, which rotates about Z-axis. In the following figures in the Rotor system, the rotor axis is X, and X-Y plane is determined by the Unbcord vector.



8. Kink is calculated in the rotor system X-Y plane, where the phase is zero (See Remark 7.), by rotating the shaft at the station point about Z-axis to the defined kink angle, then the Kink shaft is rotated about the shaft axis (X) to the defined phase angle from the X-Y plane.
9. Offset is calculated in the rotor X-Y plane, where the phase is zero (See Remark 7.) then the Offset is rotated about the shaft axis (X) to the defined phase angle from the X-Y plane and applied to the rotor.
10. Offsets and Kinks are calculated separately at the shaft axis or the shaft center line. All the Kinks and Offsets are then accumulated to generate the initial unconstrained rotor geometry. Grid coordinate of unconstrained rotor will be printed out in f06 file.



11. Brgdpr entry needs at least one pair of existing GRID IDs. First id of the pair of entries of Brgdpr keyword should be part of referenced rotor and second id should be part of bearing (stator) connected to the rotor. All the first grid ids of each pair of entry should be added to AXIS option of reference ROTOR. It is recommended that both grid ids should not be part of any MPC, RBEs definition.
12. The initial step (NLSTATIC) updates the rotor geometry, writes the bulk data (program internally created MPC, SPOINTs, SPC1 and SPCD) entries to pull the rotor into the bearings and runs the static solution. This static solution step will result in the deformed shape of the model with the rotor pulled into the bearings. At this point, the rotor and bearing GRIDs of Brgdpr entry are coincident.
13. It is advised to have both grid entries of Brgdpr keyword to be at the same axial location. A small Kink angle is assumed ( $<1$  deg) and the deformation in axial direction (axial and angular displacement) of the rotor grids is ignored.
14. The following figures are the brief description of the Kinks and Offsets in a rotor, where B1 and B2 are support locations and are paired with rotor grids in Brgdpr keyword.



## ROTHYBD

## Hybrid damping for rotors

Specifies hybrid damping data for rotors.

### Format:

1	2	3	4	5	6	7	8	9	10
ROTHYBD	ROTORID1	HYBDAMP1	ROTORID2	HYBDAMP2	ROTORID3	HYBDAMP3	ROTORID4	HYBDAMP4	
	ROTORID5	HYBDAMP5	-etc.-						

### Examples:

ROTHYBD	1	15							
ROTHYBD	10	100	20	200	30	300			

Describer	Meaning
ROTORID <sub>i</sub>	Identification number of rotor. (Integer > 0). See Remarks 1 and 2.
HYBDAMP <sub>i</sub>	Identification number of a HYBDAMP entry defining hybrid modal damping data. (Integer > 0). See Remarks 1 and 2.

### Remarks:

1. ROTORID<sub>i</sub> - HYBDAMP<sub>i</sub> pair values referencing non-existent rotors are ignored.
2. If there is no HYBDAMP entry defined in the data for a HYBDAMP<sub>i</sub> specified for a valid ROTORID<sub>i</sub>, the program terminates the execution with an appropriate fatal error.
3. Hybrid damping can result in very densely populated damping matrix causing significant performance penalty.

## ROTOR

### Rotor Model Definition

Specifies list of grids, elements or properties that comprise the rotor 3D model.

#### Format:

1	2	3	4	5	6	7	8	9	10
ROTOR	ROTORID	FRAME							
	LTYPE	ID1	ID2	ID3	etc.				
	AXIS	GID1	GID2	etc.					

#### Example:

ROTOR	10	ROT							
	ELEM	10	THRU	12					
	PROP	1	THRU	5					
	AXIS	101	102						

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0).
FRAME	Analysis frame (Char, ROT or FIX, Required, Default: FIX)
LTYPE	ELEM or PROP or both, indicating whether the specified list references element IDs or property IDs. (Character; Required; No default)
	Note that the order is important. In case both ELEM and PROP are specified, ELEM should be specified first.
Idi	IDs of elements or properties comprising the rotor. (Integer > 0; Required; No default)
AXIS	Defines grid points which define the axis of rotation.
GIDi	IDs of grids comprising the axis of the rotor (Integer > 0; Required; No default)

#### Remarks:

- Supported element types for analysis in rotating reference frame:
  - 0D elements : CONM1, CONM2
  - 1D elements : CBEAM, CBAR
  - 2D elements : CQUAD4, CTRIA3
  - 3D elements : CHEXA, CPENTA, CTETRA, CPYRAM
- Supported element types for analysis in fixed reference frame:
  - 0D elements : CONM1, CONM2
  - 1D elements : CBEAM, CBAR

2D elements : CQUAD4, CQUAD8, CTRIA3, CTRIA6

3D elements : CHEXA, CPENTA, CTETRA, CPYRAM

3. Current limitation for analysis in fixed reference frame:
  - a. If shell elements are defined perpendicular to the rotor axis for a 3D rotor defined using ROTOR entry, then gyroscopic matrix is not available for those elements. For example, a flat disk like CD spinning about the center. If the shell elements are replaced by SOLID elements, then correct gyroscopic effects can be obtained.
  - b. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.
4. THRU option is supported in ROTOR entry. Note that the order is important for LTYPE. In case both ELEM and PROP are specified, ELEM should be specified first.
5. Analysis can be performed using coupled mass or diagonal mass for all the elements.
6. ROTORAX, ROTORG and ROTORSE should not be used along with ROTOR in ROT frame. They can only be used with ROTOR in FIX frame. (since ROTORAX, ROTORG and ROTORSE assume fixed reference frame.)
7. At least 2 grid points need to be defined on AXIS to complete ROTOR definition, these points may not be part of rotor.
8. For unbalance loads the grid point, at which UNBALNC is defined, should be part of ROTOR AXIS list.
9. In order to include CONM1/2 elements as part of a rotor, its element ID should be listed using ELEM in ROTOR definition.
10. Stator portion of the model should only be defined in residual for external superelement runs in rotating system.
11. Only permanent glue option is supported for contact analysis for rotating portion of the model.

## ROTORAX Axisymmetric Model Rotor Definition

Specifies list of elements, properties or grid points that comprise an axisymmetric model rotor.

### Format:

1	2	3	4	5	6	7	8	9	10
ROTORAX	ROTORID	LTYPE	ID1	ID2	ID3	ID4	ID5	ID6	
	ID7	ID8	-etc.-						

or

ROTORAX	ROTORID	LTYPE	ID1	THRU	ID2	BY	INC		

### Example:

ROTORAX	100	ELEM	5	THRU	25				
ROTORAX	10	PROP	10						
ROTORAX	15	GRID	50	69	70				

Descriptor	Meaning
ROTORID	Identification number of rotor. (Integer > 0). See Remarks 1. and 2..
LTYPE	ELEM, PROP or GRID, indicating whether the specified list references element IDs, property IDs or grid point IDs, respectively. (Character; Required; No default). See Remarks 3. through 6..
IDI	IDs of elements, properties or grid points comprising the rotor. (Integer > 0; Required; No default)
THRU	Implies a range of identification numbers. (Optional).
BY	Implies that an increment is being specified for the THRU option (Optional)
INC	ID increment. (Integer > 0; Optional)

### Remarks:

1. ROTORIDs of ROTORAX entries may not be the same as the ROTORIDs of ROTORG entries or ROTORSE entries.
2. Multiple ROTORAX entries with the same ROTORID are supported.
3. When the GRID option is used for the LTYPE field, a minimum of two grid points must be specified in the list. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
4. For every rotor defined by a ROTORAX entry, there should be at least one entry using the GRID option for the LTYPE field to define the axis of symmetry.

5. Grid points specified on ROTORAX entries using the GRID option for the LTYPE field must be collinear. If this condition is not satisfied, the program terminates the execution with an appropriate user fatal message.
6. Only grid points specified on ROTORAX entries using the GRID option for the LTYPE field may be referenced by RSPINR, RSPINT and UNBALNC entries.

## ROTORG

### Rotor Line Model Grids

Specifies grids that compose the rotor line model.

#### Format:

1	2	3	4	5	6	7	8	9	10
ROTORG	ROTORID	GRID1	GRID2	GRID3	GRID4	GRID5	GRID6	GRID7	
	GRID8	GRID9	-etc.-						

or

ROTORG	ROTORID	GRID1	THRU	GRID2	BY	INC			
--------	---------	-------	------	-------	----	-----	--	--	--

#### Example:

ROTORG	100	101	1002	103	4001				
--------	-----	-----	------	-----	------	--	--	--	--

ROTORG	200	1001	THRU	1100	BY	2			
--------	-----	------	------	------	----	---	--	--	--

Describer	Meaning
ROTORID	Identification number of rotor. (Integer > 0; Required). See Remarks 1. and 2..
GRIDi	Grids comprising the rotor. (Integer > 0; Required; no Default). See Remarks 3. through 7.
THRU	Specifies a range of identification numbers. (Optional)
BY	Specifies an increment for a THRU specification (Optional)
INC	Grid point number increment. (Integer)

#### Remarks:

1. ROTORIDs of ROTORG entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORSE entries.
2. Multiple ROTORG entries with the same ROTORID are supported.
3. Grid IDs must be unique. Duplicate grid IDs will produce a fatal error.
4. All grids specified on ROTORG entries for a specific ROTORID must be collinear. Collinearity will be checked.
5. If not using superelements, no element stiffness can be connected between any GRID listed on a ROTORG entry and any GRID not listed on the ROTORG. In this case, any connections to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.
6. If superelements are used, no element stiffness in the residual structure may connect between a GRID listed on a ROTORG and any GRID not listed on the ROTORG. In this case, any connections in the residual structure to GRIDs listed on a ROTORG must be done using MPC equations or R-elements.



7. All mass for any rotor defined using a ROTORG entry should be defined on the GRIDs listed on the ROTORG entry. Any mass which should belong to the rotor, but is placed on GRIDs not listed on the ROTORG entry, will be ignored when calculating the gyroscopic terms for the rotor. Only the mass shown in the ROTOR DYNAMICS MASS SUMMARY in the .f06 file is included in the calculation of the gyroscopic terms for the rotor.
8. Bar elements (default version) do not have polar moment of inertia. Thus, if CBAR elements are defined along the axis of rotation (for both ROTOR and ROTORG entries), then they do not provide any contribution to gyroscopic matrix. The CABR elements can be replaced by CBEAM elements in case gyroscopic terms are needed for those elements.

## ROTORSE

### Rotor Superelement Identification

Specifies grids that compose the rotor line model. An alternate to the ROTORG entry when superelements are used.

#### Format:

1	2	3	4	5	6	7	8	9	10
ROTORSE	ROTORID	SEID	SEOPT						

#### Example:

ROTORSE	10	1							
---------	----	---	--	--	--	--	--	--	--

Describer	Meaning
ROTORID	Identification number of rotor line model. (Integer > 0; Required). See Remark 1.
SEID	Superelement identification number of rotor superelement. (Integer > 0; Required)
SEOPT	Form of superelement for calculation of gyroscopic terms. (Integer = 1 or 2; Default = 1) See Remark 3.

#### Remarks:

- ROTORIDs of ROTORSE entries may not be the same as the ROTORIDs of ROTORAX entries or ROTORG entries.
- A ROTORSE entry can be used when the rotor is placed in a superelement.
- The rotor line model may be the boundary of a 3D rotor superelement or the rotor line model may be a superelement itself. SEOPT is chosen to distinguish between these cases. The options are:
  - If the user has a 3D model of the rotor and places it in a superelement with SEID, the boundary (a-set) of this superelement must consist of no more and no less than the collinear rotor line model. This will be checked. Specify SEOPT as 1 to identify this configuration. When this option is used, the A-set matrices of the superelement are used to calculate the gyroscopic terms. This option should be used for external superelements.
  - If the user has a line model of the rotor and places it in a superelement with SEID, this superelement (g-set) may be partially or completely reduced in the SE reduction process. This superelement must consist of no more and no less than the rotor line model. Specify SEOPT as 2 to identify this configuration. When this option is used, the G-set matrices of the superelement are used to calculate the gyroscopic terms.
- Rotors specified using the ROTORSE entry can be connected directly to the support structure. In contrast, rotors specified using the ROTORG entry must employ rigid elements to keep the rotor disconnected from the support in the G-set of the residual structure.
- Static and component mode reduction of the rotor line model are supported when using ROTORSE entries.

## RROD Rigid Pin-Ended Element Connection

Defines a pin-ended element that is rigid in translation.

### Format:

1	2	3	4	5	6	7	8	9	10
RROD	EID	GA	GB	CMA	CMB	ALPHA	TREF		

### Example:

RROD	14	1	2	2		6.5-6			
------	----	---	---	---	--	-------	--	--	--

Describer	Meaning
EID	Element identification number. (0 < Integer < 100,000,000)
GA, GB	Grid point identification numbers of connection points. (Integer > 0)
CMA, CMB	Component number of one and only one dependent translational degree-of-freedom in the global coordinate system assigned by the user to either GA or GB. (Integer 1, 2, or 3. Either CMA or CMB must contain the integer, and the other must be blank for the linear RROD. For Lagrange RROD, both CMA and CMB can be blank.) See Remark 3.
ALPHA	Thermal expansion coefficient. See Remark 11. (Real or blank)
TREF	Reference temperature for the calculation of thermal loads. (Real; Default=0.0).

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally one Lagrange multiplier degree-of-freedom in addition to the displacement degrees-of-freedom given by connected grid points.
- For the Lagrange method, if both CMA and CMB are blanks, Nastran will compute the best degree-of-freedom for the dependent degree-of-freedom.
- The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
- Element identification numbers should be unique with respect to all other element identification numbers.
- RROD, among other eligible rigid element types, can be selected via MPC and SET3.
- Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
9. The degree-of-freedom selected to be dependent must have a nonzero component along the axis of the element. This implies that the element must have finite length.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is taken as the average temperature of the two connected grid points GA and GB.

**RSPINR****Relative Spin Rates Between Rotors**

Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis and rotor damping parameters.

**Format:**

1	2	3	4	5	6	7	8	9	10
RSPINR	ROTORID	GRIDA	GRIDB	SPDUNT	SPTID	ROTRSEID			
	GR	ALPHAR1	ALPHAR2	WR3R	WR4R	WRHR			

**Example:**

1	2	3	4	5	6	7	8	9	10
RSPINR	100	1001	1002	RPM	100	5			
	0.02	1	3	1000.	1500.	1200.			

Describer	Meaning
ROTORID	Identification number of rotor. (Integer > 0; Required). See Remark 1.
GRIDA/GRIDB	Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4. (Integer > 0; Required)
SPDUNIT	Specifies whether the listing of relative spin rates is given in terms of RPM (revolutions/minute) or frequency (revolutions (cycles)/sec). (Character; 'RPM' or 'FREQ'; Required)
SPTID	Table for relative rotor spin rates. See Remark 5. (Real or Integer, if integer, must be > 0; Required)
GR	Rotor structural damping factor. See Remarks 8. and 10. (Real; Default = 0.0)
ALPHAR1	Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remarks 9. and 10. (Real; Default = 0.0)
ALPHAR2	Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remarks 9. and 10. (Real; Default = 0.0)
ROTRSEID	Identification number of the superelement in which the rotor specified in the ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.
WR3R	Specifies "average" excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field. (Real, Default = 0.)

Describer	Meaning
WR4R	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)
WRHR	Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.)

**Remarks:**

1. The ROTORID-ROTRSEID pair must be unique across all RSPINR entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINR must be part of the SEs ASET during creation run. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.

If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.

5. If SPTID is a real number, the value is considered constant. For a reference rotor identified using a RGYRO entry, the speed of the rotor is equal to that defined in RGYRO entry itself.
6. For a rotor which is not a reference rotor, the speed is calculated based on the ratio of SPDIT entry of its RSPINR entry to SPDIT field of its reference rotor RSPINR entry, times reference rotor speed from the RGYRO entry.

**Example 1:**

```

$-----2-----3-----4-----5-----6-----7-----8-----9-----
$RSPINR ROTORID GRIDA  GRIDB  SPDUNT  SPDIT  ROTRSEID
$      GR      ALPHAR1 ALPHAR2 WR3R   WR4R   WRHR
RSPINR 40      2        1      RPM    2.0
      0.1
RSPINR 50      4        5      RPM    3.0
      0.1
RSPINR 70     10       12     RPM    1.0
  
```



```

0.1
$-----2-----3-----4-----5-----6-----7-----8-----9-----
$RGYRO  RID      SYNCFLG REFROTR SPRUNIT SPDLOW  SPDHIGH  SPEED  ROTRSEID
$      WR3WRL  WR4WRL  WRHWRL
RGYRO   33      ASYNC   70      RPM                3000.0
0.07                0.07

```

- i. reference rotor id 70, speed = 3000 RPM
- ii. non-reference rotor id 40, speed =  $2.0/1.0 * 3000 = 6000$  RPM
- iii. non-reference rotor id 50, speed =  $3.0/1.0 * 3000 = 9000$  RPM

\*\*\* USER INFORMATION MESSAGE 11152 (ROTDR2)

For SOL107 analysis with RGYRO ID 33 call, having ASYNC option,  
 ROTOR with ID 40 in SEID 0, is having SPIN SPEED = 6.000000E+03 RPM

\*\*\* USER INFORMATION MESSAGE 11152 (ROTDR2)

For SOL107 analysis with RGYRO ID 33 call, having ASYNC option,  
 ROTOR with ID 50 in SEID 0, is having SPIN SPEED = 9.000000E+03 RPM

\*\*\* USER INFORMATION MESSAGE 11152 (ROTDR2)

For SOL107 analysis with RGYRO ID 33 call, having ASYNC option,  
 ROTOR with ID 70 in SEID 0, is having SPIN SPEED = 3.000000E+03 RPM

### Example 2:

```

$-----2-----3-----4-----5-----6-----7-----8-----9-----
-
$RSPINR ROTORID GRIDA  GRIDB  SPDUNT  SPDIT  ROTRSEID
$      GR      ALPHAR1 ALPHAR2 WR3R   WR4R   WRHR
RSPINR  40      2        1      RPM    2.0
0.1
RSPINR  50      4        5      RPM    3.0
0.1
RSPINR  70     10       12     RPM    2.0
0.1
$-----2-----3-----4-----5-----6-----7-----8-----9-----
-
$RGYRO  RID      SYNCFLG REFROTR SPRUNIT SPDLOW  SPDHIGH  SPEED  ROTRSEID
$      WR3WRL  WR4WRL  WRHWRL
RGYRO   33      ASYNC   70      RPM                3000.0
0.07                0.07

```

- i. reference rotor id 70, speed = 3000 RPM

- ii. non-reference rotor id 40, speed = 2.0/2.0 \* 3000 = 3000 RPM
  - iii. non-reference rotor id 50, speed = 3.0/2.0 \* 3000 = 4500 RPM
- ```

*** USER INFORMATION MESSAGE 11152 (ROTDR2)
    For SOL107 analysis with RGYRO ID 33, having ASYNC option,
    ROTOR with ID 40 in SEID 0, has SPIN SPEED = 3.000000E+03 RPM

*** USER INFORMATION MESSAGE 11152 (ROTDR2)
    For SOL107 analysis with RGYRO ID 33, having ASYNC option,
    ROTOR with ID 50 in SEID 0, has SPIN SPEED = 4.500000E+03 RPM

*** USER INFORMATION MESSAGE 11152 (ROTDR2)
    For SOL107 analysis with RGYRO ID 33, having ASYNC option,
    ROTOR with ID 70 in SEID 0, has SPIN SPEED = 3.000000E+03 RPM

```

7. If SPTID is an integer number, the value references a DDVAL entry that specifies the relative rotor spin rates. The number of spin rates for each rotor must be the same. Relative spin rates are determined by correlation of table entries. The  $i^{\text{th}}$  entry for each rotor specifies the relative spin rates between rotors at  $\text{RPM}_i/\text{FREQ}_i$ . Spin rates for the reference rotor must be in ascending or descending order.
8. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping or structural damping depending on the solution. That is,

$$[B_{rotor}]_{structural} = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

where  $WR3$  is a user parameter, or

$$[K_{rotor}] = (1 + iGR)[K_{rotor}]$$

depending on the solution sequence, SYNC/ASYNC and value of PARAM, GYROAVG. See Remark 10. for all the damping and circulation terms added to the equation in the different cases.

In case WR3R is specified in RSPINR, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

9. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$



10. The various rotor damping and circulation matrices applicable to frequency response and complex eigenvalue solutions are shown in the following table. For the fixed reference frame, the circulation terms are generated from the rotating structure. For the rotating reference frame, the circulation terms are generated from the stator.

| Solution                                               |                    |                                                                                                                                                                                                                                                            |
|--------------------------------------------------------|--------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Frequency Response - ASYNC option                      | <b>Damping</b>     | $i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$                                                                                                                                                               |
|                                                        | <b>Circulation</b> | $\Omega_R(\Omega_{ref}) \left( \begin{array}{l} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] + \left(\frac{1}{\omega}\right)[KH_R^C] \end{array} \right)$ |
| Frequency Response - ASYNC option w/ PARAM,GYROA VG,-1 | <b>Damping</b>     | $i\omega \left( \begin{array}{l} [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{array} \right)$                                    |
|                                                        | <b>Circulation</b> | $\Omega_R(\Omega_{ref}) \left( \begin{array}{l} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{array} \right)$       |
| Frequency Response - SYNC option                       | <b>Damping</b>     | $i\omega([B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R]) + i(GR[K_R] + [K4_R] + [KH_R])$                                                                                                                                                               |
|                                                        | <b>Circulation</b> | $\Omega_R(\omega) \left( \begin{array}{l} [B_R^C] + \alpha 1_R[M_R^C] + \alpha 2_R[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{\omega}\right)[K_R^C] + \left(\frac{1}{\omega}\right)[K4_R^C] \\ + \left(\frac{1}{\omega}\right)[KH_R^C] \end{array} \right)$    |

| Solution                                              |                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |
|-------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Frequency Response - SYNC option w/ PARAM,GYROA VG,-1 | <b>Damping</b> $i\omega \left( \begin{aligned} & [B_R] + \alpha_{1R}[M_R] + \alpha_{2R}[K_R] + [BH_R] \\ & + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ & + \left(\frac{1}{WRH}\right)[KH_R] \end{aligned} \right)$                                                                                                                                                                                                                                                                 |
|                                                       | <b>Circulation</b> $\omega\beta_R \left( \begin{aligned} & [B_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] + [BH_R^C] \\ & + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] + \left(\frac{1}{WRH}\right)[KH_R^C] \end{aligned} \right)$ $+ \alpha_R \left( \begin{aligned} & [B_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] + [BH_R^C] \\ & + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] + \left(\frac{1}{WRH}\right)[KH_R^C] \end{aligned} \right)$ |

| Solution                        |                    |                                                                                                                                                                                                                                                                                                                                                                                                                                                                                     |
|---------------------------------|--------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Complex Modes -<br>ASYNC option | <b>Damping</b>     | $i\omega \left( \begin{array}{l} [B_R] + \alpha_{1R}[M_R] + \alpha_{2R}[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{array} \right)$                                                                                                                                                                                                                                                           |
|                                 | <b>Circulation</b> | $\Omega_R(\Omega_{ref}) \left( \begin{array}{l} [B_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{array} \right)$                                                                                                                                                                                                                              |
| Complex Modes -<br>SYNC option  | <b>Damping</b>     | $i\omega \left( \begin{array}{l} [B_R] + \alpha_{1R}[M_R] + \alpha_{2R}[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{array} \right)$                                                                                                                                                                                                                                                           |
|                                 | <b>Circulation</b> | $\omega\beta_R \left( \begin{array}{l} [B_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] + \left(\frac{1}{WRH}\right)[KH_R^C] \end{array} \right) \\ + \alpha_R \left( \begin{array}{l} [B_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] + [BH_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] + \left(\frac{1}{WRH}\right)[KH_R^C] \end{array} \right)$ |

where:

|          |   |                            |
|----------|---|----------------------------|
| $[B_R]$  | = | the rotor viscous damping  |
| $[M_R]$  | = | the rotor mass             |
| $[K_R]$  | = | the rotor stiffness        |
| $[K4_R]$ | = | the rotor material damping |

|                          |   |                                                                                                                                                                                                   |
|--------------------------|---|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| $[BH_R]$                 | = | the rotor viscous hybrid damping                                                                                                                                                                  |
| $[KH_R]$                 | = | the rotor structural hybrid damping                                                                                                                                                               |
| $[B_R^C]$                | = | the circulation due to rotor viscous damping                                                                                                                                                      |
| $M_R^C$                  | = | the circulation due to rotor 'mass'                                                                                                                                                               |
| $K_R^C$                  | = | the circulation due to rotor structural 'stiffness'                                                                                                                                               |
| $[K4_R^C]$               | = | the circulation due to rotor material damping                                                                                                                                                     |
| $[BH_R^C]$               | = | the circulation due to rotor viscous hybrid damping                                                                                                                                               |
| $[KH_R^C]$               | = | the circulation due to rotor structural hybrid damping                                                                                                                                            |
| $\alpha 1_R, \alpha 2_R$ | = | used to specify Rayleigh viscous damping<br>( $[B_R]_{Rayleigh} = \alpha 1_R[M_R] + \alpha 2_R[K_R]$ )                                                                                            |
| $\alpha_R, \beta_R$      | = | scale factors of linear fit of rotor speed to reference rotor speed. The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor. |
| $WR3, WR4, WRH$          | = | User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.                                          |

- Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.
- All rotor damping terms are cumulative. Multiple damping options should be selected with caution.
- If RDBOTH=16, the following damping and stiffness term will be used for Complex Modes (either ASYNC or SYNC option).

$$\text{The damping } i\omega \left( \begin{array}{l} [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \\ + \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R] \\ + \left(\frac{1}{WRH}\right)[KH_R] \end{array} \right) \text{ changes to}$$

$i\omega \{ [B_R] + \alpha 1_R[M_R] + \alpha 2_R[K_R] + [BH_R] \}$  and the  $[K_R]/[K4_R]/[KH_R]$  will be added into the stiffness term with the imaginary formation of  $i \{ GR[K_R] + [K4_R] + [KH_R] \}$ .

## RSPINT Rotor Spin Rates

Specifies rotor spin rates for transient analysis.

### Format:

| 1      | 2       | 3       | 4       | 5       | 6     | 7      | 8       | 9 | 10 |
|--------|---------|---------|---------|---------|-------|--------|---------|---|----|
| RSPINT | ROTORID | GRIDA   | GRIDB   | SPDUNIT | SPTID | SPDOUT | ROTSEID |   |    |
|        | GR      | ALPHAR1 | ALPHAR2 | WR3R    | WR4R  | WRHR   |         |   |    |

### Example:

| 1      | 2    | 3    | 4     | 5     | 6     | 7     | 8 | 9 | 10 |
|--------|------|------|-------|-------|-------|-------|---|---|----|
| RSPINT | 100  | 1001 | 1002  | RPM   | 1001  |       |   |   |    |
|        | 0.01 | 0.01 | 0.002 | 1000. | 1500. | 1200. |   |   |    |

| Descriptor  | Meaning                                                                                                                                                                       |
|-------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ROTORID     | Identification number of rotor. (Integer > 0; Required). See Remark 1.                                                                                                        |
| GRIDA/GRIDB | Positive rotor spin direction is defined from GRIDA to GRIDB. See Remark 4. (Integer > 0; Required)                                                                           |
| SPDUNIT     | Specifies whether the spin rates are given in terms of RPM (revolutions/minute) or frequency (revolutions(cycles)/sec). (Character; 'RPM' or 'FREQ'; Required)                |
| SPTID       | Rotor spin rate. See Remark 5. (Integer > 0; Required)                                                                                                                        |
| SPDOUT      | EPOINT to output the rotor speed vs. time. Output will be in SPDUNITs (Integer > 0 or blank)                                                                                  |
| GR          | Rotor structural damping factor. See Remark 6. and 8. (Real; Default = 0.0)                                                                                                   |
| ALPHAR1     | Scale factor applied to the rotor mass matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)                                                                |
| ALPHAR2     | Scale factor applied to the rotor stiffness matrix for Rayleigh damping. See Remark 7. and 8. (Real; Default = 0.0)                                                           |
| ROTRSEID    | Identification number of the superelement in which the rotor specified in the ROTORID field is defined. (Integer >= 0, Default = 0). See Remark 1.                            |
| WR3R        | Specifies "average" excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through GR field. (Real, Default = 0.) |

| Describer | Meaning                                                                                                                                                                                  |
|-----------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| WR4R      | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural damping specified through material entries. (Real, Default = 0.)    |
| WRHR      | Specifies “average” excitation frequency for calculation of rotor damping and circulation terms for rotor structural hybrid damping specified through ROTHYBD card. (Real, Default = 0.) |

**Remarks:**

1. The ROTORID-ROTRSEID pair must be unique across all RSPINT entries, but the ROTORID and ROTRSEID fields individually need not be unique.
2. Depending upon the type of analysis being performed, a RSPINR/RSPINT entry should be present for each rotor defined by a ROTORG or ROTORAX entry whose gyroscopic effects are to be included in the analysis. If missing for a rotor, then that rotor is assumed to be stationary and the analysis proceeds accordingly. The absence of a RSPINR/RSPINT entry for a rotor is noted in the .f06 file via an appropriate user information message.
3. RSPINR/RSPINT entries are honored and processed only in residual or assembly jobs and are ignored in external superelement creation jobs. For rotor defined in external SE, the GRIDA and GRIDB specified in RSPINT must be part of the SEs ASET during creation. This requirement is relaxed for rotors defined in PART SE.
4. The rotor spin axis is determined from the ROTORG or ROTORAX entries. The positive rotation vector is from GRIDA to GRIDB. GRIDA and GRIDB must be specified on the ROTORG or ROTORAX entry.

If ROTRSEID refers to a secondary external superelement, then GRIDA and GRIDB must be points associated with the ROTORID of the primary external superelement since the secondary external superelement derives all of its properties from the primary external superelement.

5. SPTID references a TABLED1 entry that specifies the rotor spin rate history.
6. Rotor structural damping specified by the GR entry will be added as equivalent viscous damping. The equivalent damping will be calculated using:

$$[B_{rotor}]_{structural} = \left( \frac{GR}{WR3} \right) [K_{rotor}]$$

where WR3 is a user parameter.

In case WR3R is specified in RSPINT, then WR3R is used for determining equivalent viscous damping for this particular rotor instead of WR3.

7. Rayleigh damping for the rotor will be calculated as

$$[B_{rotor}]_{Rayleigh} = \alpha_{R1}(M_{rotor}) + \alpha_{R2}[K_{rotor}]$$

8. The various rotor damping and circulation matrices used in transient analysis are shown in the following table. For the fixed reference frame, the circulation terms are generated from the rotating structure. For the rotating reference frame, the circulation terms are generated from the stator.

| Damping                                                                                                                                                             | Circulation (added to stiffness)                                                                                                                                                                                                            |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| $[B_R] + [BH_R] + \alpha_{1R}[M_R] + \alpha_{2R}[K_R]$ $+ \left(\frac{GR}{WR3}\right)[K_R] + \left(\frac{1}{WR4}\right)[K4_R]$ $+ \left(\frac{1}{WRH}\right)[KH_R]$ | $\Omega_R(t) \left( \begin{array}{l} [B_R^C] + [BH_R^C] + \alpha_{1R}[M_R^C] + \alpha_{2R}[K_R^C] \\ + \left(\frac{GR}{WR3}\right)[K_R^C] + \left(\frac{1}{WR4}\right)[K4_R^C] \\ + \left(\frac{1}{WRH}\right)[KH_R^C] \end{array} \right)$ |

where:

- $[B_R]$  = the rotor viscous damping
- $[M_R]$  = the rotor mass
- $[K_R]$  = the rotor stiffness
- $[K4_R]$  = the rotor material damping
- $[BH_R]$  = the rotor viscous hybrid damping
- $[KH_R]$  = the rotor structural hybrid damping
- $[B_R^C]$  = the circulation due to rotor viscous damping
- $M_R^C$  = the circulation due to rotor 'mass'
- $K_R^C$  = the circulation due to rotor structural 'stiffness'
- $[K4_R^C]$  = the circulation due to rotor material damping
- $[BH_R^C]$  = the circulation due to rotor viscous hybrid damping
- $[KH_R^C]$  = the circulation due to rotor structural hybrid damping
- $\alpha_{1R}, \alpha_{2R}$  = used to specify Rayleigh viscous damping  
 $([B_R]_{Rayleigh} = \alpha_{1R}[M_R] + \alpha_{2R}[K_R])$
- $\alpha_R, \beta_R$  = scale factors of linear fit of rotor speed to reference rotor speed.  
 The linear fit is calculated between the SPDLOW and SPDHIGH speeds (values specified on RGYRO entry) of the reference rotor.
- $WR3, WR4, WRH$  = User parameters specified by PARAM statement. If the parameter values are zero (Default), the corresponding damping and circulation terms are not added.

- a. Parameter and hybrid damping applied to the rotor does not apply to the support and vice versa.
- b. For hybrid damping of the rotors, only the rotor mass and stiffness are used for the modes calculation.
- c. All rotor damping terms are cumulative. Multiple damping options should be selected with caution.



**RSPLINE**

## Interpolation Constraint Element

Defines multipoint constraints for the interpolation of displacements at grid points.

**Format:**

| 1       | 2   | 3   | 4  | 5  | 6      | 7  | 8  | 9  | 10 |
|---------|-----|-----|----|----|--------|----|----|----|----|
| RSPLINE | EID | D/L | G1 | G2 | C2     | G3 | C3 | G4 |    |
|         | C4  | G5  | C5 | G6 | -etc.- |    |    |    |    |

**Example:**

|         |     |     |     |    |        |    |  |    |  |
|---------|-----|-----|-----|----|--------|----|--|----|--|
| RSPLINE | 73  | .05 | 27  | 28 | 123456 | 29 |  | 30 |  |
|         | 123 | 75  | 123 | 71 |        |    |  |    |  |

| Describer | Meaning                                                                                                          |
|-----------|------------------------------------------------------------------------------------------------------------------|
| EID       | Element identification number. (0 < Integer < 100,000,000)                                                       |
| D/L       | Ratio of the diameter of the elastic tube to the sum of the lengths of all segments. (Real > 0.0; Default = 0.1) |
| Gi        | Grid point identification number. (Integer > 0)                                                                  |
| Ci        | Components to be constrained. See Remark 2. (Blank or any combination of the Integers 1 through 6.)              |

**Remarks:**

1. Displacements are interpolated from the equations of an elastic beam passing through the grid points. This is a linear method only element, and not controlled with the Case Control command RIGID.
2. A blank field for Ci indicates that all six degrees-of-freedom at Gi are independent. Since G1 must be independent, no field is provided for C1. Since the last grid point must also be independent, the last field must be a Gi, not a Ci. For the example shown G1, G3, and G6 are independent. G2 has six constrained degrees-of-freedom while G4 and G5 each have three.
3. Dependent (i.e., constrained) degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
4. Degrees-of-freedom declared to be independent by one rigid body element can be made dependent by another rigid body element or by a multipoint constraint.
5. EIDs must be unique.
6. RSPLINE, among other eligible rigid element types, can be selected via MPC and SET3.
7. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.

8. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
9. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
10. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
11. The constraint coefficient matrix is affected by the order of the Gi Ci pairs on the RSPLINE entry. The order of the pairs should be specified in the same order that they appear along the line that joins the two regions. If this order is not followed then the RSPLINE will have folds in it that may yield some unexpected interpolation results.
12. The independent degrees-of-freedom that are the rotation components most nearly parallel to the line joining the regions should not normally be constrained.
13. The RSPLINE has a limit of 100 grid points.

**RSSCON**

## Shell-to-Solid Element Connector

Defines multipoint constraints to model clamped connections of shell-to-solid elements.

**Format:**

| 1      | 2    | 3    | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|------|------|-----|-----|-----|-----|-----|-----|----|
| RSSCON | RBID | TYPE | ES1 | EA1 | EB1 | ES2 | EA2 | EB2 |    |

**Examples:**

|        |     |      |    |    |    |    |    |    |  |
|--------|-----|------|----|----|----|----|----|----|--|
| RSSCON | 110 | GRID | 11 | 12 | 13 | 14 | 15 | 16 |  |
|--------|-----|------|----|----|----|----|----|----|--|

|        |     |      |    |    |    |  |  |  |  |
|--------|-----|------|----|----|----|--|--|--|--|
| RSSCON | 111 | GRID | 31 | 74 | 75 |  |  |  |  |
|--------|-----|------|----|----|----|--|--|--|--|

|        |     |      |     |     |  |  |  |  |  |
|--------|-----|------|-----|-----|--|--|--|--|--|
| RSSCON | 115 | ELEM | 311 | 741 |  |  |  |  |  |
|--------|-----|------|-----|-----|--|--|--|--|--|

| Descriptor | Meaning                                                                                                                                                                                                                         |
|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| RBID       | Element identification number. (0 < Integer < 100,000,000)                                                                                                                                                                      |
| TYPE       | Type of connectivity:<br>ELEM      Connection is described with element identification numbers.<br>GRID      Connection is described with grid point identification numbers.<br>(Character: "GRID" or "ELEM"; Default = "ELEM") |
| ES1        | Shell element identification number if TYPE = "ELEM". Shell grid point identification number if TYPE = "GRID". See <a href="#">Figure 9-141</a> . (Integer > 0)                                                                 |
| EA1        | Solid element identification number if TYPE = "ELEM". Solid grid point identification number if TYPE = "GRID". (Integer > 0)                                                                                                    |
| EB1        | Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)                                                                                                                                           |
| ES2        | Shell grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)                                                                                                                                           |
| EA2        | Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)                                                                                                                                           |
| EB2        | Solid grid-point identification number for TYPE = "GRID" only. (Integer > 0 or blank)                                                                                                                                           |

**Remarks:**

1. RSSCON generates a multipoint constraint that models a clamped connection between a shell and a solid element. The shell degrees-of-freedom are put in the dependent set (m-set). The translational degrees-of-freedom of the shell edge are connected to the translational degrees-of-freedom of the upper and lower solid edge. The two rotational degrees-of-freedom of the shell are connected to the translational degrees-of-freedom of the lower and upper edges of the solid element face. Poisson's ratio effects are considered in the translational degrees-of-freedom.

2. The shell grid point must lie on the line connecting the two solid grid points. It can have an offset from this line, which can not be more than 5% of the distance between the two solid grid points. The shell grid points that are out of the tolerance will not be constrained, and a fatal message will be issued. This tolerance is adjustable. Please see `PARAM,TOLRSC` and `PARAM,SEPIXOVR`.
3. When using the `TYPE = "ELEM"` option
  - The solid elements are `CHEXA`, `CPENTA`, and `CTETRA` with and without midside nodes. The shell elements are `CQUAD4`, `CTRIA3`, `CQUADR`, `CTRIAR`, `CQUAD8`, or `CTRIA6`. If the solid element has mid-side nodes, then the shell element needs mid-side nodes on the common edge.
  - Both the shell and solid elements have to belong to the same superelement. This restriction can be bypassed using `SEELT` entry to reassign the downstream boundary element to an upstream superelement.
  - It is not recommended to connect more than one shell element to the same solid using the `ELEM` option. If attempted, conflicts in the multipoint constraint relations may lead to `UFM 6692`.
4. When using `TYPE = "GRID"` option
  - The `GRID` option does not verify that the grids used are valid shell and/or solid grids.
  - The grids in the `GRID` option can be in different superelements. The shell grid must be in the upstream superelement.
5. It is recommended that the height of the solid element's face is approximately equal to the shell element's thickness of the shell. The shell edge should then be placed in the middle of the solid face.
6. The shell edge may coincide with the upper or lower edge of the solid face.
7. `RSSCON`, among other eligible rigid element types, can be selected via `MPC` and `SET3`.
8. Forces of multipoint constraints may be recovered with the `MPCFORCE` Case Control command.
9. The `RSSCON` is ignored in heat-transfer problems. If used in a multi-physics coupled problem using `SUBSTEP`, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the `DEACTEL` keyword under the `NLMOPTS` Bulk Data entry and the associated Remark 10. for that entry.
10. The m-set coordinates (shell degrees-of-freedom) may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.

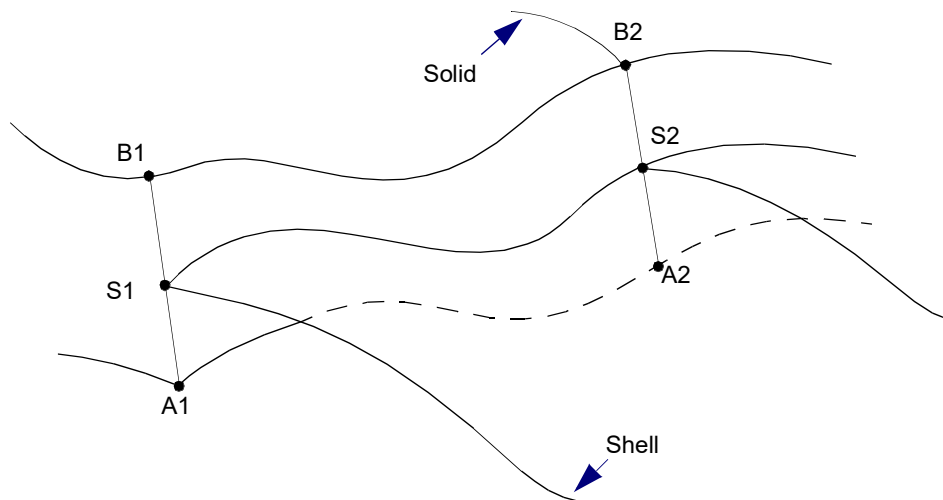


Figure 9-141 Shell Elements Connected to the Faces of Solid Elements

## RTRPLT

## Rigid Triangular Plate

Defines a rigid triangular plate.

### Format:

| 1      | 2   | 3   | 4   | 5     | 6    | 7   | 8   | 9 | 10 |
|--------|-----|-----|-----|-------|------|-----|-----|---|----|
| RTRPLT | EID | GA  | GB  | GC    | CNA  | CNB | CNC |   |    |
|        | CMA | CMB | CMC | ALPHA | TREF |     |     |   |    |

### Example:

|        |   |   |   |   |      |   |   |  |  |
|--------|---|---|---|---|------|---|---|--|--|
| RTRPLT | 7 | 1 | 2 | 3 | 1236 | 3 | 3 |  |  |
|--------|---|---|---|---|------|---|---|--|--|

| Describer     | Meaning                                                                                                                                                                                                                  |
|---------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID           | Element identification number. (0 < Integer < 100,000,000)                                                                                                                                                               |
| GA, GB, GC    | Grid point identification number of connection points.                                                                                                                                                                   |
| CNA, CNB, CNC | Independent degrees-of-freedom in the global coordinate system for the element at grid points GA, GB, and GC, indicated by any of the Integers 1 through 6 with no embedded blanks. See Remark 3. (Integer ≥ 0 or blank) |
| CMA, CMB, CMC | Component numbers of dependent degrees-of-freedom in the global coordinate system. (Any of the Integers 1 through 6 with no embedded blanks, or 0 or blank; Default = 0.)                                                |
| ALPHA         | Thermal expansion coefficient. See Remark 12. (Real or blank)                                                                                                                                                            |
| TREF          | Reference temperature for the calculation of thermal loads. (Real; Default=0.0).                                                                                                                                         |

### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- For the linear method, the total number of components in CNA, CNB, and CNC must equal six; for example, CNA = 1236, CNB = 3, CNC = 3. Furthermore, they must jointly be capable of representing any general rigid body motion of the element. For the Lagrange method, the total number of components must also be six. However, only CNA= 123456 (Default) or CNB = 123456 or CNC = 123456 is allowed. For this type of element, RTRPLT1 gives a simpler input format.

4. For the linear method, the dependent degrees-of-freedom will be made members of the m-set. For the Lagrange method, they may or may not be members of the m-set, depending on the method selected on the RIGID Case Control command. However, the rules regarding the m-set described below apply to both types of methods.
5. Dependent degrees-of-freedom assigned by one rigid element may not also be assigned dependent by another rigid element or by a multipoint constraint.
6. Element identification numbers should be unique with respect to all other element identification numbers.
7. RTRPLT, among other eligible rigid element types, can be selected via MPC and SET3.
8. Forces of multipoint constraint may be recovered in all solution sequences, except SOL 129, with the MPCFORCE Case Control command.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.
10. See [Rigid Elements and Multipoint Constraints \(R-type, MPC\)](#) in the *MSC Nastran Reference Guide* for a discussion of rigid elements.
11. The m-set coordinates specified on this entry may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
12. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The temperature of the element is defined as the following. the bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.

## RTRPLT1

### Rigid Triangular Plate (Alternative Format)

Alternative format to define a rigid triangular plate element connecting three grid points.

#### Format:

| 1       | 2   | 3  | 4  | 5  | 6   | 7   | 8     | 9    | 10 |
|---------|-----|----|----|----|-----|-----|-------|------|----|
| RTRPLT1 | EID | GA | GB | GC | CMB | CMC | ALPHA | TREF |    |

#### Example:

|         |   |   |   |   |      |   |       |  |  |
|---------|---|---|---|---|------|---|-------|--|--|
| RTRPLT1 | 7 | 1 | 2 | 3 | 1236 | 3 | 6.0-6 |  |  |
|---------|---|---|---|---|------|---|-------|--|--|

| Describer  | Meaning                                                                                                                                                                                    |
|------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID        | Element identification number. (0 < Integer < 100,000,000)                                                                                                                                 |
| GA, GB, GC | Grid point identification number of connection points. (Integer > 0)                                                                                                                       |
| CMB, CMC   | Component numbers at GB and GC in the global coordinate systems, which are constrained to move with the rigid body. See Remark 4. (Integers 1 through 6 with no embedded blanks or blank.) |
| ALPHA      | Thermal expansion coefficient. See Remark 9. (Real or blank)                                                                                                                               |
| TREF       | Reference temperature for the calculation of thermal loads. (Real; Default=0.0).                                                                                                           |

#### Remarks:

- Two methods are available to process rigid elements: equation elimination or Lagrange multipliers. The Case Control command, RIGID, selects the method.
- For the Lagrange method, Nastran will create internally the Lagrange multiplier degrees-of-freedom in addition to the 18 displacement degrees-of-freedom given by grid points GA, GB, and GC. The number of Lagrange multiplier degrees-of-freedom is equal to the number of dependent degrees-of-freedom.
- RTRPLT1 is a preferred input format for the Lagrange method.
- When CMB = "123456" or blank, CMC = "123456" or blank, the grid points GB and BC are constrained to move with GA as a rigid triangular plate. For default, CMB = "123456" and CMC = "123456". Any number of degrees-of-freedom at grid points GB and GC can be released not to move with the rigid body.
- The length of any two connected grid points must be greater than zero.
- For the Lagrange method, the theory is formulated such that a consistent rigid body motion for grid points GA, GB, and GC will be computed even if these three points have different global coordinate systems.



7. For the Lagrange method, the thermal expansion effect will be computed, if user supplies the thermal expansion coefficient, ALPHA, and the thermal load is requested by the TEMPERATURE(INITIAL) and TEMPERATURE(LOAD) Case Control commands. The bar GA-GB will have the average temperature of grid points GA and GB. The bar GA-GC will have the average temperature of the grid points GA and GC.
8. Element identification numbers should be unique with respect to all other element identification numbers.
9. Rigid elements are ignored in heat transfer problems. If used in a multi-physics coupled problem using SUBSTEP, they participate in the mechanical substep but are ignored in the heat transfer substep through automatic deactivation. For more information on deactivation, see the DEACTEL keyword under the NLMOPTS Bulk Data entry and the associated Remark 10. for that entry.

## **RVDOF**      Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

**Format:**

|       |     |    |     |    |     |    |     |    |    |
|-------|-----|----|-----|----|-----|----|-----|----|----|
| 1     | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
| RVDOF | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

**Example:**

|       |     |   |     |   |  |  |  |  |  |
|-------|-----|---|-----|---|--|--|--|--|--|
| RVDOF | 800 | 1 | 850 | 2 |  |  |  |  |  |
|-------|-----|---|-----|---|--|--|--|--|--|

| Describer | Meaning                                                                                                              |
|-----------|----------------------------------------------------------------------------------------------------------------------|
| IDI       | Grid or scalar identification number. (Integer > 0)                                                                  |
| CI        | Component numbers. (Any one of the integers 1 through 6 for grid points and integer zero or blank for scalar points) |

**Remarks:**

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.

## RVDOF1 Degrees-of-Freedom Specification for Residual Vector Computations

Specifies the degrees-of-freedom where unit loads are to be applied to obtain static solutions for use in residual vector computations.

### Format:

| 1      | 2   | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|--------|-----|-----|--------|-----|-----|-----|-----|-----|----|
| RVDOF1 | C   | ID1 | ID2    | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|        | ID8 | ID9 | -etc.- |     |     |     |     |     |    |

### Example:

|        |   |     |     |     |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|
| RVDOF1 | 3 | 100 | 210 | 450 |  |  |  |  |  |
|--------|---|-----|-----|-----|--|--|--|--|--|

### Alternate Format and Example:

|        |    |     |        |     |  |  |  |  |  |
|--------|----|-----|--------|-----|--|--|--|--|--|
| RVDOF1 | C  | ID1 | “THRU” | ID2 |  |  |  |  |  |
| RVDOF1 | 12 | 6   | THRU   | 21  |  |  |  |  |  |

| Describer | Meaning                                                                                                                                                                    |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| C         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.) |
| IDi       | Grid or scalar identification numbers. (Integer > 0 or “THRU”; for “THRU” option, ID1 < ID2)                                                                               |

### Remarks:

1. In multiple superelement analysis, the IDi points may be interior to any superelement. The program automatically partitions the data for allocation to the appropriate superelements. Separate entries for separate superelements are not required as in the case of USETi,U6 and SEUSETi,U6 entries.
2. The unit loads applied to the interior points of a superelement due to the RVDOF/RVDOF1 entries are passed downstream all the way down to the residual for the purpose of residual vector processing by all superelements in its downstream path, resulting in more accurate results. This is in contrast to the way residual vector processing is performed when USETi,U6 or SEUSETi,U6 entries are employed. In the latter case, unit loads on a superelement are not passed downstream for residual vector processing by the downstream superelements.
3. If the alternate format is used, points in the sequence ID1 through ID2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
4. When alternative format is used, single point IDs cannot be specified.

## SANGLE Analytical contact threshold angle in SOL 600

Defines automatic analytical contact threshold angle for multiple subcases in SOL 600 only.

### Format:

| 1      | 2   | 3   | 4     | 5   | 6   | 7     | 8 | 9 | 10 |
|--------|-----|-----|-------|-----|-----|-------|---|---|----|
| SANGLE | IDC | IDB | Angle | IDC | IDB | Angle |   |   |    |

### Example:

|        |   |   |      |   |   |      |  |  |  |
|--------|---|---|------|---|---|------|--|--|--|
| SANGLE | 1 | 4 | 50.0 | 1 | 6 |      |  |  |  |
|        | 2 | 4 | -1.0 | 2 | 6 | 55.0 |  |  |  |

| Describer | Meaning                                                                                                                                            |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------|
| IDC       | Identification number of a SUBCASE Case Control command. (Integer; no Default) To enter a value corresponding to Marc's increment zero, set IDC=0. |
| IDB       | Identification of a contact body (must be the same as a BCBODY ID) (Integer; no Default)                                                           |
| Angle     | Threshold automatic analytical contact angle (SANGLE). (Real; Default = 60.0)<br>A value of -1.0 turns off analytical.                             |

### Remarks:

1. This entry should only be made if IDSPL=1 and if SANGLE is a non-zero integer value on one or more BCBODY entry.
2. This entry is available in SOL 600 only.
3. For the example, BCBODY with id=4 has a threshold angle of 50.0 degrees in subcase 1 and analytical contact is turned off in subcase 2. For bcbody=6, the analytical contact is on for subcases 1 and 2 and the threshold angle is 60.0 degrees (the default) and 55.0 degrees for subcases 1 and 2 respectively.
4. Only those contact bodies whose SANGLE changes from subcase to subcase or is turned on/off need be described here. Those with constant SANGLE may be described on the BCBODY entry.

## SEBNDRY Superelement Boundary-Point Definition

Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.

### Format:

| 1       | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| SEBNDRY | SEIDA | SEIDB | GIDA1  | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | -etc.- |       |       |       |       |       |    |

### Example 1:

|         |     |   |    |    |    |    |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|
| SEBNDRY | 400 | 4 | 10 | 20 | 30 | 40 |  |  |  |
|---------|-----|---|----|----|----|----|--|--|--|

### Example 2:

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| SEBNDRY | 400 | ALL | 10 | 20 | 30 | THRU | 40 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Describer | Meaning                                                                                                                    |
|-----------|----------------------------------------------------------------------------------------------------------------------------|
| SEIDA     | Partitioned superelement identification number. (Integer > 0)                                                              |
| SEIDB     | Superelement identification. See Remark 2. (Integer > 0 or Character "ALL"; Default = "ALL")                               |
| GIDAi     | Identification number of a boundary grid point in superelement SEIDA. (Integer > 0 or "THRU"; For "THRU" option, G1 < G2.) |

### Remarks:

1. SEBNDRY may only be specified in the main Bulk Data Section and is not recognized after the BEGIN SUPER=n.
2. SEIDB may reference partitioned superelements or superelements in the main Bulk Data Section.

## SEBSET Fixed Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.

### Format:

| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|--------|------|-----|----|-----|----|-----|----|---|----|
| SEBSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

### Example:

|        |   |   |     |    |   |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|
| SEBSET | 5 | 2 | 135 | 14 | 6 |  |  |  |  |
|--------|---|---|-----|----|---|--|--|--|--|

| Describer | Meaning                                                                                                                                                 |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. (Integer > 0)                                                                                                       |
| Ci        | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points) |
| Idi       | Grid or scalar point identification numbers. (Integer > 0)                                                                                              |

### Remarks:

1. If there are no SECSETi or SEBSETi entries present, all boundary points are, by default, fixed during component mode analysis. If only SEBSETi are entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If both SEBSETi and SECSETi entries are present, the c-set degrees-of-freedom are defined by the SECSETi entries and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSETi entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSETi entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SEBSET1**

## Fixed Boundary Degree-of-Freedom, Alternate Form of SEBSET

Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode calculations.

**Format:**

| 1       | 2    | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|----|--------|----|----|----|----|----|
| SEBSET1 | SEID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|         | G7   | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|         |     |     |     |    |   |    |    |    |  |
|---------|-----|-----|-----|----|---|----|----|----|--|
| SEBSET1 | 5   | 2   | 135 | 14 | 6 | 23 | 24 | 25 |  |
|         | 122 | 127 |     |    |   |    |    |    |  |

**Alternate Format and Example:**

|         |      |   |    |        |    |  |  |  |  |
|---------|------|---|----|--------|----|--|--|--|--|
| SEBSET1 | SEID | C | G1 | "THRU" | G2 |  |  |  |  |
| SEBSET1 | 5    | 3 | 6  | THRU   | 32 |  |  |  |  |

| Describer | Meaning                                                                                                                                       |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. (Integer > 0)                                                                                             |
| C         | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points, 0 or blank for scalar points.) |
| Gi        | Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for THRU option G1 < G2.)                                                |

**Remarks:**

1. If there are no SECSET<sub>i</sub> or SEBSET<sub>i</sub> entries present, all boundary points are, by default, fixed during component mode analysis. If there are only SEBSET<sub>i</sub> entries present, any boundary degrees-of-freedom not listed are placed in the free boundary set (c-set). If there are both SEBSET<sub>i</sub> and SECSET<sub>i</sub> entries present, the c-set degrees-of-freedom are defined by the SECSET<sub>i</sub> entries, and any remaining boundary points are placed in the b-set.
2. Degrees-of-freedom listed on SEBSET<sub>i</sub> entries must be exterior degrees-of-freedom of the superelement and may not be specified on SECSET<sub>i</sub> entries.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive b-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.

4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.



**SEBULK****Partition Superelement Connection**

Defines superelement boundary search options and a repeated, mirrored, or collector superelement.

**Format:**

| 1      | 2    | 3    | 4     | 5      | 6   | 7   | 8      | 9 | 10 |
|--------|------|------|-------|--------|-----|-----|--------|---|----|
| SEBULK | SEID | TYPE | RSEID | METHOD | TOL | LOC | UNITNO |   |    |

**Example:**

|        |    |        |   |      |        |  |  |  |  |
|--------|----|--------|---|------|--------|--|--|--|--|
| SEBULK | 14 | REPEAT | 4 | AUTO | 1.0E-3 |  |  |  |  |
|--------|----|--------|---|------|--------|--|--|--|--|

| Describer | Meaning                                                                                                                                       |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Partitioned superelement identification number. (Integer > 0). See Remark 11.                                                                 |
| TYPE      | Superelement type. (Character; no Default)                                                                                                    |
|           | PRIMARY Primary. See Remarks 2. and 5.                                                                                                        |
|           | COLLECTR Collector. See Remark 3.                                                                                                             |
|           | EXTERNAL External. See Remarks 4. and 5.                                                                                                      |
|           | EXTOP2 External using an OUTPUT2 file created in an earlier run. See Remarks 4. and 5.                                                        |
|           | EXTOP4 External using an OUTPUT4 file created in an earlier run. See Remarks 4. and 5.                                                        |
|           | REPEAT Identical. See Remark 6.                                                                                                               |
|           | MIRROR Mirror. See Remark 6.                                                                                                                  |
| RSEID     | Identification number of the reference superelement, used only if TYPE = "REPEAT" or "MIRROR". (Integer ≥ 0; Default = 0). See Remark 6.      |
| METHOD    | Method to be used when searching for boundary grid points. (Character: "AUTO" or "MANUAL"; Default = "AUTO"). See Remarks 7. and 8.           |
| TOL       | Location tolerance to be used when searching for boundary grid points. (Real; Default = 10E-5). See Remarks 10. and 11.                       |
| LOC       | Coincident location check option for manual connection option. (Character: "YES" or "NO"; Default = "YES"). See Remark 10.                    |
| UNITNO    | FORTTRAN unit number for the OUTPUT2 or OUTPUT4 file (applicable and meaningful only when TYPE = "EXTOP2" or "EXTOP4"). See Remarks 4. and 5. |

**Remarks:**

1. This specification must be specified in the MAIN Bulk Data and is meaningful only if part superelements (using BEGIN SUPER) or external superelements created by employing the EXTSEOUT Case Control command exist in the data.

2. TYPE = PRIMARY indicates that the superelement is a part superelement (using BEGIN SUPER).
3. TYPE = "COLLCTR" indicates a collector superelement which does not contain any grids or scalar points.
4. TYPE = "EXTERNAL", "EXTOP2" or "EXTOP4" indicates an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. See discussion under the description of the EXTSEOUT Case Control command. (For employing external superelements using the old three-step procedure, see discussion under the description of parameter EXTDROUT in Chapter 5.)
5. A superelement whose TYPE is "PRIMARY", "EXTERNAL" or "EXTOP2" may itself be re-positioned by the use of an SELOC or an SEMPLN entry or both.

If it references an SELOC entry, then the primary superelement will be positioned at the location implied by the SELOC entry.

If it references an SEMPLN entry, then a mirror image of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

6. TYPE = "REPEAT" and TYPE = "MIRROR" have many common features, but they also have some important differences. These are described below.

The following comments apply to both TYPE= "REPEAT" and TYPE = "MIRROR":

- a. An SEID whose TYPE is "REPEAT" or "MIRROR" is referred to as a secondary superelement. The RSEID specified in this case is regarded as the primary superelement.
- b. The primary superelement must be either a part superelement or an external superelement created by employing the EXTSEOUT Case Control command in an earlier job. It must be defined via its own SEBULK entry wherein TYPE must be either "PRIMARY", "EXTERNAL" or "EXTOP2".
- c. The primary superelement does not include superelements upstream of the primary superelement.
- d. The primary superelement may be re-positioned as indicated in Remark 5.
- e. If the primary superelement specified by RSEID is a part superelement, then the secondary superelement is a "G-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement can be different from those of its primary superelement.
- f. If the primary superelement specified by RSEID is an external superelement resulting from the use of the EXTSEOUT Case Control command in an earlier job, then the secondary superelement is an "A-set" copy of the primary superelement. In this case, the boundary, loads, constraints and reduction procedure of the secondary superelement are set and are the same as those of its primary superelement.

The following comments apply only to TYPE= "REPEAT":

- g. The secondary superelement in this case may reference an SELOC entry, an SEMPLN entry or both or none.

If it references an SELOC entry, then an identical copy of its primary superelement will be positioned at the location implied by the SELOC entry.

If it references an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.

If it references neither an SELOC entry nor an SEMPLN entry, then the secondary superelement will merely be a duplicate of the primary superelement positioned at the same location as the primary superelement. This usage is extremely uncommon. Hence the program cautions the user about this usage by issuing a user warning message.

The following comments apply only to TYPE= "MIRROR":

- h. The secondary superelement in this case must reference an SEMPLN entry. (Otherwise, the program will terminate the execution with an appropriate user fatal message.) In addition, the secondary superelement may also reference an SELOC entry.
 

If the secondary superelement references only an SEMPLN entry, then a mirror image copy of the primary superelement will be positioned using the plane defined by the SEMPLN entry.

If it references both an SELOC entry and an SEMPLN entry, then a mirror image copy of the primary superelement will first be created using the plane defined by the SEMPLN entry and then it will be positioned at the location implied by the SELOC entry.
7. METHOD = "MANUAL" requires SECONCT entries. SEBNDRY and SEEXCLD, which reference SEID, will produce a fatal message.
8. SECONCT, SEBNDRY, and SEEXCLD entries can be used to augment the search procedure and/or override the global tolerance.
9. For combined automatic and manual boundary search, the METHOD = "AUTO" should be specified and connections should be specified on a SECONCT entry.
10. TOL and LOC are the default values that can be modified between two superelements by providing the required tolerance on the SECONCT entry.
11. A SEID=0 is valid for the residual part only if it is desired to set a TOL value for the residual.

## SECONCT Partitioned Superelement Boundary-Point Connection

Explicitly defines grid and scalar point connection procedures for a partitioned superelement.

### Format:

| 1       | 2     | 3     | 4     | 5     | 6     | 7     | 8      | 9 | 10 |
|---------|-------|-------|-------|-------|-------|-------|--------|---|----|
| SECONCT | SEIDA | SEIDB | TOL   | LOC   |       |       |        |   |    |
|         | GIDA1 | GIDB1 | GIDA2 | GIDB2 | GIDA3 | GIDB3 | -etc.- |   |    |

### Example:

|         |      |      |        |     |      |      |  |  |  |
|---------|------|------|--------|-----|------|------|--|--|--|
| SECONCT | 10   | 20   | 1.0E-4 | YES |      |      |  |  |  |
|         | 1001 | 4001 |        |     | 2222 | 4444 |  |  |  |

### Alternate Format and Example:

|         |       |        |       |       |        |       |  |  |  |
|---------|-------|--------|-------|-------|--------|-------|--|--|--|
| SECONCT | SEIDA | SEIDB  | TOL   | LOC   |        |       |  |  |  |
|         | GIDA1 | 'THRU' | GIDA2 | GIDB1 | 'THRU' | GIDB2 |  |  |  |

|         |     |        |     |     |        |     |  |  |  |
|---------|-----|--------|-----|-----|--------|-----|--|--|--|
| SECONCT | 10  | 20     |     |     |        |     |  |  |  |
|         | 101 | 'THRU' | 110 | 201 | 'THRU' | 210 |  |  |  |

| Descriptor | Meaning                                                                                                    |
|------------|------------------------------------------------------------------------------------------------------------|
| SEIDA      | Identification number of superelement for connection to SEIDB. (Integer $\geq 0$ )                         |
| SEIDB      | Identification number of superelement for connection to SEIDA. (Integer $\geq 0$ )                         |
| TOL        | Location tolerance to be used when searching for or checking boundary grid points. (Real; Default = 10E-5) |
| LOC        | Coincident location check option for manual connection. (Character; "YES" or "NO"; Default = "YES")        |
| GIDAi      | Identification number of a grid or scalar point in superelement SEIDA, which will be connected to GIDBi.   |
| GIDBi      | Identification number of a grid or scalar point in superelement SEIDB, which will be connected to GIDAi.   |

### Remarks:

1. SECONCT can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER = n command.
2. TOL and LOC can be used to override the default values specified on the SEBULK entries.
3. The continuation entry is optional.
4. The (GIAi, GIBi) pair must both be grids or scalar points.

5. All six degrees-of-freedom of grid points will be defined as boundary degrees-of-freedom.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.
7. Blank fields are allowed after the first GIDA1-GIDB1 pair. Blank fields must also occur in pairs. This remark does not apply to the alternate format.
8. For Alternate Format 1, the thru ranges must be closed sets. That is, all IDs listed between 101 and 110 in the example must exist in the model.

**SECSET**

Free Boundary Degree-of-Freedom

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

|        |      |     |    |     |    |     |    |   |    |
|--------|------|-----|----|-----|----|-----|----|---|----|
| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
| SECSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

**Example:**

|        |   |     |   |   |    |   |    |  |  |
|--------|---|-----|---|---|----|---|----|--|--|
| SECSET | 3 | 124 | 1 | 5 | 23 | 6 | 15 |  |  |
|--------|---|-----|---|---|----|---|----|--|--|

| Describer | Meaning                                                                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. (Integer > 0)                                                                                                      |
| Ci        | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| IDI       | Grid or scalar point identification number. (Integer > 0)                                                                                              |

**Remarks:**

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Coordinates listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined by the program and listed in the SEMAP table output.
2. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
3. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
4. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.

**SECSET1**

## Free Boundary Degree-of-Freedom, Alternate Form of SECSET

Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.

**Format:**

| 1       | 2    | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|---------|------|----|----|--------|----|----|----|----|----|
| SECSET1 | SEID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|         | G7   | G8 | G9 | -etc.- |    |    |    |    |    |

**Example:**

|         |     |     |     |    |   |    |    |    |  |
|---------|-----|-----|-----|----|---|----|----|----|--|
| SECSET1 | 5   | 2   | 135 | 14 | 6 | 23 | 24 | 25 |  |
|         | 122 | 127 |     |    |   |    |    |    |  |

**Alternate Formats and Example:**

|         |      |   |    |        |    |  |  |  |  |
|---------|------|---|----|--------|----|--|--|--|--|
| SECSET1 | SEID | C | G1 | "THRU" | G2 |  |  |  |  |
| SECSET1 | 5    | 3 | 6  | THRU   | 32 |  |  |  |  |

|         |      |  |       |  |  |  |  |  |  |
|---------|------|--|-------|--|--|--|--|--|--|
| SECSET1 | SEID |  | "ALL" |  |  |  |  |  |  |
| SECSET1 | SEID |  | ALL   |  |  |  |  |  |  |

**Describer****Meaning**

|      |                                                                                                                                                                             |
|------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID | Superelement identification number. (Integer > 0)                                                                                                                           |
| C    | Component numbers of degree-of-freedom. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi   | Grid or scalar point identification number. (Integer > 0)                                                                                                                   |

**Remarks:**

1. Exterior grid and scalar points are, by default, fixed during component mode analysis and placed in the b-set unless listed on SECSETi or SESUP entries. Degrees-of-freedom listed on this entry are considered free (c-set) during component mode calculations. Exterior grid and scalar points are determined automatically and listed in the SEMAP table output.
2. If the alternate formats are used, the grid points Gi are not required to exist or to be exterior degrees-of-freedom and may be listed on SECSET1 entries. Points of this type will cause one warning message but will otherwise be ignored.
3. Degrees-of-freedom specified on this entry are assigned to the mutually exclusive c-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.

4. There must be a sufficient number of degrees-of-freedom specified on SESUP entries to discard any free body modes of the superelement.
5. If PARAM,AUTOSPC is YES, then singular b-set and c-set degrees-of-freedom will be reassigned as follows:
  - If there are no o-set (omitted) degrees-of-freedom, then singular b-set and c-set degrees-of-freedom are reassigned to the s-set.
  - If there are o-set (omitted) degrees-of-freedom, then singular c-set degrees-of-freedom are reassigned to the b-set. Singular b-set degrees-of-freedom are not reassigned.



## SECTAX Conical Shell Sector

Defines a sector of a conical shell.

### Format:

| 1      | 2  | 3   | 4 | 5    | 6    | 7 | 8 | 9 | 10 |
|--------|----|-----|---|------|------|---|---|---|----|
| SECTAX | ID | RID | R | PHI1 | PHI2 |   |   |   |    |

### Example:

|        |   |   |     |      |      |  |  |  |  |
|--------|---|---|-----|------|------|--|--|--|--|
| SECTAX | 1 | 2 | 3.0 | 30.0 | 40.0 |  |  |  |  |
|--------|---|---|-----|------|------|--|--|--|--|

| Describer  | Meaning                                                     |
|------------|-------------------------------------------------------------|
| ID         | Sector identification number. (Unique Integer > 0)          |
| RID        | Ring identification number. See RINGAX entry. (Integer > 0) |
| R          | Effective radius. (Real)                                    |
| PHI1, PHI2 | Azimuthal limits of sector in degrees. (Real)               |

### Remarks:

1. SECTAX is allowed only if an AXIC entry is also present.
2. SECTAX identification numbers must be unique with respect to all other POINTAX, RINGAX and SECTAX identification numbers.
3. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

## SEDLINK

## Multiple Design Variable Linking Across PART SE Boundary

Relates one design variable of a PART SE to one or more other design variables from other PART SEs.

### Format:

| 1       | 2      | 3     | 4     | 5      | 6     | 7      | 8    | 9  | 10 |
|---------|--------|-------|-------|--------|-------|--------|------|----|----|
| SEDLINK | ID     | DSEID | DDVID | C0     | CMULT | ISEID1 | IDV1 | C1 |    |
|         | ISEID2 | IDV2  | C2    | ISEID3 | IDV3  | C3     |      |    |    |
|         | ISEID4 | IDV4  | C4    | -etc.- |       |        |      |    |    |

### Example:

|         |    |    |      |     |      |     |   |      |  |
|---------|----|----|------|-----|------|-----|---|------|--|
| SEDLINK | 10 | 10 | 2    | 0.1 | 0.33 | 8   | 8 | -1.0 |  |
|         | 11 | 6  | -1.0 | 20  | 8    | 7.0 |   |      |  |
|         | 11 | 2  | 2.0  |     |      |     |   |      |  |

| Descriptor         | Meaning                                                          |
|--------------------|------------------------------------------------------------------|
| ID                 | Unique identification number. (Integer > 0)                      |
| DSEID              | PART SE identification number for DDVID (Integer ≥ 0)            |
| DDVID              | Dependent design variable identification number. (Integer > 0)   |
| C0                 | Constant term. (Real; Default = 0.0)                             |
| CMULT              | Constant multiplier. (Real; Default = 1.0)                       |
| ISEID <sub>i</sub> | PART SE identification number for IDV <sub>i</sub> (Integer ≥ 0) |
| IDV <sub>i</sub>   | Independent design variable identification number. (Integer > 0) |
| C <sub>i</sub>     | Coefficient <i>i</i> corresponding to IDV <sub>i</sub> . (Real)  |

### Remarks:

1. SEDLINK is provided for inter-PART SE design variables linking and must be placed in main Bulk Data Section for consideration. SEDLINK will be ignored if it shows up under 'BEGIN SUPER=seid' where seid>0.
2. SEDLINK defines the relationship

$$DDVID = C0 + CMULT \sum_i C_{i_{ISEIDi}} \cdot IDV_{i_{ISEIDi}}$$

3. This capability provides a means of linking physical design variables such as element thicknesses to nonphysical design variables such as the coefficients of interpolating functions.

4. *CMULT* provides a simple means of scaling the  $C_i$ . For example if  $C_i = 1/7, 2/7, 4/7$ , etc. is desired, then  $CMULT = 1/7$  and  $C_i = 1, 2, 4$ , etc., may be input.
5. An independent  $IDV_i_{ISEID_i}$  must not occur on the same SEDLINK entry more than once.
6. ID is for user reference only.
7. If a design variable of a PART SE is specified as dependent on a SEDLINK entry, then it cannot be specified as independent on another SEDLINK or DLINK entry.

## SEDRSP2

## Design Sensitivity Equation Response Quantities for PART SE

Defines equation responses that are used in the design, either as constraints or as an objective with quantities from multiple PART SEs.

### Format:

| 1       | 2         | 3       | 4            | 5       | 6       | 7       | 8      | 9  | 10 |
|---------|-----------|---------|--------------|---------|---------|---------|--------|----|----|
| SEDRSP2 | ID        | LABEL   | EQID or FUNC | REGION  | METHOD  | C1      | C2     | C3 |    |
|         | "DESVAR"  | DVSEID1 | DVID1        | DVSEID2 | DVID2   | DVSEID3 | DVID3  |    |    |
|         |           | DVSEID4 | DVID4        | -etc.-  |         |         |        |    |    |
|         | "DTABLE"  | LBSEID1 | LABL1        | LBSEID2 | LABL2   | LBSEID3 | LABL3  |    |    |
|         |           | LBSEID4 | LABL4        | -etc.-  |         |         |        |    |    |
|         | "DRESP1"  | R1SEID1 | NR1          | R1SEID2 | NR2     | R1SEID3 | NR3    |    |    |
|         |           | R1SEID4 | NR4          | -etc.-  |         |         |        |    |    |
|         | "DNODE"   | NDSEID1 | G1           | CMP1    | NDSEID2 | G2      | CMP2   |    |    |
|         |           | NDSEID3 | G3           | CMP3    | -etc.-  |         |        |    |    |
|         | "DVPREL1" | P1SEID1 | DPIP1        | P1SEID2 | DPIP2   | P1SEID3 | DPIP3  |    |    |
|         |           | P1SEID4 | DPIP4        | -etc.-  |         |         |        |    |    |
|         | "DVCREL1" | C1SEID1 | DCIC1        | C1SEID2 | DCIC2   | C1SEID3 | DCIC3  |    |    |
|         |           | C1SEID4 | DCIC4        | -etc.-  |         |         |        |    |    |
|         | "DVMREL1" | M1SEID1 | DMIM1        | M1SEID2 | DMIM2   | M1SEID3 | DMIM3  |    |    |
|         |           | M1SEID4 | DMIM4        | -etc.-  |         |         |        |    |    |
|         | "DVPREL2" | P2SEID1 | DPI2P1       | P2SEID2 | DPI2P2  | P2SEID3 | DPI2P3 |    |    |
|         |           | P2SEID4 | DPI2P4       | -etc.-  |         |         |        |    |    |
|         | "DVCREL2" | C2SEID1 | DC12C1       | C2SEID2 | DC12C2  | C2SEID3 | DC12C3 |    |    |
|         |           | C2SEID4 | DC12C4       | -etc.-  |         |         |        |    |    |
|         | "DVMREL2" | M2SEID1 | DM12M1       | M2SEID2 | DM12M2  | M2SEID3 | DM12M3 |    |    |
|         |           | M2SEID4 | DM12M4       | -etc.-  |         |         |        |    |    |

### Example:

|         |         |       |     |    |     |   |   |  |  |
|---------|---------|-------|-----|----|-----|---|---|--|--|
| SEDRSP2 | 1       | LBUCK | 5   | 3  |     |   |   |  |  |
|         | DESVAR  | 0     | 101 | 1  | 3   | 1 | 4 |  |  |
|         |         | 10    | 201 |    |     |   |   |  |  |
|         | DTABLE  | 1     | YM  | 10 | L   |   |   |  |  |
|         | DRESP1  | 0     | 14  | 1  | 1   | 1 | 4 |  |  |
|         | DNODE   | 10    | 14  | 0  | 1   |   |   |  |  |
|         | DVPREL1 | 0     | 101 | 1  | 102 |   |   |  |  |

|  |         |    |     |    |     |    |     |  |
|--|---------|----|-----|----|-----|----|-----|--|
|  | DVCREL1 | 1  | 201 | 10 | 202 |    |     |  |
|  | DVMREL1 | 0  | 301 |    |     |    |     |  |
|  | DVPREL2 | 10 | 401 | 1  | 402 |    |     |  |
|  | DVCREL2 | 0  | 501 |    |     |    |     |  |
|  | DVMREL2 | 0  | 601 | 1  | 602 | 10 | 603 |  |

| Descriptor | Meaning                                                                                                                                                                                                                                                                                  |
|------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID         | Unique identification number. (Integer > 0)                                                                                                                                                                                                                                              |
| LABEL      | User-defined label. (Character)                                                                                                                                                                                                                                                          |
| EQID       | DEQATN entry identification number. (Integer > 0)                                                                                                                                                                                                                                        |
| FUNC       | Function to be applied to the arguments. See Remark 8. of the DRESP2 entry. (Character)                                                                                                                                                                                                  |
| REGION     | Region identifier for constraint screening. See Remark 5. of the DRESP2 entry. (Integer > 0)                                                                                                                                                                                             |
| METHOD     | When used with FUNC = BETA, METHOD = MIN indicates a minimization task while MAX indicates a maximization task. (Default = MIN) When used with FUNCT = MATCH, METHOD = LS indicated a least squares while METHOD = BETA indicated minimization of the maximum difference. (Default = LS) |
| Ci         | Constants used when FUNC = BETA or FUNC = MATCH in combination with METHOD = BETA. See Remark 8. of the DRESP2 entry. (Real; Defaults: C1 = 100., C2 = .005)                                                                                                                             |
| “DESVAR”   | Flag indicating DESVAR entry identification numbers. (Character)                                                                                                                                                                                                                         |
| DVSEIDi    | PART SE identification number for DVIDi (Integer ≥ 0)                                                                                                                                                                                                                                    |
| DVIDi      | DESVAR entry identification number. (Integer > 0)                                                                                                                                                                                                                                        |
| “DTABLE”   | DTABLE flag. Indicates that the LABLs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character)                                                                                                  |
| LBSEIDj    | PART SE identification number for LABLj. (Integer ≥ 0)                                                                                                                                                                                                                                   |
| LABLi      | Label for a constant on the DTABLE or DTABLE2 entry. (Character)                                                                                                                                                                                                                         |
| “DRESP1”   | Flag indicating DRESP1 entry identification numbers. (Character)                                                                                                                                                                                                                         |
| R1SEIDk    | PART SE identification number for NRk. (Integer ≥ 0)                                                                                                                                                                                                                                     |
| NRk        | DRESP1 entry identification number. (Integer > 0)                                                                                                                                                                                                                                        |
| “DNODE”    | Flag indicating grid point and component identification numbers. (Character)                                                                                                                                                                                                             |
| NDSEIDm    | PART SE identification number for (Gm,Cm). (Integer ≥ 0)                                                                                                                                                                                                                                 |
| Gm         | Identification number for any grid point in the model. (Integer > 0)                                                                                                                                                                                                                     |
| Cm         | Component number of grid point Gm. (1 < Integer < 3)                                                                                                                                                                                                                                     |
| “DVPREL1”  | Flag indicating DVPREL1 entry identification number. (Character)                                                                                                                                                                                                                         |

| Describer | Meaning                                                          |
|-----------|------------------------------------------------------------------|
| P1SEIDi   | PART SE identification number for DPIPi. (Integer $\geq 0$ )     |
| DPIPi     | DVPREL1 entry identification number. (Integer $> 0$ )            |
| “DVCREL1” | Flag indicating DVCREL1 entry identification number. (Character) |
| C1SEIDi   | PART SE identification number for DCiCi. (Integer $\geq 0$ )     |
| DCiCi     | DVCREL1 entry identification number. (Integer $> 0$ )            |
| “DVMREL1” | Flag indicating DVPREL2 entry identification number. (Character) |
| M1SEIDi   | PART SE identification number for DMiMi. (Integer $\geq 0$ )     |
| DMiMi     | DVMREL1 entry identification number. (Integer $> 0$ )            |
| “DVPREL2” | Flag indicating DVPREL2 entry identification number. (Character) |
| P2SEIDi   | PART SE identification number for DPI2Pi. (Integer $\geq 0$ )    |
| DPI2Pi    | DVPREL2 entry identification number. (Integer $> 0$ )            |
| “DVCREL2” | Flag indicating DVCREL2 entry identification number. (Character) |
| C2SEIDi   | PART SE identification number for DCi2Ci. (Integer $\geq 0$ )    |
| DCi2Ci    | DVCREL2 entry identification number. (Integer $> 0$ )            |
| “DVMREL2” | Flag indicating DVMREL2 entry identification number. (Character) |
| M2SEIDi   | PART SE identification number for DMi2Mi. (Integer $\geq 0$ )    |
| DMi2Mi    | DVMREL2 entry identification number. (Integer $> 0$ )            |

**Remarks:**

1. SEDRSP2 is provided specifically for creating synthetic response with quantities from multiple PART SEs and must be placed in main bulk data section for consideration. SEDRSP2 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where  $seid > 0$ .
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR. DVxRELy or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP2 follows rules and shares limitations of DRESP2. Rules and limitations of DRESP2 will not be repeated here.
4. SEDRSP2 does not support DRESP2 flag.

**SEDRSP3****Defines External Response with User-Supplied Routines**

Defines constituents from multiple PART SE for an external response using user-supplied routine(s).

**Format:**

| 1       | 2         | 3       | 4      | 5       | 6       | 7       | 8      | 9 | 10 |
|---------|-----------|---------|--------|---------|---------|---------|--------|---|----|
| SEDRSP3 | ID        | LABEL   | GROUP  | TYPE    | REGION  |         |        |   |    |
|         | “DESVAR”  | DVSEID1 | DVID1  | DVSEID2 | DVID2   | DVSEID3 | DVID3  |   |    |
|         |           | DVSEID4 | DVID4  | -etc.-  |         |         |        |   |    |
|         | “DTABLE”  | LBSEID1 | LABL1  | LBSEID2 | LABL2   | LBSEID3 | LABL3  |   |    |
|         |           | LBSEID4 | LABL4  | -etc.-  |         |         |        |   |    |
|         | “DRESP1   | R1SEID1 | NR1    | R1SEID2 | NR2     | R1SEID3 | NR3    |   |    |
|         |           | R1SEID4 | NR4    | -etc.-  |         |         |        |   |    |
|         | “DNODE”   | NDSEID1 | G1     | CMP1    | NDSEID2 | G2      | CMP2   |   |    |
|         |           | NDSEID3 | G3     | CMP3    | -etc.-  |         |        |   |    |
|         | “DVPREL1” | P1SEID1 | DPIP1  | P1SEID2 | DPIP2   | P1SEID3 | DPIP3  |   |    |
|         |           | P1SEID4 | DPIP4  | -etc.-  |         |         |        |   |    |
|         | “DVCREL1” | C1SEID1 | DCIC1  | C1SEID2 | DCIC2   | C1SEID3 | DCIC3  |   |    |
|         |           | C1SEID4 | DCIC4  | -etc.-  |         |         |        |   |    |
|         | “DVMREL1: | M1SEID1 | DMIM1  | M1SEID2 | DMIM2   | M1SEID3 | DMIM3  |   |    |
|         |           | M1SEID4 | DMIM4  | -etc.-  |         |         |        |   |    |
|         | “DVPREL1” | P2SEID  | DPI2P1 | P2SEID2 | DPI2P2  | P2SEID3 | DPI2P3 |   |    |
|         |           | P2SEID4 | DPI2P4 | -etc.-  |         |         |        |   |    |
|         | “DVCREL2” | C2SEID1 | DCI2C1 | C2SEID2 | DCI2C2  | C2SEID3 | DCI2C3 |   |    |
|         |           | C2SEID4 | DCI2C4 | -etc.-  |         |         |        |   |    |
|         | “DVMREL2  | M2SEID  | DMI2M1 | M2SEID2 | DMI2M2  | M2SEID3 | DMI2M3 |   |    |
|         |           | M2SEID4 | DMI2M4 | -etc.-  |         |         |        |   |    |
|         | “USRDATA” |         |        |         |         | String  |        |   |    |
|         |           |         |        |         |         | -etc.-  |        |   |    |

**Example:**

|         |         |       |          |      |     |   |   |  |  |
|---------|---------|-------|----------|------|-----|---|---|--|--|
| SEDRSP3 | 10      | LBUCK | TAILWING | BUCK |     |   |   |  |  |
|         | DESVAR  | 0     | 101      | 1    | 3   | 1 | 4 |  |  |
|         |         | 10    | 201      |      |     |   |   |  |  |
|         | DTABLE  | 1     | YM       | 10   | L   |   |   |  |  |
|         | DRESP1  | 0     | 14       | 1    | 1   | 1 | 4 |  |  |
|         | DNODE   | 10    | 14       | 0    | 1   |   |   |  |  |
|         | DVPREL1 | 0     | 101      | 1    | 102 |   |   |  |  |

|  |         |                             |     |    |     |    |     |  |  |
|--|---------|-----------------------------|-----|----|-----|----|-----|--|--|
|  | DVCREL1 | 1                           | 201 | 10 | 202 |    |     |  |  |
|  | DVMREL1 | 0                           | 301 |    |     |    |     |  |  |
|  | DVPREL2 | 10                          | 401 | 1  | 402 |    |     |  |  |
|  | DVCREL2 | 0                           | 501 |    |     |    |     |  |  |
|  | DVMREL2 | 0                           | 601 | 1  | 602 | 10 | 603 |  |  |
|  | USRDATA | Constants: 12345.6789.0 99. |     |    |     |    |     |  |  |

| Descriptor | Meaning                                                                                                                                                                                  |
|------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID         | Unique identification number. (Integer > 0)                                                                                                                                              |
| LABEL      | User-defined label. (Character)                                                                                                                                                          |
| GROUP      | Group name the external response type belongs to (Character). See Remark 2. of DRESP3.                                                                                                   |
| TYPE       | External response type (Character). See Remark 3. of the DRESP3.                                                                                                                         |
| “DESVAR”   | Flag indicating DESVAR entry identification numbers. (Character)                                                                                                                         |
| DVSEIDi    | PART SE identification number for DVIDi. (Integer ≥ 0)                                                                                                                                   |
| DVIDi      | DESVAR entry identification number. (Integer > 0)                                                                                                                                        |
| “DTABLE”   | DTABLE flag. Indicates that the LABELs for the constants in a DTABLE or DTABLE2 entry follow. This field may be omitted if there are no constants involved in this relation. (Character) |
| LBSEIDj    | PART SE identification number for LABELj. (Integer ≥ 0)                                                                                                                                  |
| LABELi     | Label for a constant on the DTABLE or DTABLE2 entry. (Character)                                                                                                                         |
| “DRESP1”   | Flag indicating DRESP1 entry identification numbers. (Character)                                                                                                                         |
| R1SEIDk    | PART SE identification number for NRk. (Integer ≥ 0)                                                                                                                                     |
| NRk        | DRESP1 entry identification number. (Integer > 0)                                                                                                                                        |
| “DNODE”    | Flag indicating grid point and component identification numbers. (Character)                                                                                                             |
| NDSEIDm    | PART SE identification number for (Gm,Cm). (Integer ≥ 0)                                                                                                                                 |
| Gm         | Identification number for any grid point in the model. (Integer > 0)                                                                                                                     |
| Cm         | Component number of grid point Gm. (1 < Integer < 3)                                                                                                                                     |
| “DVPREL1”  | Flag indicating DVPREL1 entry identification number. (Character)                                                                                                                         |
| P1SEIDi    | PART SE identification number for DPIPi. (Integer ≥ 0)                                                                                                                                   |
| DPIPi      | DVPREL1 entry identification number. (Integer > 0)                                                                                                                                       |
| “DVCREL1”  | Flag indicating DVCREL1 entry identification number. (Character)                                                                                                                         |
| C1SEIDi    | PART SE identification number for DCiCi. (Integer ≥ 0)                                                                                                                                   |
| DCiCi      | DVCREL1 entry identification number. (Integer > 0)                                                                                                                                       |
| “DVMREL1”  | Flag indicating DVPREL2 entry identification number. (Character)                                                                                                                         |
| M1SEIDi    | PART SE identification number for DMiMi. (Integer ≥ 0)                                                                                                                                   |





| Describer | Meaning                                                          |
|-----------|------------------------------------------------------------------|
| DMIMi     | DVMREL1 entry identification number. (Integer > 0)               |
| “DVPREL2” | Flag indicating DVPREL2 entry identification number. (Character) |
| P2SEIDi   | PART SE identification number for DPI2Pi. (Integer $\geq$ 0)     |
| DPI2Pi    | DVPREL2 entry identification number. (Integer > 0)               |
| “DVCREL2” | Flag indicating DVCREL2 entry identification number. (Character) |
| C2SEIDi   | PART SE identification number for DCI2Ci. (Integer $\geq$ 0)     |
| DCI2Ci    | DVCREL2 entry identification number. (Integer > 0)               |
| “DVMREL2” | Flag indicating DVMREL2 entry identification number. (Character) |
| M2SEIDi   | PART SE identification number for DMI2Mi. (Integer $\geq$ 0)     |
| DMI2Mi    | DVMREL2 entry identification number. (Integer > 0)               |
| “USRDATA” | Flag indicating user input data. (Character).                    |

#### Remarks:

1. SEDRSP3 is provided specifically for creating external response with quantities from multiple PART SEs and must be placed in main Bulk Data Section for consideration. SEDRSP3 will be ignored if it shows up under ‘BEGIN SUPER=seid’ where seid>0.
2. Items under DTABLE and DNODE flag requires companion items in the same SEID under flags of DESVAR. DVxRELy or DRESP1.
3. Other than the leading PART SE ID for each quantity, SEDRSP3 follows rules and shares limitations of DRESP3. Rules and limitations of DRESP3 will not be repeated here.
4. SEDRSP3 does not support DRESP2 flag.

## SEELT Superelement Boundary Element Reassignment

Reassigns superelement boundary elements to an upstream superelement.

### Format:

|       |      |      |        |      |      |      |      |      |    |
|-------|------|------|--------|------|------|------|------|------|----|
| 1     | 2    | 3    | 4      | 5    | 6    | 7    | 8    | 9    | 10 |
| SEELT | SEID | EID1 | EID2   | EID3 | EID4 | EID5 | EID6 | EID7 |    |
|       | EID8 | EID9 | -etc.- |      |      |      |      |      |    |

### Example:

|       |   |     |     |     |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|
| SEELT | 2 | 147 | 562 | 937 |  |  |  |  |  |
|-------|---|-----|-----|-----|--|--|--|--|--|

### Alternate Format and Example:

|       |      |       |        |       |  |  |  |  |  |
|-------|------|-------|--------|-------|--|--|--|--|--|
| SEELT | SEID | EID1  | “THRU” | EID2  |  |  |  |  |  |
| SEELT | 5    | 12006 | THRU   | 12050 |  |  |  |  |  |

| Describer | Meaning                                                                                 |
|-----------|-----------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. See Remark 7. (Integer > 0)                         |
| EIDi      | Element identification numbers. (Integer > 0 or “THRU”; for “THRU” option EID1 < EID2.) |

### Remarks:

1. Elements connected entirely to the exterior points of an upstream superelement are called boundary elements and are assigned to the downstream superelement. The SEELT entry provides the means of reassigning the element to the upstream superelement. This entry may be applied to boundary elements only.
2. Open sets are allowed with the “THRU” option.
3. Elements processed with primary superelements will also be contained in any referencing secondary superelement.
4. EIDi may refer to plot elements, general elements, and structural elements.
5. This entry does not change the exterior grid point set of the superelement.
6. SEELT can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
7. If the model contains a BEGIN SUPER, the SEELT entry will assign the specified elements to the SEID (not just the boundary elements). This is an alternative to using SESET. In this case, SEID = 0 is a valid entry. For further information, refer to the [MSC Nastran Reference Guide](#).

**SEEXCLD**

## Partitioned Superelement Exclusion

Defines grids that will be excluded during the attachment of a partitioned superelement.

**Format:**

| 1       | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| SEEXCLD | SEIDA | SEIDB | GIDA1  | GIDA2 | GIDA3 | GIDA4 | GIDA5 | GIDA6 |    |
|         | GIDA7 | GIDA8 | -etc.- |       |       |       |       |       |    |

**Example1:**

|         |     |    |    |     |     |  |  |  |  |
|---------|-----|----|----|-----|-----|--|--|--|--|
| SEEXCLD | 110 | 10 | 45 | 678 | 396 |  |  |  |  |
|---------|-----|----|----|-----|-----|--|--|--|--|

**Example 2:**

|         |     |     |    |    |    |      |    |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|
| SEEXCLD | 400 | ALL | 10 | 20 | 30 | THRU | 40 |  |  |
|---------|-----|-----|----|----|----|------|----|--|--|

| Describer | Meaning                                                                                                                                                                   |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEIDA     | Partitioned superelement identification number. (Integer $\geq 0$ )                                                                                                       |
| SEIDB     | Superelement identification. (Integer $\geq 0$ or Character = "ALL"; Default = "ALL")                                                                                     |
| GIDAi     | Identification number of a grid in superelement SEIDA to be excluded from connection to superelement SEIDB. (Integer $> 0$ or "THRU"; for "THRU" option GIDA1 $<$ GIDA2.) |

**Remarks:**

1. SEEXCLD can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. SEIDA and SEIDB may reference only substructures or the residual structure, that is, parts defined after a BEGIN SUPER = entry.
3. This entry will only work if PART superelements (BEGIN SUPER) exist.

## SELABEL Superelement Output Label

Defines a label or name to be printed in the superelement output headings.

### Format:

|         |      |       |   |   |   |   |   |   |    |  |
|---------|------|-------|---|---|---|---|---|---|----|--|
| 1       | 2    | 3     | 4 | 5 | 6 | 7 | 8 | 9 | 10 |  |
| SELABEL | SEID | LABEL |   |   |   |   |   |   |    |  |

### Example:

|         |    |                                 |  |  |  |  |  |  |  |  |
|---------|----|---------------------------------|--|--|--|--|--|--|--|--|
| SELABEL | 10 | LEFT REAR FENDER, MODEL XYZ2000 |  |  |  |  |  |  |  |  |
|---------|----|---------------------------------|--|--|--|--|--|--|--|--|

| Describer | Meaning                                                                  |
|-----------|--------------------------------------------------------------------------|
| SEID      | Partitioned superelement identification number. (Integer > 0)            |
| LABEL     | Label associated with superelement SEID for output headings. (Character) |

### Remarks:

1. SELABEL can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER=n command.
2. Only one SELABEL per superelement may be specified.
3. The label will appear in all superelement output headings. However, in some headings the label may be truncated.
4. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SELOAD****Selects and/or Scales External Superelement Loads**

Selects and/or scales external superelement loads identified by a label, subcase identification number, load case identification number or column number.

**Format:**

| 1      | 2                                            | 3              | 4                   | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----------------------------------------------|----------------|---------------------|---|---|---|---|---|----|
| SELOAD | SID                                          | S              |                     |   |   |   |   |   |    |
|        | SEID <sub>i</sub>                            | S <sub>i</sub> | LBLTYP <sub>i</sub> |   |   |   |   |   |    |
|        | Label, Subcase ID, Load ID, or Column Number |                |                     |   |   |   |   |   |    |
|        | -etc.-                                       |                |                     |   |   |   |   |   |    |
|        | -etc.-                                       |                |                     |   |   |   |   |   |    |

**Example:**

|        |           |     |       |  |  |  |  |  |  |
|--------|-----------|-----|-------|--|--|--|--|--|--|
| SELOAD | 11        | 1.0 |       |  |  |  |  |  |  |
|        | 100       | 0.5 |       |  |  |  |  |  |  |
|        | edge load |     |       |  |  |  |  |  |  |
|        | 200       | 1.2 | SUBID |  |  |  |  |  |  |
|        | 10        |     |       |  |  |  |  |  |  |

| Describer           | Meaning                                                                                                                                                                                                                               |
|---------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID                 | LOAD Case Control command identification number if static analysis or EXCITEID on TLOAD <sub>i</sub> and RLOAD <sub>i</sub> entries if dynamic analysis.                                                                              |
| S                   | Overall scale factor.                                                                                                                                                                                                                 |
| S <sub>i</sub>      | Scale factor for this superelement's load vector                                                                                                                                                                                      |
| LBLTYP <sub>i</sub> | Label type in the in the next entry: "LABEL" (default), "SUBID", "LOADID", or "COLNUM".                                                                                                                                               |
| Label               | If LBLTYP <sub>i</sub> ="LABEL" or "SUBCASE": String assigned to a column in the external superelement load matrix in the creation run by the LDLABEL Case Control command.                                                           |
| Subcase ID          | If LBLTYP <sub>i</sub> ="SUBID": Subcase identification number associated with a column in the external superelement load matrix.                                                                                                     |
| Load ID             | If LBLTYP <sub>i</sub> ="LOADID": Load identification number associated with a column in the external superelement load matrix.                                                                                                       |
| Column number       | If LBLTYP <sub>i</sub> ="COLNUM" or "COLUMN": Column number in the external superelement load matrix. Column number allows for load selection in pre-V2021.4 external superelement export files (.op2, .op4, .MASTER/.DBALL, or pch). |

**Remarks:**

1. In the static solution sequences, SID is selected by the LOAD Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
2. Multiple SELOAD entries may be specified but SID must be unique.
3. SEIDi may be duplicated on an SELOAD entry but may not select the same external superelement load case.
4. If SEIDi refers to a primary (internal) superelement or an undefined superelement then a User Fatal Message is issued.
5. If the LDLABEL Case Control command was not specified in the external superelement's creation run and SELOAD references this superelement in the assembly run then LBLTYPi must be "COLNUM" and the next continuation entry must contain a column number.
6. If in the assembly run there is no SELOAD specified for a given load case then a User Warning Message will be issued.
7. The LOAD Bulk Data entry will not combine SELOAD entries.

## SELOC Partitioned Superelement Location

Defines a partitioned superelement relocation by listing three noncolinear points in the superelement and three corresponding points not belonging to the superelement.

### Format:

| 1     | 2    | 3   | 4   | 5   | 6   | 7   | 8   | 9 | 10 |
|-------|------|-----|-----|-----|-----|-----|-----|---|----|
| SELOC | SEID | PA1 | PA2 | PA3 | PB1 | PB2 | PB3 |   |    |

### Example:

|       |     |    |     |     |      |     |    |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|
| SELOC | 110 | 10 | 100 | 111 | 1010 | 112 | 30 |  |  |
|-------|-----|----|-----|-----|------|-----|----|--|--|

| Describer | Meaning                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Partitioned identification number of the partitioned superelement. (Integer > 0)                                                                             |
| PAi       | Identification numbers of three noncolinear grids (GRID entry) or points (POINT entry) which are in the partitioned superelement. (Integer > 0)              |
| PBi       | Identification numbers of three grids (GRID entry) or points (POINT entry) defined in the main Bulk Data Section to which PAi will be aligned. (Integer > 0) |

### Remarks:

- SELOC can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
- The superelement will be rotated and translated for alignment of the G<sub>Ai</sub> and G<sub>Bi</sub> locations.
- The PA<sub>i</sub> and PB<sub>i</sub> can be either GRIDs or POINTs.
- PA<sub>1</sub>, PA<sub>2</sub>, and PA<sub>3</sub> must be contained in superelement SEID.
- PB<sub>1</sub>, PB<sub>2</sub>, and PB<sub>3</sub> must be specified in the main Bulk Data Section. If they belong to a superelement that is also relocated, then the original (unmoved) positions of PB<sub>1</sub>, PB<sub>2</sub>, and PB<sub>3</sub> are used.
- PB<sub>1</sub>, PB<sub>2</sub>, and PB<sub>3</sub> must have the same relative locations as PA<sub>1</sub>, PA<sub>2</sub>, and PA<sub>3</sub>.
- Three grids or points are required even if the superelement connects to only one or two exterior grids.
- Coordinate systems, global displacement directions, and element coordinate systems for the superelement will rotated and translated.
- The global coordinate directions of the boundary grid points of the upstream superelement will be transformed internally to the global coordinate directions of the attachment grid points in the downstream superelement. For displacement data recovery, the output will be in the original global coordinate system.
- The translation and rotation of the superelement to the new position is accomplished by defining local rectangular coordinate systems based on the specified grid locations:

- The local systems have their origin at PX1 and the x-axis points from PX1 to PX2.
  - The y-axis lies in the plane containing PX1, PX2, and PX3, is perpendicular to the x-axis, and points toward PX3.
  - The z-axis is defined by the cross product of the x-axis into the y-axis.
  - The rotation and translation transformation aligns the local system defined by the superelement grids with the local system defined by the main Bulk Data Section grids.
11. This entry will only work if PART superelements (BEGIN SUPER) exist.



## SEMPLN Superelement Mirror Plane

Defines a mirror plane for mirroring a partitioned superelement.

### Format:

| 1      | 2    | 3       | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
|--------|------|---------|----|----|----|---|---|---|----|
| SEMPLN | SEID | "PLANE" | P1 | P2 | P3 |   |   |   |    |

### Example:

|        |     |       |    |    |      |  |  |  |  |
|--------|-----|-------|----|----|------|--|--|--|--|
| SEMPLN | 110 | PLANE | 12 | 45 | 1125 |  |  |  |  |
|--------|-----|-------|----|----|------|--|--|--|--|

| Describer | Meaning                                                                                |
|-----------|----------------------------------------------------------------------------------------|
| SEID      | Partitioned superelement identification number. (Integer > 0).                         |
| "PLANE"   | Flag indicating that the plane is defined by three noncolinear points.                 |
| Pi        | GRID or POINT entry identification numbers of three noncolinear points. (Integer > 0). |

### Remarks:

1. SEMPLN can only be specified in the main Bulk Data Section and will cause a fatal error message if it appears after the BEGIN SUPER=n command.
2. Grids or points referenced on this entry must be defined in the main Bulk Data Section.

**SENQSET** Superelement Internal Generalized Degree-of-Freedom

Defines number of internally generated scalar points for superelement dynamic reduction.

**Format:**

|         |      |   |   |   |   |   |   |   |    |
|---------|------|---|---|---|---|---|---|---|----|
| 1       | 2    | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
| SENQSET | SEID | N |   |   |   |   |   |   |    |

**Example:**

|         |     |    |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|
| SENQSET | 110 | 45 |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|

| Descriptor | Meaning                                                                                                                |
|------------|------------------------------------------------------------------------------------------------------------------------|
| SEID       | Partitioned superelement identification number. See Remark 3. (Integer > 0 or Character = "ALL")                       |
| N          | Number of internally generated scalar points for dynamic reduction generalized coordinates. (Integer > 0; Default = 0) |

**Remarks:**

1. SENQSET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.
2. SENQSET is only required if the user wants to internally generated scalar points used for dynamic reduction.
3. SEID = "ALL" will automatically generate N q-set degrees-of-freedom for all superelements, except the residual structure (SEID = 0). Specifying additional SENQSET entries for specific superelements will override the value of N specified on this entry.
4. If the user manually specifies q-set degrees-of-freedom using a SEQSETi or QSETi entries, then the internally generated scalar points will not be generated.
5. See PARAM,NQSET for an alternate method of specifying QSET degree-of-freedoms.
6. This entry will only work if PART superelements (BEGIN SUPER) exist.

**SEQGP****Grid and Scalar Point Resequencing**

Used to manually order the grid points and scalar points of the problem. This entry is used to redefine the sequence of grid and scalar points to optimize bandwidth.

**Format:**

| 1     | 2   | 3      | 4   | 5      | 6   | 7      | 8   | 9      | 10 |
|-------|-----|--------|-----|--------|-----|--------|-----|--------|----|
| SEQGP | ID1 | SEQID1 | ID2 | SEQID2 | ID3 | SEQID3 | ID4 | SEQID4 |    |

**Example:**

|       |      |      |     |     |   |     |   |   |  |
|-------|------|------|-----|-----|---|-----|---|---|--|
| SEQGP | 5392 | 15.6 | 596 | 0.2 | 2 | 1.9 | 3 | 2 |  |
|-------|------|------|-----|-----|---|-----|---|---|--|

| Describer | Meaning                                                      |
|-----------|--------------------------------------------------------------|
| IDI       | Grid or scalar point identification number. (Integer > 0)    |
| SEQIDI    | Sequenced identification number. (Real > 0.0 or Integer > 0) |

**Remarks:**

1. The real format is used to insert a point ID between two consecutively numbered and existing point IDs. In the example above, point ID 5392 is inserted between IDs 15 and 16 by specifying 15.6 for SEQID. If the SEQID is real and < 1.0 the value must be entered as 0.2 not .2 (see field 5 of example).
2. The SEQIDI numbers must be unique and may not be the same as a point IDi which is not being changed. No grid point IDi may be referenced more than once.
3. From one to four grid or scalar points may be resequenced on a single entry.
4. If a point IDi is referenced more than once, the last reference will determine its sequence.
5. Automatic resequencing is also available. See [OLDSEQ](#).

## SEQSEP Superelement Sequences

Used with the CSUPER entry to define the correspondence of the exterior grid points between an identical or mirror-image superelement and its primary superelement.

**Format:**

|        |      |      |        |     |     |     |     |     |    |
|--------|------|------|--------|-----|-----|-----|-----|-----|----|
| 1      | 2    | 3    | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
| SEQSEP | SSID | PSID | GP1    | GP2 | GP3 | GP4 | GP5 | GP6 |    |
|        | GP7  | GP8  | -etc.- |     |     |     |     |     |    |

**Example:**

|        |     |    |     |     |     |     |  |  |  |
|--------|-----|----|-----|-----|-----|-----|--|--|--|
| SEQSEP | 121 | 21 | 109 | 114 | 124 | 131 |  |  |  |
|--------|-----|----|-----|-----|-----|-----|--|--|--|

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

- SSID Identification number for secondary superelement. (Integer > 0).
- PSID Identification number for the primary superelement. (Integer ≥ 0).
- GPI Exterior grid point identification numbers for the primary superelement. (Integer > 0).

**Remarks:**

- This entry is not needed if the grid points listed on the CSUPER entry with the same SSID are in the order of the corresponding exterior grid points of the primary superelement.
- In [Figure 9-142](#), the exterior grid points of 10, 20, and 30 of SEID = 1 correspond to the points 13, 12, and 11, respectively, of image SEID = 2. The CSUPER entry may be defined alone or with a SEQSEP entry as shown in [Figure 9-142](#).

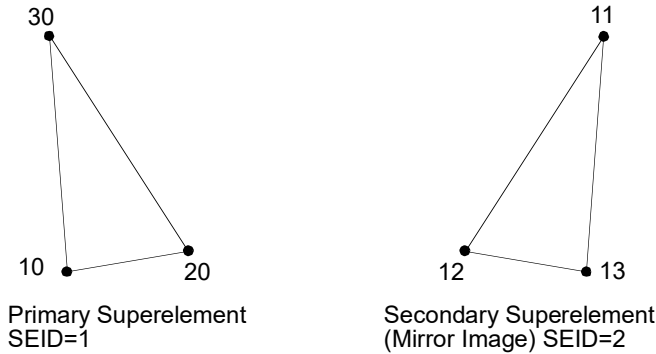


Figure 9-142 Grid Point Correspondence Between Primary and Secondary Superelements

**CSUPER Entry Only:**

|        |   |   |    |    |    |   |   |   |    |
|--------|---|---|----|----|----|---|---|---|----|
| 1      | 2 | 3 | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
| CSUPER | 2 | 1 | 13 | 12 | 11 |   |   |   |    |

**CSUPER and SEQSEP Entries:**

|        |   |   |    |    |    |  |  |  |  |
|--------|---|---|----|----|----|--|--|--|--|
| CSUPER | 2 | 1 | 11 | 12 | 13 |  |  |  |  |
| SEQSEP | 2 | 1 | 30 | 20 | 10 |  |  |  |  |

## SEQSET

## Superelement Generalized Degree-of-Freedom

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

### Format:

| 1      | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|--------|------|-----|----|-----|----|-----|----|---|----|
| SEQSET | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

### Example:

|        |    |   |        |   |   |    |   |  |  |
|--------|----|---|--------|---|---|----|---|--|--|
| SEQSET | 15 | 1 | 123456 | 7 | 5 | 22 | 3 |  |  |
|--------|----|---|--------|---|---|----|---|--|--|

| Describer | Meaning                                                                                                                                                  |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. Must be a primary superelement. (Integer > 0)                                                                        |
| Ci        | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer zero or blank for scalar points.) |
| Idi       | Grid or scalar point identification numbers. Must be an exterior point. (Integer > 0)                                                                    |

### Remarks:

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of the mutually exclusive q-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user's responsibility to place them in a downstream superelement.

Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these degrees-of-freedom for secondary superelements.

## SEQSET1

## Superelement Generalized Degree-of-Freedom, Alternate Form

Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.

### Format:

|         |      |    |        |    |    |    |    |    |    |
|---------|------|----|--------|----|----|----|----|----|----|
| 1       | 2    | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
| SEQSET1 | SEID | C  | G1     | G2 | G3 | G4 | GS | G6 |    |
|         | G7   | G8 | -etc.- |    |    |    |    |    |    |

### Example:

|         |    |        |   |   |   |    |     |   |  |
|---------|----|--------|---|---|---|----|-----|---|--|
| SEQSET1 | 15 | 123456 | 1 | 7 | 9 | 22 | 105 | 6 |  |
|         | 52 | 53     |   |   |   |    |     |   |  |

### Alternate Format and Example:

|         |      |   |     |        |     |  |  |  |  |
|---------|------|---|-----|--------|-----|--|--|--|--|
| SEQSET1 | SEID | C | G1  | “THRU” | G2  |  |  |  |  |
| SEQSET1 | 16   | 0 | 101 | THRU   | 110 |  |  |  |  |

| Describer | Meaning                                                                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. Must be a primary superelement. (Integer > 0)                                                                      |
| C         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi        | Grid or scalar point identification numbers. Must be exterior points. (Integer > 0 or “THRU”; for THRU option G1 < G2.)                                |

### Remarks:

- Degrees-of-freedom specified on this entry may not be specified for another superelement.
- Generalized degrees-of-freedom are interior to the residual structure.
- Connectivity to the superelement is provided by this entry. There is no need to use a CSUPEXT entry for this purpose.
- Degrees-of-freedom specified on this entry form members of a mutually exclusive set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
- This entry describes the set used for generalized degrees-of-freedom only for the SEID listed. Degrees-of-freedom listed on this entry must also be members of a downstream superelement. The set used for these variables in downstream superelements must be prescribed by user action using other entries. If they are scalar points, they are automatically in the residual structure, which is the recommended procedure. If they are grid points, it is the user’s responsibility to place them in a downstream superelement.



Generalized degrees-of-freedom of superelements that are also members of the residual structure are included as dynamic variables by placing them in the a-set. It is also necessary to place some or all residual structure physical degrees-of-freedom in the a-set, to allow the boundary points to participate in the system mode shapes.

Grid points of downstream superelements used as generalized degrees-of-freedom may be used for advanced applications, such as omitting upstream generalized degrees-of-freedom from assembly into downstream superelements. Again, it is the user's responsibility to place these variables in the proper set in all downstream superelements of which they are members.

6. This entry may be applied only to primary superelements. The CSUPER entry automatically defines these entries for secondary superelements.

## SESET Superlement Interior Point Definition

Defines interior grid points for a superlement.

### Format:

| 1     | 2    | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|-------|------|----|----|----|----|----|----|----|----|
| SESET | SEID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

### Example:

|       |   |   |    |    |    |     |  |  |  |
|-------|---|---|----|----|----|-----|--|--|--|
| SESET | 5 | 2 | 17 | 24 | 25 | 165 |  |  |  |
|-------|---|---|----|----|----|-----|--|--|--|

### Alternate Format and Example:

|       |      |    |        |     |  |  |  |  |  |
|-------|------|----|--------|-----|--|--|--|--|--|
| SESET | SEID | G1 | “THRU” | G2  |  |  |  |  |  |
| SESET | 2    | 17 | THRU   | 165 |  |  |  |  |  |

| Describer | Meaning                                                                                      |
|-----------|----------------------------------------------------------------------------------------------|
| SEID      | Superlement identification number. Must be a primary superlement. (Integer $\geq 0$ )        |
| Gi        | Grid or scalar point identification number. ( $0 < \text{Integer} < 100000000$ ; $G1 < G2$ ) |

### Remarks:

1. Interior grid points may also be defined via field 9 of the GRID and GRIDG Bulk Data entries. The SESET entry takes precedence over the SEID field on the GRID on GRIDG entries. SESET defines grid and scalar points to be included as interior to a superlement. SESET may be used as the primary means of defining superlements or it may be used in combination with SEELT entries which define elements interior to a superlement.
2. Gi may appear on an SESET entry only once.
3. Scalar points are ignored unless a BEGIN SUPER is in the file.
4. Open sets are allowed with the “THRU” option. Missing grid points (whether in “THRU” range or mentioned explicitly) are not identified.
5. All degrees-of-freedom for Gi are placed in the o-set of the superlement. See [Degree-of-Freedom Sets](#).
6. SESET can only be specified in the main Bulk Data Section and is ignored after the BEGIN SUPER = n command.

## SESUP Fictitious Support

Defines determinate reaction superelement degrees-of-freedom in a free-body analysis.

### Format:

| 1     | 2    | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|-------|------|-----|----|-----|----|-----|----|---|----|
| SESUP | SEID | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

### Example:

|       |   |    |     |  |  |  |  |  |  |
|-------|---|----|-----|--|--|--|--|--|--|
| SESUP | 5 | 16 | 215 |  |  |  |  |  |  |
|-------|---|----|-----|--|--|--|--|--|--|

| Describer | Meaning                                                                                                                           |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. Must a primary superelement. (Integer > 0)                                                    |
| IDI       | Grid or scalar point identification number. Must be exterior points. (Integer > 0)                                                |
| Ci        | Component numbers. (Integer zero or blank for scalar points; Any unique combination of the Integers 1 through 6 for grid points.) |

### Remarks:

1. The degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
2. The Ci degrees-of-freedom must be exterior degrees-of-freedom of the SEID superelement.
3. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
4. There must be a sufficient number of degrees-of-freedom on SESUP entries to discard any free body modes of the superelement.
5. SESUP Bulk Data entries are not allowed for part (partitioned bulk data) superelements. Use the SUPPORT Bulk Data records to identify component rigid body modes.

**SET1****Set Definition**

Defines a list of structural grid points or element identification numbers.

**Format:**

| 1    | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| SET1 | SID | ID1    | ID2 | ID3 | ID4 | ID5 | ID6 | ID7 |    |
|      | ID8 | -etc.- |     |     |     |     |     |     |    |

**Example 1:**

|      |    |    |    |    |     |    |    |    |  |
|------|----|----|----|----|-----|----|----|----|--|
| SET1 | 3  | 31 | 62 | 93 | 124 | 16 | 17 | 18 |  |
|      | 19 |    |    |    |     |    |    |    |  |

**Example 2:**

|      |    |    |    |      |    |    |      |    |  |
|------|----|----|----|------|----|----|------|----|--|
| SET1 | 6  | 29 | 32 | THRU | 50 | 61 | THRU | 70 |  |
|      | 17 | 57 |    |      |    |    |      |    |  |

**Example 3:**

|      |   |      |  |  |  |  |  |  |  |
|------|---|------|--|--|--|--|--|--|--|
| SET1 | 7 | SKIN |  |  |  |  |  |  |  |
|------|---|------|--|--|--|--|--|--|--|

| Describer | Meaning                                                                                                                                          |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Unique identification number. (Integer > 0)                                                                                                      |
| IDI       | List of structural grid point or element identification numbers. (Integer > 0 or "THRU"; for the "THRU" option, ID1 < ID2 or "SKIN"; in field 3) |

**Remarks:**

1. When using the "THRU" option for SPLINEi or PANEL data entries, all intermediate grid points must exist.
2. When using the "THRU" option for XYOUTPUT or AECOMP requests, missing grid points are ignored. The first and last points must exist.
3. When using the "SKIN" option, a panel will be generated consisting of the structural portion of the fluid-structural boundary. This option works ONLY with all fields of ACMODL having default value.
4. THRU may not appear in field 3 or 9 (2 or 9 for continuations).
5. RC network solver does not support SET1 for thermal analysis.

## SET2 Grid Point List

Defines a list of structural grid points in terms of aerodynamic macro elements.

### Format:

| 1    | 2   | 3     | 4   | 5   | 6   | 7   | 8    | 9    | 10 |
|------|-----|-------|-----|-----|-----|-----|------|------|----|
| SET2 | SID | MACRO | SP1 | SP2 | CH1 | CH2 | ZMAX | ZMIN |    |

### Example:

|      |   |     |     |      |     |       |      |  |  |
|------|---|-----|-----|------|-----|-------|------|--|--|
| SET2 | 3 | 111 | 0.0 | 0.75 | 0.0 | 0.667 | 3.51 |  |  |
|------|---|-----|-----|------|-----|-------|------|--|--|

| Describer     | Meaning                                                                                                                                               |
|---------------|-------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID           | Unique identification number. (Integer > 0)                                                                                                           |
| MACRO         | Element identification number of an aerodynamic macro element. (Integer > 0)                                                                          |
| SP1, SP2      | Lower and higher span division points defining the prism containing the set. (Real)                                                                   |
| CH1, CH2      | Lower and higher chord division points defining the prism containing the set. (Real)                                                                  |
| ZMAX,<br>ZMIN | Z-coordinates of top and bottom (using right-hand rule with the order of the corners as listed on a CAEROi entry) of the prism containing set. (Real) |

### Remarks:

1. The SET2 entry is referenced by the SPLINEi entry.
2. Every grid point within the defined prism and within the height range will be in the list. For example:

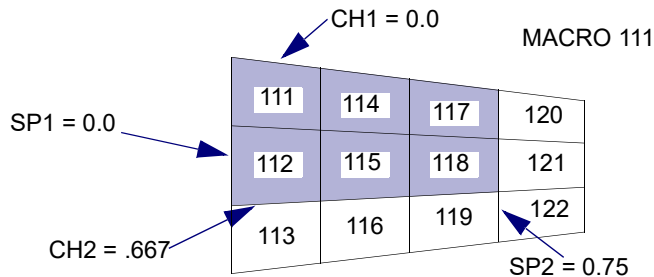


Figure 9-143 SET2 Entry Example.

The shaded area in [Figure 9-143](#) defines the cross section of the prism for the sample data given above. Points exactly on the boundary may be missed; therefore, to get all the grid points within the area of the macro element, SP1=-.01, SP2=1.01, etc. should be used.

3. A zero value for ZMAX or ZMIN implies a value of infinity. Usually,  $ZMAX \geq 0.0$  and  $ZMIN \leq 0.0$ .
4. To print the (internal) grid IDs found, use DIAG 18.

**SET3** Defines a List of Grids, Elements, Points or Modules

Defines a list of grids, elements, points or modules.

**Format:**

|      |     |     |     |     |     |     |   |   |    |
|------|-----|-----|-----|-----|-----|-----|---|---|----|
| 1    | 2   | 3   | 4   | 5   | 6   | 7   | 8 | 9 | 10 |
| SET3 | SID | DES | ID1 | ID2 | ... | Idi |   |   |    |

**Example:**

|      |   |       |    |    |    |    |    |    |  |
|------|---|-------|----|----|----|----|----|----|--|
| SET3 | 1 | POINT | 11 | 12 | 13 | 15 | 18 | 21 |  |
|------|---|-------|----|----|----|----|----|----|--|

**Alternate Format and Example:**

|      |     |       |     |        |     |  |  |  |  |
|------|-----|-------|-----|--------|-----|--|--|--|--|
| SET3 | SID | DES   | ID1 | "THRU" | ID2 |  |  |  |  |
| SET3 | 33  | POINT | 20  | THRU   | 60  |  |  |  |  |

| Describer | Meaning                                                                                                            |
|-----------|--------------------------------------------------------------------------------------------------------------------|
| SID       | Unique identification number. (Integer > 0)                                                                        |
| DES       | Set description (Character). Valid options are "GRID", "ELEM", "POINT", "PROP", "RBEin", and "RBEex" and "MODULE". |
| Idi       | Identifiers of grids points, elements, points or properties. (Integer ≥ 0)                                         |

**Remarks:**

1. If a SET3 entry is referenced on a PBMSECT or PBRSECT entry, the POINTs must lie in the (xy) plane of the basic coordinate system, and be in the order when traversing the boundary or the profile.
2. When the SET3 entry is referenced by a panel, describers can be "GRID", "ELEM" or "PROP".
3. THRU may not appear in field 4 or 9 (2 or 9 for continuations).
4. When SET3 is referenced by SOLs 400 entry only GRID or ELEM may be used.
5. When SET3 is referenced by RFORCE (IDRF field) for SOL 600, only ELEM may be used.
6. When SET3 is referenced by SOL 400 DEACTEL entries, only ELEM may be used.
7. When a SET3 is referenced by a ELSIDi or XELSIDi field on an FTGDEF entry, only ELEM may be used. When SET3 is referenced by a NDSIDi field on a FTGDEF entry, only GRID may be used.
8. When DES="RBEin", the SET selects rigid elements to be included for MPC=sid and is applicable to Rigid Element types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1. Note that Rigid Elements with duplicate ID across Rigid Element types will all be utilized.
9. For DES="RBEex", the SET selects rigid elements to be excluded for MPC=sid and is applicable to Rigid Elements types of RBAR, RBAR1, RBE1, RBE2, RBE2GS, RBE3, RROD, RSPLINE, RSSCON, RTRPLT and RTRPLT1.



10. Note that "RBEin" and "RBEex" are mutually exclusive and should not appear together for a single SET.
11. By default, without SET3,SID,RBExx, all Rigid Elements in the input deck will be used.
12. SET selection for rigid elements does not cover additional IDs on MPCADD bulk data entry. This kind of selection is not supported in SOL 106 either.
13. Rigid element set selection is supported in SOL 400 if 'RIGID=LINEAR' is present in case control deck.
14. IDi can be 0 when DES is "MODULE", otherwise it must be greater than 0.

## SET4

### Property Set Definition

Defines a list of property IDs

#### Format:

| 1    | 2   | 3     | 4    | 5     | 6   | 7   | 8   | 9   | 10 |
|------|-----|-------|------|-------|-----|-----|-----|-----|----|
| SET4 | ID  | CLASS | TYPE | ID1   | ID2 | ID3 | ID4 | ID5 |    |
|      | ID6 | ID7   | ID8  | -etc- |     |     |     |     |    |

#### Example:

|      |    |      |        |   |      |    |  |  |  |
|------|----|------|--------|---|------|----|--|--|--|
| SET4 | 22 | PROP | PSOLID | 1 | THRU | 20 |  |  |  |
|------|----|------|--------|---|------|----|--|--|--|

| Describer | Meaning                                                                          |
|-----------|----------------------------------------------------------------------------------|
| ID        | Unique identification number. (Integer>0).                                       |
| CLASS     | Set to "PROP". (Character = PROP; no default)                                    |
| TYPE      | Property type. Valid options are PSOLID, PSHELL, PSHEAR, PBAR, PBEAM, and PWELD. |
| IDI       | Property IDs of the specified TYPE flag.                                         |

#### Remarks:

1. THRU option may not appear in field 5 or 9 on first line or field 2 or 9 for continuation lines.
2. Currently referenced from the FTGDEF entry.



## SETREE

### Superelement Tree Definition (Alternate Form of DTI,SETREE)

Specifies superelement reduction order.

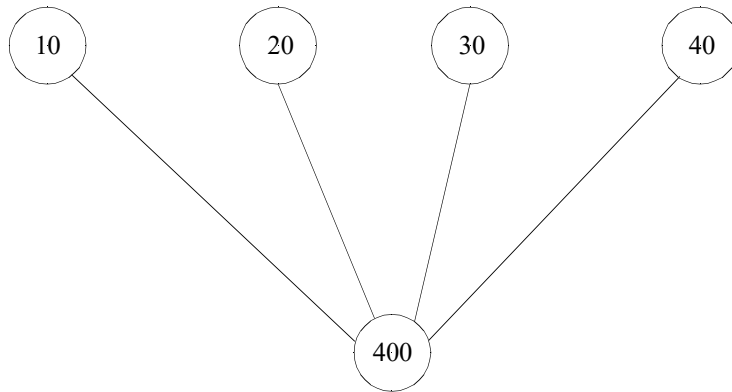
#### Format:

|        |       |       |        |       |       |       |       |       |    |
|--------|-------|-------|--------|-------|-------|-------|-------|-------|----|
| 1      | 2     | 3     | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
| SETREE | SEID  | SEUP1 | SEUP2  | SEUP3 | SEUP4 | SEUP5 | SEUP6 | SEUP7 |    |
|        | SEUP8 | SEUP9 | -etc.- |       |       |       |       |       |    |

#### Example:

|        |     |    |    |    |    |  |  |  |  |
|--------|-----|----|----|----|----|--|--|--|--|
| SETREE | 400 | 10 | 20 | 30 | 40 |  |  |  |  |
|--------|-----|----|----|----|----|--|--|--|--|

| Describer         | Meaning                                                                                          |
|-------------------|--------------------------------------------------------------------------------------------------|
| SEID              | Partitioned superelement identification number of a downstream superelement. (Integer $\geq 0$ ) |
| SEUP <sub>i</sub> | Identification number of superelements that are upstream of SEID. (Integer $> 0$ )               |



#### Remarks:

1. SETREE entries or DTI,SETREE entry are required for multilevel superelement configurations.
2. At least one SETREE entry is required for each nontip superelement, including the residual structure (SEID = 0). Multiple SETREE entries with the same SEID are allowed.
3. A superelement may appear only once in an SEUP<sub>i</sub> field on all SETREE entries.
4. If an DTI,SETREE entry is provided, then SETREE entries are not required.
5. If both SETREE entries and a DTI,SETREE entry exist, then the DTI,SETREE entry will be ignored.

6. If a superelement is not referenced on the DTI,SETREE or SETREE entry, then the manner in which it is handled depends on the type of that superelement. If it is a PART superelement, then the residual will be regarded as its downstream superelement and the undefined superelement will therefore be placed immediately above the residual in the tree. If it is a Main Bulk Data superelement, then it will also be handled like an undefined PART superelement as above *if all of its exterior points belong to the residual*. However, if one or more of its exterior points do not belong to the residual, then the program will terminate with a user fatal error complaining that one or more of the superelements are not in the same path.
7. The SETREE entry will only work if PART (BEGIN SUPER) superelements exist in the model. If there are no PARTs in the model, the SETREE entries will be ignored.

## SEUSET Superelement Degree-of-Freedom Set Definition

Defines a degree-of-freedom set for a superelement.

### Format:

| 1      | 2    | 3     | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|--------|------|-------|-----|----|-----|----|-----|----|----|
| SEUSET | SEID | SNAME | ID1 | C1 | ID2 | C2 | ID3 | C3 |    |

### Example:

|        |    |    |   |        |   |   |    |   |  |
|--------|----|----|---|--------|---|---|----|---|--|
| SEUSET | 15 | U4 | 1 | 123456 | 7 | 5 | 22 | 3 |  |
|--------|----|----|---|--------|---|---|----|---|--|

| Describer | Meaning                                                                                                                                              |
|-----------|------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. (Integer > 0)                                                                                                    |
| SNAME     | Set name. (One to four characters or string "ZERO", followed by the set name.)                                                                       |
| IDI       | Grid or scalar point identification numbers. (Integer > 0)                                                                                           |
| CI        | Component number. (Any unique combination of the Integers 1 through 6 with no embedded blank for grid points; Integer 0 or blank for scalar points.) |

### Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names "U1" through "U6" or their new names on the DEFUSET entry.
2. If SNAME = "ZEROi", where i is a set name, then the degrees-of-freedom are omitted from set i.

## SEUSET1 Superelement Degree-of-Freedom Set Definition, Alternate Form

Defines a degree-of-freedom set for a superelement.

### Format:

| 1       | 2    | 3     | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|-------|--------|----|----|----|----|----|----|
| SEUSET1 | SEID | SNAME | C      | G1 | G2 | G3 | G4 | G5 |    |
|         | G6   | G7    | -etc.- |    |    |    |    |    |    |

### Example:

|         |    |    |   |    |    |    |    |    |  |
|---------|----|----|---|----|----|----|----|----|--|
| SEUSET1 | 15 | U4 | 1 | 12 | 15 | 17 | 22 | 25 |  |
|         | 52 | 53 |   |    |    |    |    |    |  |

### Alternate Format and Example:

|         |      |       |   |    |        |    |  |  |  |
|---------|------|-------|---|----|--------|----|--|--|--|
| SEUSET1 | SEID | SNAME | C | G1 | "THRU" | G2 |  |  |  |
| SEUSET1 | 15   | U4    | 1 | 12 | THRU   | 27 |  |  |  |

| Describer | Meaning                                                                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| SEID      | Superelement identification number. (Integer > 0)                                                                                                      |
| SNAME     | Set name. (One to four characters or string "ZERO", followed by the set name.)                                                                         |
| C         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0 or blank for scalar points.) |
| Gi        | Grid or scalar point identification number. (Integer > 0)                                                                                              |

### Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets](#) or their new names on the DEFUSET entry. However, in the Solution Sequences 0 through 200, it is recommended that SNAME refer only to the set names "U1" through "U6" or their new names on the DEFUSET entry.
2. If SNAME= "ZEROi", where i is a set name, then the degrees-of-freedom are omitted from set i.
3. If the alternate format is used, all of the points G1 through G2 are assigned to the set.

**SHREL**

## Elastic Shear Model

Defines an elastic shear model with a constant shear modulus. Used in SOL 700 only.

**Format:**

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|---|---|---|---|---|---|----|
| SHREL | SID | G |   |   |   |   |   |   |    |

**Example:**

|       |     |       |  |  |  |  |  |  |  |
|-------|-----|-------|--|--|--|--|--|--|--|
| SHREL | 250 | 80.E6 |  |  |  |  |  |  |  |
|-------|-----|-------|--|--|--|--|--|--|--|

| Describer | Meaning                                                                            |
|-----------|------------------------------------------------------------------------------------|
| SID       | Unique shear model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| G         | Shear-modulus value. (Real; Default = 0.0)                                         |

**Remark:**

1. SID must unique among all SHRxx entries in one model.

**SHRPOL****Polynomial Shear Model**

Defines an elastic shear model with a polynomial shear modulus. Used in SOL 700 only.

**Format:**

| 1      | 2   | 3     | 4     | 5     | 6     | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|-------|-------|---|---|---|----|
| SHRPOL | SID | $G_0$ | $G_1$ | $G_2$ | $G_3$ |   |   |   |    |

**Example:**

|        |     |       |  |  |  |  |  |  |  |
|--------|-----|-------|--|--|--|--|--|--|--|
| SHRPOL | 250 | 80.E6 |  |  |  |  |  |  |  |
|--------|-----|-------|--|--|--|--|--|--|--|

| Describer | Meaning                                                                            |
|-----------|------------------------------------------------------------------------------------|
| SID       | Unique shear model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| $G_0$     | Coefficient $G_0$ . (Real; Default = 0.0)                                          |
| $G_1$     | Coefficient $G_1$ . (Real; Default = 0.0)                                          |
| $G_2$     | Coefficient $G_2$ . (Real; Default = 0.0)                                          |
| $G_3$     | Coefficient $G_3$ . (Real; Default = 0.0)                                          |

**Remark:**

1. SID must unique among all SHRxx entries in one model.
2. The shear modulus is computed from

$$G = G_0 + G_1\gamma + G_2\gamma^2 + G_3\gamma^3$$

where  $\gamma$  = effective plastic shear strain

and  $G_0$ ,  $G_1$ ,  $G_2$  and  $G_3$  are constants

## SHRUDS

## User-defined Shear Model for Elements.

Specifies that a user subroutine is being used to define the shear modulus. Use in SOL700 only.

### Format:

| 1      | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|---|---|---|---|---|----|
| SHRUDS | SID | GROUP | UNAME |   |   |   |   |   |    |

### Example:

In FMS Section of the MSC Nastran input stream:

```
CONNECT SERVICE mymat 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

In Bulk Data:

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| SHRUDS | 12 | mymat | EXSHR |   |   |   |   |   |    |

| Describer | Meaning                                                                                    |
|-----------|--------------------------------------------------------------------------------------------|
| SID       | Unique porosity model ID. (Integer > 0; Required)                                          |
| GROUP     | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME     | User subroutine name associated with the entry. (Character; default=EXSHR)                 |

### Remarks:

1. Only can be used for SOL 700.
2. The SID must be referenced by a MATDEUL or MAT1 entry.
3. UNAME=EXSHR can only be used.

## SLBDY

## Slot Boundary List

### Format:

| 1     | 2   | 3      | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-----|--------|-----|-----|-----|-----|-----|-----|----|
| SLBDY | RHO | M      | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|       | ID7 | -etc.- |     |     |     |     |     |     |    |

### Example:

|       |       |   |    |    |    |    |    |    |  |
|-------|-------|---|----|----|----|----|----|----|--|
| SLBDY | 0.002 | 6 | 16 | 17 | 18 | 25 | 20 | 21 |  |
|       | 22    |   |    |    |    |    |    |    |  |

| Describer       | Meaning                                                                                                                      |
|-----------------|------------------------------------------------------------------------------------------------------------------------------|
| RHO             | Density of fluid at boundary. (Real > 0.0 or blank)                                                                          |
| M               | Number of slots. (Integer $\geq$ 0 or blank)                                                                                 |
| ID <sub>j</sub> | Identification numbers of GRIDS slot points at boundary with axisymmetric fluid cavity, $j = 1, 2, \dots, J$ . (Integer > 0) |

### Remarks:

1. SLBDY is allowed only if an AXSLOT entry is also present.
2. If RHO or M is blank, the default value on the AXSLOT entry is used. The effective value must not be zero for RHO. If the effective value of M is zero, no matrices at the boundary will be generated.
3. The order of the list of points determines the topology of the boundary. The points are listed sequentially as one travels along the boundary in either direction. At least two points must be defined.



## SLOAD Static Scalar Load

Defines concentrated static loads on scalar or grid points.

### Format:

| 1     | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-------|-----|----|----|----|----|----|----|---|----|
| SLOAD | SID | S1 | F1 | S2 | F2 | S3 | F3 |   |    |

### Example:

|       |    |   |     |    |      |    |       |  |  |
|-------|----|---|-----|----|------|----|-------|--|--|
| SLOAD | 16 | 2 | 5.9 | 17 | -6.3 | 14 | -2.93 |  |  |
|-------|----|---|-----|----|------|----|-------|--|--|

| Describer | Meaning                                                   |
|-----------|-----------------------------------------------------------|
| SID       | Load set identification number. (Integer > 0)             |
| Si        | Scalar or grid point identification number. (Integer > 0) |
| Fi        | Load magnitude. (Real)                                    |

### Remarks:

1. In the static solution sequences, SID is selected by the LOAD Case Control command.
2. In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
3. Up to three loads may be defined on a single entry.
4. If Si refers to a grid point, the load is applied to component T1 of the displacement coordinate system (see the CD field on the GRID entry).

## SLOADN1

Describes TOP/BOT/MID Scalar Load for Heat Shell Element in SOL 400

Defines concentrated static loads on grid points of heat shell elements with linear or quadratic temperature distribution through the thickness direction.

### Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|---------|-----|----|----|----|----|----|----|---|----|
| SLOADN1 | SID | G1 | C1 | Q1 | G2 | C2 | Q2 |   |    |

### Example:

|         |    |    |    |       |    |   |       |  |  |
|---------|----|----|----|-------|----|---|-------|--|--|
| SLOADN1 | 10 | 10 | 12 | 1300. | 20 | 2 | 1300. |  |  |
|---------|----|----|----|-------|----|---|-------|--|--|

| Describer | Meaning                                                                                                                                                                             |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Load set identification number. (Integer > 0)                                                                                                                                       |
| Gi        | Grid point identification number. (Integer > 0)                                                                                                                                     |
| Ci        | Component numbers. ( $0 \leq \text{Integer} \leq 3$ ; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1) |
| Qi        | Power. (Real)                                                                                                                                                                       |

### Remarks:

1. This entry is for shell elements defined on a PSHLN1 heat transfer to specify power input.
2. In the steady-state heat transfer analysis, SID is selected by the LOAD Case Control command.
3. In the transient heat transfer analysis, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.

## SNORM Surface Normal Vector at Grid Point

Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements.

### Format:

|       |     |     |    |    |    |   |   |   |    |
|-------|-----|-----|----|----|----|---|---|---|----|
| 1     | 2   | 3   | 4  | 5  | 6  | 7 | 8 | 9 | 10 |
| SNORM | GID | CID | N1 | N2 | N3 |   |   |   |    |

### Example:

|       |   |   |    |     |    |  |  |  |  |
|-------|---|---|----|-----|----|--|--|--|--|
| SNORM | 3 | 2 | 0. | -1. | 0. |  |  |  |  |
|-------|---|---|----|-----|----|--|--|--|--|

| Describer | Meaning                                                                                                                                                                       |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| GID       | Unique grid point identification number. (Integer > 0)                                                                                                                        |
| CID       | Identification number of coordinate system in which the components of the normal vector are defined. See Remark 3. (Integer ≥ 0; Default = 0 for the basic coordinate system) |
| Ni        | Components of normal vector. The three components of the normal need not define a unit vector. (Real; Default = 0.0)                                                          |

### Remarks:

1. The SNORM Bulk Data entry overrides any unique, internally-generated grid point normals that may have been requested with the user parameter SNORM, described in Chapter 6 of this guide.
2. The normal is used in CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. For all other elements, the normal is ignored.
3. If CID is a cylindrical or spherical coordinate system, the components Ni are in the local tangent system at grid GID. For example, if CID=10 is a spherical coordinate system and normals must be defined pointing outwards in the radial direction of the sphere, see , then the SNORM entries for all grids GID on the sphere are simply  
SNORM, GID, 10, 1., 0., 0.

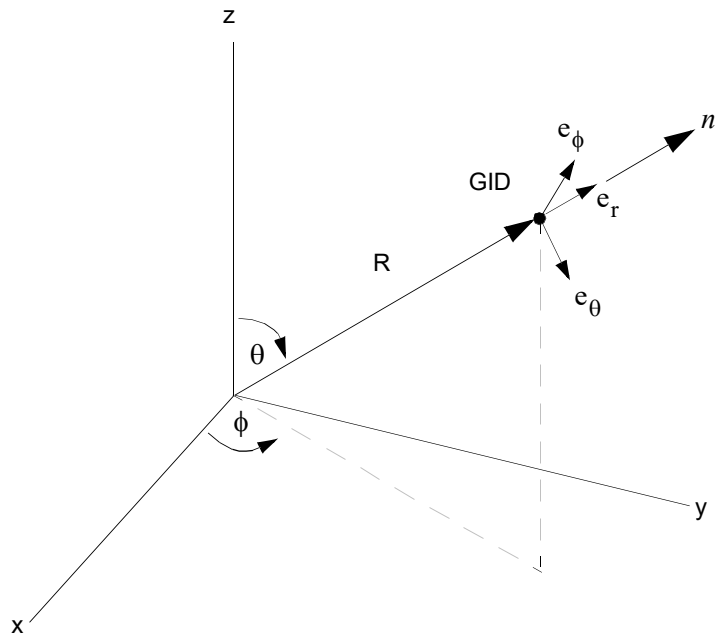


Figure 9-144

## SPBLND1 Strip Based Spline Blending

Defines a strip based blending of two splines.

### Format:

| 1       | 2   | 3    | 4    | 5   | 6  | 7   | 8  | 9  | 10 |
|---------|-----|------|------|-----|----|-----|----|----|----|
| SPBLND1 | SID | SID1 | SID2 | OPT | W1 | GID | D1 | D2 |    |
|         | X1  | X2   | X3   | CID |    |     |    |    |    |

### Example:

|         |     |     |     |     |  |     |      |      |  |
|---------|-----|-----|-----|-----|--|-----|------|------|--|
| SPBLND1 | 130 | 110 | 120 | CUB |  | 227 | 4.05 | 4.05 |  |
|         | 1.0 | 0.0 | 0.0 | 110 |  |     |      |      |  |

| Describer  | Meaning                                                                                                                                                          |
|------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID        | Identification number of blended spline. (Integer > 0)                                                                                                           |
| SID1       | Identification number of first spline (may be a blended spline). (Integer > 0)                                                                                   |
| SID2       | Identification number of second spline (may be a blended spline). (Integer > 0)                                                                                  |
| OPT        | Blending Option:<br>WAVG      Weighted Average (Default)<br>LIN        Linear Blending Functions<br>CUB        Cubic Blending Functions                          |
| W1         | Weight to be used with first spline. ( $0.0 < \text{Real} \leq 1.0$ ; Default = 0.5) (Used only with option WAVG)                                                |
| GID        | Identification number of an aerodynamic grid to be used as reference grid. (Integer>0)                                                                           |
| D1         | Blending Depth of first spline. (Real > 0.0)                                                                                                                     |
| D2         | Blending Depth of second spline. (Real > 0.0)                                                                                                                    |
| X1, X2, X3 | Components of a direction vector $v$ , in the coordinate system defined by CID, which is used to measure the distance from the reference grid. (See Remark 3.)   |
| CID        | Identification number of a rectangular coordinate system used to define the direction vector. (Integer > 0; Default = 0; indicating the basic coordinate system) |

### Remarks:

1. The blending depth defines the maximum distance from the reference grid point of an aerodynamic grid point to be used in blending. It is also needed to evaluate the blending functions.
2. With option WAVG, the definition of GID, D1, D2 and the direction vector is optional. The weight W2 to be used with the second spline is computed from  $W2 = 1 - W1$ .

3. The blended displacement is computed from

$$u_b = f_1(x)u_1 + f_2(x)u_2$$

where  $f_1(x)$  and  $f_2(x)$  are the blending functions (see Figure 9-145) and  $x$  is the distance from the reference grid point, measured in the direction of the direction vector  $\mathbf{v}$ . Functions  $f_1(x)$  and  $f_2(x)$  sum up to 1.

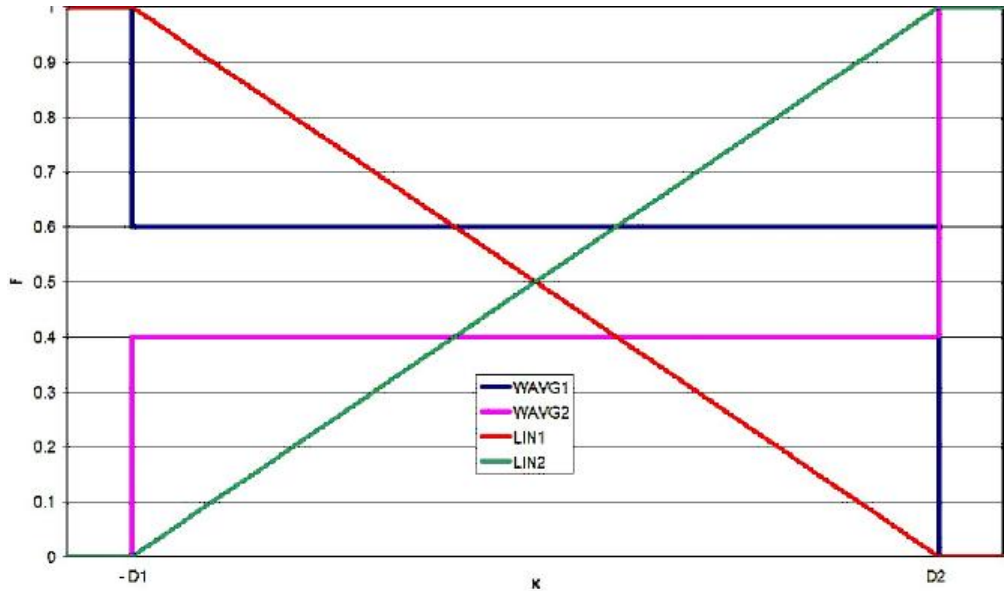


Figure 9-145 Blending Functions

4. If the overlap region extends beyond  $-D1 < x < D2$ , then
- $$f_1(x) = 1.0 \text{ and } f_2(x) = 0 \text{ for } x < -D1$$
- $$f_1(x) = 0 \text{ and } f_2(x) = 1.0 \text{ for } x > D2$$
5. The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
6. If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

## SPBLND2 Curve Based Spline Blending

Defines a curve based blending of two splines.

### Format:

| 1       | 2   | 3    | 4    | 5   | 6      | 7  | 8  | 9 | 10 |
|---------|-----|------|------|-----|--------|----|----|---|----|
| SPBLND2 | SID | SID1 | SID2 | OPT | AELIST | D1 | D2 |   |    |

### Example:

|         |     |     |     |     |   |     |     |  |  |
|---------|-----|-----|-----|-----|---|-----|-----|--|--|
| SPBLND2 | 130 | 110 | 120 | LIN | 4 | 1.5 | 2.5 |  |  |
|---------|-----|-----|-----|-----|---|-----|-----|--|--|

| Describer | Meaning                                                                                                                   |
|-----------|---------------------------------------------------------------------------------------------------------------------------|
| SID       | Identification number of blended spline. (Integer > 0)                                                                    |
| SID1      | Identification number of first spline (may be a blended spline). (Integer > 0)                                            |
| SID2      | Identification number of second spline (may be a blended spline). (Integer > 0)                                           |
| OPT       | Blending Option:<br>LIN      Linear Blending Functions (Default)<br>CUB      Cubic Blending Functions                     |
| AELIST    | Identification number of an AELIST entry listing the aerodynamic grid points that define a reference curve. (Integer > 0) |
| D1        | Blending Depth of first spline. (Real > 0.0)                                                                              |
| D2        | Blending Depth of second spline. (Real > 0.0)                                                                             |

### Remarks:

1. The blending depth defines the maximum value of the distance of an aerodynamic grid point from the reference curve. It is also needed to evaluate the blending functions.
2. Blending functions are evaluated based on the distance of an aerodynamic grid point from the reference curve.
3. The reference curve is approximated by a polygon through the grid points listed on the AELIST entry referenced by AELIST. The list may contain coincident grid points. The order of the grid points is arbitrary.
4. The referenced splines must have the same USAGE flag. This USAGE flag defines the USAGE flag of the blended spline.
5. If the splined aero components are of type CAERO, a MDLPRM,MLTSPLIN,1 entry must be used to enable blending. This is not required with AEGRID based aerodynamics.

**SPC****Single-Point Constraint**

Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis).

**Format:**

| 1   | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|-----|-----|----|----|----|----|----|----|---|----|
| SPC | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

**Example:**

|     |   |    |   |      |   |  |  |  |  |
|-----|---|----|---|------|---|--|--|--|--|
| SPC | 2 | 32 | 3 | -2.6 | 5 |  |  |  |  |
|-----|---|----|---|------|---|--|--|--|--|

| Describer | Meaning                                                                                                                                                                                                                                 |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Identification number of the single-point constraint set. (Integer > 0)                                                                                                                                                                 |
| Gi        | Grid or scalar point identification number. (Integer > 0)                                                                                                                                                                               |
| Ci        | Component number. See Remark 8. ( $0 \leq \text{Integer} \leq 6$ ; up to six Unique Integers, 1 through 6, may be placed in the field with no embedded blanks. 0 or 1 applies to scalar points and 1 through 6 applies to grid points.) |
| Di        | Value of enforced motion for components Gi at grid Ci. (Real; Default = 0.0)                                                                                                                                                            |

**Remarks:**

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
3. Single-point forces of constraint are recovered during stress data recovery.
4. From 1 to 12 degrees-of-freedom may be specified on a single entry.
5. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
6. For reasons of efficiency, the SPCD entry is the preferred method for applying enforced motion rather than the Di field described here.
7. For SOL 400, the SPC entry requests enforced total displacement (Di) while the SPC1 entry requests null enforced relative displacements for a step. See the SPCD and SPCR entries for additional information.



| Enforced Relative Displacement |                                                                       |
|--------------------------------|-----------------------------------------------------------------------|
| SPCR                           | associated with SPC1 or GRID/PS                                       |
| Enforced Total Displacement    |                                                                       |
| SPCD                           | associated with SPC and overwrite the value of enforced motion on SPC |
| SPCD                           | associated with SPC1 or GRID/PS                                       |
| SPC                            | No association required                                               |

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPC entry requests enforced total displacement just like SPCD, however SPC may only be used in SOL 600 to enforce displacements if there is just one subcase in the analysis. If two or more subcases exist, SPCD or SPCR must be used to prescribe the enforced motion.

8. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
9. In thermal analysis, this entry specifies a constant temperature boundary condition applied on the selected grid or scalar point. For SOL 400, transient thermal analysis with any time-varying boundary condition, a constant temperature condition should be specified using the SPCD and SPC1 Bulk Data entries. For SOLs 153 and 159 thermal analysis, no TEMPBC Bulk Data entries with TYPE="STAT" may be specified with this entry.
10. SOL129 transient analysis does not support Di option for enforced motion. Use SOL400.

## SPC1 Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

### Format:

| 1    | 2   | 3  | 4  | 5      | 6  | 7  | 8  | 9  | 10 |
|------|-----|----|----|--------|----|----|----|----|----|
| SPC1 | SID | C  | G1 | G2     | G3 | G4 | G5 | G6 |    |
|      | G7  | G8 | G9 | -etc.- |    |    |    |    |    |

### Example:

|      |   |   |   |   |    |   |   |   |  |
|------|---|---|---|---|----|---|---|---|--|
| SPC1 | 3 | 2 | 1 | 3 | 10 | 9 | 6 | 5 |  |
|      | 2 | 8 |   |   |    |   |   |   |  |

### Alternate Format and Example:

|      |     |       |    |        |    |  |  |  |  |
|------|-----|-------|----|--------|----|--|--|--|--|
| SPC1 | SID | C     | G1 | “THRU” | G2 |  |  |  |  |
| SPC1 | 313 | 12456 | 6  | THRU   | 32 |  |  |  |  |

| Describer | Meaning                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Identification number of single-point constraint set. (Integer > 0)                                                                                                                         |
| C         | Component numbers. See Remark 7. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0, 1 or blank for scalar points.) |
| Gi        | Grid or scalar point identification numbers. (Integer > 0 or “THRU”; For “THRU” option, G1 < G2.)                                                                                           |

### Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. Enforced displacements are available via this entry when used with the recommended SPCD entry.
3. Degrees-of-freedom specified on this entry form members of the mutually exclusive s-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
4. Degrees-of-freedom on this entry may be redundantly specified as permanent constraints using the PS field on the GRID entry.
5. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points that do not exist will collectively produce a warning message but will otherwise be ignored.
6. For SOL 400, the SPC1 entry requests null enforced relative displacement for a step while the SPC entry requests enforced total displacements. SPC1 can always hold the displacement from the previous STEP. See the SPC, SPCD and SPCR entries for additional information.

| Enforced Relative Displacement                                                                                                                                                                                                                                                                                                                         |                         |                 |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-------------------------|-----------------|
| SPCR                                                                                                                                                                                                                                                                                                                                                   | associated              | SPC1 or GRID/PS |
| Enforced Total Displacement                                                                                                                                                                                                                                                                                                                            |                         |                 |
| SPCD                                                                                                                                                                                                                                                                                                                                                   | associated              | SPC             |
| SPCD                                                                                                                                                                                                                                                                                                                                                   | associated              | SPC1 or GRID/PS |
| SPC                                                                                                                                                                                                                                                                                                                                                    | No association required |                 |
| <p>In SOL 400 if a “control” GRID has an applied load associated with it in the previous STEP and if the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is still to use the SPCR, however, the user may apply instead an SPC1 containing the “control” grid ID.</p> |                         |                 |

7. For heat shell element with linear or quadratic nodal distribution (see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID.)
8. In thermal analysis, this entry is used with SPCD to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPC Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE=“TRAN” for SOL 159.

## SPCADD

## Single-Point Constraint Set Combination

Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries.

### Format:

|        |     |    |        |    |    |    |    |    |    |
|--------|-----|----|--------|----|----|----|----|----|----|
| 1      | 2   | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
| SPCADD | SID | S1 | S2     | S3 | S4 | S5 | S6 | S7 |    |
|        | S8  | S9 | -etc.- |    |    |    |    |    |    |

### Example:

|        |     |   |   |   |   |  |  |  |  |
|--------|-----|---|---|---|---|--|--|--|--|
| SPCADD | 101 | 3 | 2 | 9 | 1 |  |  |  |  |
|--------|-----|---|---|---|---|--|--|--|--|

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

|     |                                                                                                          |
|-----|----------------------------------------------------------------------------------------------------------|
| SID | Single-point constraint set identification number. (Integer > 0)                                         |
| Si  | Identification numbers of single-point constraint sets defined via SPC or by SPC1 entries. (Integer > 0) |

### Remarks:

1. Single-point constraint sets must be selected with the Case Control command SPC = SID.
2. No Si may be the identification number of a single-point constraint set defined by another SPCADD entry.
3. The Si values must be unique.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.
5. SPCADD entries take precedence over SPC entries. If both have the same SID, only the SPCADD entry will be used.

## SPCAX Conical Shell Single-Point Constraint

Defines a set of single-point constraints or enforced displacements for conical shell coordinates.

### Format:

| 1     | 2   | 3   | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|-----|-----|---|---|---|---|---|----|
| SPCAX | SID | RID | HID | C | D |   |   |   |    |

### Example:

|       |   |   |   |    |     |  |  |  |  |
|-------|---|---|---|----|-----|--|--|--|--|
| SPCAX | 2 | 3 | 4 | 13 | 6.0 |  |  |  |  |
|-------|---|---|---|----|-----|--|--|--|--|

| Describer | Meaning                                                                                |
|-----------|----------------------------------------------------------------------------------------|
| SID       | Identification number of a single-point constraint set. (Integer > 0)                  |
| RID       | Ring identification number. See RINGAX entry. (Integer ≥ 0)                            |
| HID       | Harmonic identification number. (Integer ≥ 0)                                          |
| C         | Component identification number. (Any unique combination of the Integers 1 through 6.) |
| D         | Enforced displacement value. (Real)                                                    |

### Remarks:

1. SPCAX is allowed only if an AXIC entry is also present.
2. Single-point constraint sets must be selected with the Case Control command SPC = SID.
3. Coordinates appearing on SPCAX entries may not appear on MPCAX, SUPAX, or OMITAX entries.
4. For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

**SPCD**                      Enforced Motion Value

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

**Format:**

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| SPCD | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

**Example:**

|      |     |    |   |      |   |  |     |  |  |
|------|-----|----|---|------|---|--|-----|--|--|
| SPCD | 100 | 32 | 3 | -2.6 | 5 |  | 2.9 |  |  |
|------|-----|----|---|------|---|--|-----|--|--|

| Describer | Meaning                                                                                                                                                                                               |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Set identification number of the SPCD entry. (Integer > 0)                                                                                                                                            |
| Gi        | Grid or scalar point identification number. (Integer > 0)                                                                                                                                             |
| Ci        | Component numbers. See Remark 10. ( $0 \leq \text{Integer} \leq 6$ ; any unique combination of Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points) |
| Di        | Value of enforced motion for components Ci at grid Gi. (Real)                                                                                                                                         |

**Remarks:**

- In the static solution sequences, the set ID of the SPCD entry (SID) is selected by the LOAD Case Control command. For SOL 600, it may also be selected by Li on the Bulk Data entry LOAD.
- In dynamic analysis, the selection of SID is determined by the presence of the LOADSET request in Case Control as follows:
  - There is no LOADSET request in Case Control  
SID is selected by the EXCITEID field of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field
  - There is a LOADSET request in Case Control  
SID is selected by LID in the selected LSEQ entries that correspond to the EXCITEID entry of an RLOAD1, RLOAD2, TLOAD1 or TLOAD2 Bulk Data entry that has enforced motion specified in its TYPE field.
- Enforced Motion is not supported for dynamic subcases in SOL 200 (ANALYSIS=MTRAN,DFREQ or MFREQ).
- A global coordinate (Gi and Ci) referenced on this entry must also be referenced on a SPC or SPC1 Bulk Data entry and selected by the SPC Case Control command. This requirement is optional for SOL 600.

5. Enforced motion may be specified on both SPCD and SPC Bulk Data entries. The SPCD entry is selected according to Remark 2. and the SPC entry is selected by the SPC Case Control command. The Di values from both of the selected SPCD and SPC entries will be combined to form a static (or spatial) loading condition. But if a degree-of-freedom is duplicated on both SPCD and SPC then the value of Di on the selected SPCD entry will override the value of Di on the selected SPC entry. In addition, any degrees-of-freedom from the selected SPC entries and not duplicated will still be included in the loading condition. This is illustrated in the examples below.

- Static analysis: In the example below in SUBCASE 11, D=1.0 at GRID 101 overrides D=1.2 on the SPC entry. But GRID 201 is not specified on the selected SPCD and therefore D=1.2 from the SPC entry is also enforced. Similarly, in subcase 12, D=1.0 is enforced at GRID 201 and D=1.2 at GRID 101.

```

SPC=77
SUBCASE 11
LOAD=701
SUBCASE 12
LOAD=702
BEGIN BULK
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2

```

- Dynamic analysis: In the example below, there are two unique EXCITEIDs specified on TLOAD2 and thusly two static loading conditions are created and identical to the conditions created in the static analysis example above.

```

SPC=77
DLOAD=201
BEGIN BULK
DLOAD,201,1.0,1.0,401,1.0,402
TLOAD2, 401, 701, , DISP, 0., 10.0, 60.,90.
TLOAD2, 402, 702, , DISP, 0., 10.0, 20.,270.
SPCD,701,101,2,1.0
SPCD,702,201,2,1.0
SPC ,77 ,101,2,1.2
SPC ,77 ,201,2,1.2

```

6. The LOAD Bulk Data entry will not combine an SPCD load entry except for SOL 600.
7. In static analysis, this method of applying enforced displacements is more efficient than the SPC entry when more than one enforced displacement condition is applied. It provides equivalent answers.

8. In dynamic analysis, this direct method of specifying enforced motion is more accurate, efficient and elegant than the large mass and Lagrange multiplier techniques.
9. For SOL 400, the SPCD entry requests enforced total displacement for a STEP while the SPCR entry requests enforced relative displacements. SPCD is the position at the end of the applied STEP for ANALYSIS=NLSTAT.

| Enforced Relative Displacement                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                   |                                                                       |
|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|-----------------------------------------------------------------------|
| SPCR                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | associated with SPC1 or GRID/PS                                       |
| Enforced Total Displacement                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |                                                                       |
| SPCD                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | associated with SPC and overwrite the value of enforced motion on SPC |
| SPCD                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             | associated with SPC1 or GRID/PS                                       |
| SPC                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              | No association required                                               |
| <p>In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.</p> <p>For SOL 600, the SPCD entry requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR requests displacement or rotation for the current subcase relative to that of the previous subcase. SPCD may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.</p> |                                                                       |

10. For SOL 400 using heat shell element with linear or quadratic nodal distribution, see option TEMPP for NLMOPTS entry 1=TOP, 2=BOT or 3=MID).
11. In thermal analysis, this entry is used with SPC1 to specify a temperature boundary condition applied on the selected grid or scalar point. To define time-varying boundary conditions, the user should specify SPC1 and SPCD Bulk Data entries for SOL 400 while using TEMPBC Bulk Data entries with TYPE="TRAN" for SOL 159 and SOL 600.
12. If it is desired to enforce motion on the boundary point (a-set) of a superelement then define a high stiffness element attached to the boundary point and apply the SPCD to the opposite point of the high stiffness element. High stiffness elements may be easily defined with CBUSH or CBAR.
13. SOL129 transient analysis does not support Di option for enforced motion. Use SOL400.



**SPCOFF****Excludes Degrees-of-Freedom from the AUTOSPC Operation**

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

**Format:**

| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|--------|----|----|----|----|----|----|----|----|----|
| SPCOFF | G1 | C1 | G2 | C2 | G3 | C3 | G4 | C4 |    |

**Example:**

|        |    |     |   |   |  |  |  |  |  |
|--------|----|-----|---|---|--|--|--|--|--|
| SPCOFF | 32 | 436 | 5 | 1 |  |  |  |  |  |
|--------|----|-----|---|---|--|--|--|--|--|

| Describer | Meaning                                                                                                                     |
|-----------|-----------------------------------------------------------------------------------------------------------------------------|
| Gi        | Grid or scalar point identification number. (Integer > 0)                                                                   |
| Ci        | Component numbers. (Integer 0, 1 or blank for scalar points; Integers 1 through 6 with no embedded blanks for grid points.) |

**Remarks:**

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
2. Degrees-of-freedom that are specified as both SPC and SPCOFF will be considered as SPC.

## SPCOFF1

Excludes DOF's from AUTOSPC Processing, Alternate Form

Defines a set of degrees-of-freedom to be excluded from the AUTOSPC operation. See [Constraint and Mechanism Problem Identification in SubDMAP SEKR](#) in the *MSC Nastran Reference Guide* for a description of the AUTOSPC operation.

### Format:

|         |    |    |        |    |    |    |    |    |    |
|---------|----|----|--------|----|----|----|----|----|----|
| 1       | 2  | 3  | 4      | 5  | 6  | 7  | 8  | 9  | 10 |
| SPCOFF1 | C  | G1 | G2     | G3 | G4 | G5 | G6 | G7 |    |
|         | G8 | G9 | -etc.- |    |    |    |    |    |    |

### Example:

|         |   |   |   |    |   |   |   |   |  |
|---------|---|---|---|----|---|---|---|---|--|
| SPCOFF1 | 2 | 1 | 3 | 10 | 9 | 6 | 5 | 4 |  |
|         | 8 |   |   |    |   |   |   |   |  |

### Alternate Format and Example:

|         |       |    |        |    |  |  |  |  |  |
|---------|-------|----|--------|----|--|--|--|--|--|
| SPCOFF1 | C     | G1 | "THRU" | G2 |  |  |  |  |  |
| SPCOFF1 | 12456 | 6  | THRU   | 32 |  |  |  |  |  |

| Describer | Meaning                                                                                                                                                   |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------------------|
| C         | Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points; Integer 0, 1 or blank for scalar points.) |
| Gi        | Grid or scalar point identification numbers. (Integer > 0 or "THRU"; for "THRU" option, G1 < G2.)                                                         |

### Remarks:

1. Degrees-of-freedom specified on this entry are to be excluded from the AUTOSPC operation. If any degree-of-freedom in this set is found to be singular, a warning message is issued and no constraint is applied.
2. Degrees-of-freedom that are both specified as SPC and SPCOFF will be considered as SPC.
3. If the alternate format is used, points in the sequence G1 through G2 are not required to exist. Points which do not exist will collectively produce a warning message but will otherwise be ignored.

## SPCR Enforced Relative Motion Value

Defines an enforced relative displacement value for a load step in SOL 400 and SOL 600.

### Format:

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| SPCR | SID | G1 | C1 | D1 | G2 | C2 | D2 |   |    |

### Example:

|      |     |    |     |      |   |  |     |  |  |
|------|-----|----|-----|------|---|--|-----|--|--|
| SPCR | 100 | 32 | 436 | -2.6 | 5 |  | 2.9 |  |  |
|------|-----|----|-----|------|---|--|-----|--|--|

| Describer | Meaning                                                                                                                                                                    |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Set identification number of the SPCR entry. (Integer > 0)                                                                                                                 |
| Gi        | Grid or scalar point identification number. (Integer > 0)                                                                                                                  |
| Ci        | Component number. ( $0 \leq \text{Integer} \leq 6$ ; up to six unique Integers may be placed in the field with no embedded blanks, a blank or 0 is treated the same as 1.) |
| Di        | Value of enforced motion for Gi and Ci. (Real)                                                                                                                             |

### Remarks:

1. SPCR requests relative motion for a load step with respect to the previous step in SOL 400 or subcase in SOL 600. SPCR is the companion entry to SPCD. SPCD requests total motion while SPCR requests relative motion. SPCR is the position at activation of the boundary condition.

| Enforced Relative Displacement |                                                                       |
|--------------------------------|-----------------------------------------------------------------------|
| SPCR                           | associated with SPC1 or GRID/PS                                       |
| Enforced Total Displacement    |                                                                       |
| SPCD                           | associated with SPC and overwrite the value of enforced motion on SPC |

### Enforced Relative Displacement

|      |                                 |
|------|---------------------------------|
| SPCD | associated with SPC1 or GRID/PS |
| SPC  | No association required         |

In SOL 400, when a GRID has an applied load associated with it in the previous STEP and the user wishes to pick up the resulting displacement as an enforced displacement in the current STEP, then the recommended procedure is to use the SPCR. Alternatively, the user may apply instead an SPC1 to the GRID to lock in the position of the grid from the previous STEP.

For SOL 600, the SPCR entry requests displacement or rotation for the current subcase relative to that of the previous subcase. The SPCD requests enforced total displacement for the subcase and is the position at the end of the SUBCASE. SPCR may be used for nonlinear statics or dynamics and does not need to be associated with SPC, SPC1 or GRID/PS. In other words, the association can be made but is not required for SOL 600.

2. The enforced motion for a step, for SOL 400 or subcase for SOL 600, can be either total value or relative value. For SOL 400, SPC and SPCD request total enforced motion. SPC1 and SPCR request the relative value. For example, if a DOF is specified on a SPCR with 0.0 for step 2, the relative displacement of this DOF for step 2 with respect to step 1 is 0.0. The total displacement of step 2 is 0.2 if the solution of step 1 for this DOF is 0.2.
3. The SPCD and SPCR entries can have the same SID, but they cannot be specified on the same DOF. A user fatal error will be issued if SPCD and SPCR are specified on the same DOF.
4. In the static solution sequences, the SID of the SPCR entry (SID), same as SPCD, is selected by the LOAD Case Control command.
5. For SOL 400, a global coordinate (Gi and CI) referenced on this entry must also be referenced on a SPC1 Bulk Data entry and selected by the SPC Case Control command. Please note that, for this purpose, SPC cannot be used together with SPCR. If SPC is used, a user fatal error will be issued.
6. For SOL 400, the LOAD Bulk Data entry will not combine an SPCR load entry.
7. For SOL 600 if SPCR is entered, PARAM,MARCTOTT,1 must also be entered.

**SPHERE**

Defines the Shape of a Sphere

Spherical shape used in the initial condition definition on the TICEUL1 entry. Used in SOL 700 only.

**Format:**

|        |     |   |   |   |   |        |   |   |    |
|--------|-----|---|---|---|---|--------|---|---|----|
| 1      | 2   | 3 | 4 | 5 | 6 | 7      | 8 | 9 | 10 |
| SPHERE | VID |   | X | Y | Z | RADIUS |   |   |    |

**Example:**

|        |     |  |    |    |    |    |  |  |  |
|--------|-----|--|----|----|----|----|--|--|--|
| SPHERE | 100 |  | 1. | 1. | 1. | .5 |  |  |  |
|--------|-----|--|----|----|----|----|--|--|--|

| Describer | Meaning                                                        |
|-----------|----------------------------------------------------------------|
| VID       | Number of the sphere. (Integer > 0; Required)                  |
| X, Y, Z   | Coordinates of the center of the sphere. (Real; Default = 0.0) |
| RADIUS    | Radius of the sphere. (Real > 0; Required)                     |

**SPLINE1**

## Surface Spline Methods

Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

**Format:**

| 1       | 2     | 3     | 4    | 5    | 6    | 7  | 8    | 9     | 10 |
|---------|-------|-------|------|------|------|----|------|-------|----|
| SPLINE1 | EID   | CAERO | BOX1 | BOX2 | SETG | DZ | METH | USAGE |    |
|         | NELEM | MELEM |      |      |      |    |      |       |    |

**Example:**

|         |   |     |     |     |    |    |  |  |  |
|---------|---|-----|-----|-----|----|----|--|--|--|
| SPLINE1 | 3 | 111 | 115 | 122 | 14 | 0. |  |  |  |
|---------|---|-----|-----|-----|----|----|--|--|--|

| Describer  | Meaning                                                                                                                                                                                     |
|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID        | Unique spline identification number. (Integer > 0)                                                                                                                                          |
| CAERO      | Aero-element (CAEROi entry ID) that defines the plane of the spline. (Integer > 0)                                                                                                          |
| BOX1, BOX2 | First and last box with motions that are interpolated using this spline; see Remark 3. when using Mach Box method. (Integer > 0; BOX2 ≥ BOX1)                                               |
| SETG       | Refers to the SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                               |
| DZ         | Linear attachment flexibility. (Real ≥ 0.0; Default = 0.0)                                                                                                                                  |
| METH       | Method for the spline fit. IPS, TPS or FPS. See Remark 1. (Character; Default = IPS)                                                                                                        |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 5. (Character; Default = BOTH) |
| NELEM      | The number of FE elements along the local spline x-axis if using the FPS option. (Integer > 0; Default = 10)                                                                                |
| MELEM      | The number of FE elements along the local spline y-axis if using the FPS option. (Integer > 0; Default = 10)                                                                                |

**Remarks:**

1. The default METHOD will result in the use of the Harder-Desmarais Infinite Plate Spline (IPS). The other options are the Thin Plate Spline (TPS) and the Finite Plate Spline (FPS). The continuation applies only to the FPS option and is required only if the defaults are not adequate.
2. The interpolated points (k-set) will be defined by aero boxes. Figure 9-146 shows the cells for which  $u_k$  is interpolated if BOX1 = 115 and BOX2 = 122.

|     |     |     |     |
|-----|-----|-----|-----|
| 111 | 114 | 117 | 120 |
| 112 | 115 | 118 | 121 |
| 113 | 116 | 119 | 122 |

Figure 9-146 SPLINE1 Entry Example

3. The attachment flexibility (units of area) is used for smoothing the interpolation. If  $DZ = 0.0$ , the spline will pass through all deflected grid points. If  $DZ$  is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
4. When using the Mach Box method, BOX1 and BOX2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACCT entry) which will be used for interpolation to structural grids. BOX1 and BOX2 do not refer to Mach Boxes.
5. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$$Fg = [GPkg]^T \{Pk\} \quad (\text{FORCE/BOTH splines are in the transform})$$

$$Uk = [GDkg] \{Ug\} \quad (\text{DISP/BOTH splines are in the transform})$$

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship  $[GPkg]^T = [GDkg]$  satisfied. The default behavior (BOTH for all splines) is compatible with versions of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCEs from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). MSC Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

6. The SPLINE1 EID must be unique with respect to all SPLINEi entries.

**SPLINE2**

## Linear Spline

Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.

**Format:**

| 1       | 2    | 3     | 4   | 5     | 6    | 7  | 8    | 9   | 10 |
|---------|------|-------|-----|-------|------|----|------|-----|----|
| SPLINE2 | EID  | CAERO | ID1 | ID2   | SETG | DZ | DTOR | CID |    |
|         | DTHX | DTHY  |     | USAGE |      |    |      |     |    |

**Example:**

|         |    |   |    |    |    |    |     |   |  |
|---------|----|---|----|----|----|----|-----|---|--|
| SPLINE2 | 5  | 8 | 12 | 24 | 60 | 0. | 1.0 | 3 |  |
|         | 1. |   |    |    |    |    |     |   |  |

| Describer  | Meaning                                                                                                                                                                                                                                                       |
|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID        | Unique spline identification number. (Integer > 0)                                                                                                                                                                                                            |
| CAERO      | Aero panel or body (CAEROi entry ID) that is to be interpolated. (Integer > 0)                                                                                                                                                                                |
| ID1, ID2   | First and last box or body element whose motions are interpolated using this spline. See Remark 6. when using the Mach Box method. (Integer > 0; ID2 ≥ ID1)                                                                                                   |
| SETG       | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                                                                                                  |
| DZ         | Linear attachment flexibility. (Real ≥ 0.0; Default = 0.0)                                                                                                                                                                                                    |
| DTOR       | Torsional flexibility ratio ( $EI/GJ$ ). (Real > 0.0; Default = 1.0; use 1.0 for bodies.)                                                                                                                                                                     |
| CID        | Rectangular coordinate system for which the y-axis defines the axis of the spline. Not used for bodies, CAERO2. (Integer ≥ 0)                                                                                                                                 |
| DTHX, DTHY | Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (in-plane bending rotations); however, it is not used for bodies. DTHY is for rotation about the spline's y-axis (torsion); however, it is used for slope of bodies. (Real) |
| USAGE      | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 9. (Character; Default = BOTH)                                                                   |

**Remarks:**

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the spline axis is parallel to the x-axis of the aerodynamic coordinate system.



3. The flexibilities DZ, DTHX, and DTHY are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing, whereas negative values of DTHX or DTHY will imply infinity, therefore, no attachment). See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE2 EID must be unique with respect to all SPLINEi entries.
6. When using the Mach Box method, ID1 and ID2 refer to the ID number of the first and last aerodynamic grids (x,y pairs on the AEFACCT entry) which will be used for interpolation to the structural grids. ID1 and ID2 do not refer to Mach Boxes.
7. DTOR is the ratio of rotational to linear deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur.
8. If a SPLINE2 element only references one grid point, the job will fail without a message in the GI Module.
9. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

## SPLINE3 Aeroelastic Constraint Equation

Defines a constraint equation for aeroelastic problems. Useful for control surface constraints.

### Format:

| 1       | 2   | 3     | 4     | 5    | 6     | 7  | 8  | 9     | 10 |
|---------|-----|-------|-------|------|-------|----|----|-------|----|
| SPLINE3 | EID | CAERO | BOXID | COMP | G1    | C1 | A1 | USAGE |    |
|         | G2  | C2    | A2    |      | -etc. |    |    |       |    |

### Example:

|         |      |     |      |   |   |   |     |  |  |
|---------|------|-----|------|---|---|---|-----|--|--|
| SPLINE3 | 7000 | 107 | 109  | 6 | 5 | 3 | 1.0 |  |  |
|         | 43   | 5   | -1.0 |   |   |   |     |  |  |

| Describer | Meaning                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID       | Unique spline identification number. (Integer > 0)                                                                                                                                          |
| CAERO     | Identification number of the macro-element on which the element to be interpolated lies. (Integer > 0)                                                                                      |
| BOXID     | Identification number of the aerodynamic element; i.e., the box number. (Integer > 0)                                                                                                       |
| COMP      | The component of motion to be interpolated. See Remark 4. (One of the Integers 1, 2, 3, 4, 5, or 6.)                                                                                        |
| Gi        | Grid point identification number of the independent grid point. (Integer > 0)                                                                                                               |
| Ci        | Component numbers in the displacement coordinate system. (One of the Integers 1 through 6 for grid points, or 0 for scalar points.)                                                         |
| Ai        | Coefficient of the constraint relationship. (Real)                                                                                                                                          |
| USAGE     | Spline uage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 6. (Character; Default = BOTH). |

### Remarks:

1. The independent grid points and components must refer to degrees-of-freedom in the g-set.
2. The constraint is given by

$$u_d = \sum A_i u_i$$

where:

$u_d$  = value of the dependent component of the aerodynamic element

$u_i$  = displacement at grid Gi, component Ci.

3. The SPLINE3 EID must be unique with respect to all SPLINEi entries.

4. The allowable components by CAEROi entry type are indicated by an “X” in the table below:

| Entry Type  | COMP |   |   |   |   |
|-------------|------|---|---|---|---|
|             | 1    | 2 | 3 | 5 | 6 |
| CAERO1      |      |   | X | X |   |
| CAERO2      |      | X | X | X | X |
| CAERO3      |      |   | X |   |   |
| CAERO4      |      |   | X | X | X |
| CAERO5      |      |   | X | X | X |
| 3D Geometry | X    | X | X | X | X |

COMP = 2: lateral displacement

COMP = 3 transverse displacement

COMP = 5: pitch angle

COMP = 6: relative control angle for CAERO4 and CAERO5 yaw angle for CAERO2.

For general 3D aerodynamic geometries the components numbers refer to axes of the Aerodynamic Coordinate System  $(u_x, u_y, u_z, \theta_x, \theta_y, \theta_z)$ .

5. For Strip theory and Piston theory, the COMP = 6 control surface relative angle is positive when the trailing edge has a negative z-deflection in the element coordinate system (see the [MSC Nastran Aeroelastic Analysis User's Guide](#)).
6. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.

## SPLINE4 Surface Spline Methods

Defines a curved surface spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

### Format:

| 1       | 2     | 3     | 4      | 5     | 6    | 7  | 8    | 9     | 10 |
|---------|-------|-------|--------|-------|------|----|------|-------|----|
| SPLINE4 | EID   | CAERO | AELIST |       | SETG | DZ | METH | USAGE |    |
|         | NELEM | MELEM | FTYPE  | RCORE |      |    |      |       |    |

### Example:

|         |   |     |     |  |    |    |     |  |  |
|---------|---|-----|-----|--|----|----|-----|--|--|
| SPLINE4 | 3 | 111 | 115 |  | 14 | 0. | IPS |  |  |
|---------|---|-----|-----|--|----|----|-----|--|--|

| Describer | Meaning                                                                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Unique spline identification number. (Integer > 0)                                                                                                                                                           |
| CAERO     | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)                                                                                                         |
| AELIST    | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)                                                             |
| SETG      | Identification number of a SET1 entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                                   |
| DZ        | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)                                                                                                                                             |
| METH      | Spline method:<br>IPS      Infinite Plate Spline (Default)<br>TPS      Thin Plate Spline<br>FPS      Finite Plate Spline<br>RIS      Radial Interpolation Spline                                             |
| USAGE     | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. See Remark 3. (Character, Default = BOTH) |
| NELEM     | The number of FE elements along the local x-axis if using the FPS option. (Integer > 0; Default = 10)                                                                                                        |
| MELEM     | The number of FE elements along the local y-axis if using the FPS option. (Integer > 0; Default = 10)                                                                                                        |
| FTYPE     | Selects the radial interpolation function to be used with the RIS option:<br>WF0      C0 continuous Wendland function                                                                                        |

| Describer | Meaning                                                                      |
|-----------|------------------------------------------------------------------------------|
|           | WF2      C2 continuous Wendland function (Default)                           |
| RCORE     | Radius of support of radial interpolation function. (Real > 0.0; no Default) |

**Remarks:**

1. The attachment flexibility (units of area) is used for smoothing the interpolation. If DZ = 0.0, the spline will pass through all deflected grid points. If DZ is much greater than the spline area, a least squares plane fit will be applied. Intermediate values will provide smoothing.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACCT entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. NELEM and MELEM are used only for the METH=FPS and are required only when the defaults are not adequate.
5. FTYPE and RCORE used only with METH=RIS. FTYPE=WF0 uses a Wendland function:

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^2$$

while FTYPE = WF2 uses

$$\phi\left(\frac{r}{r_c}\right) = \left(1 - \frac{r}{r_c}\right)_t^4 \left(4\frac{r}{r_c} + 1\right)$$

where

$$(y)_t = \begin{cases} y & \text{if } y > 0 \\ 0 & \text{if } y < 0 \end{cases}$$

**SPLINE5**

## Linear Spline

Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries.

**Format:**

| 1       | 2    | 3     | 4      | 5     | 6    | 7  | 8     | 9     | 10 |
|---------|------|-------|--------|-------|------|----|-------|-------|----|
| SPLINE5 | SID  | CAERO | AELIST |       | SETG | DZ | DTOR  | CID   |    |
|         | DTHX | DTHY  |        | USAGE | METH |    | FTYPE | RCORE |    |

**Example:**

|         |    |   |    |      |    |  |  |   |  |
|---------|----|---|----|------|----|--|--|---|--|
| SPLINE5 | 5  | 8 | 12 |      | 60 |  |  | 3 |  |
|         | 1. |   |    | BOTH |    |  |  |   |  |

| Describer     | Meaning                                                                                                                                                                                                                |
|---------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID           | Unique spline identification number (Integer > 0)                                                                                                                                                                      |
| CAERO         | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0)                                                                                                                   |
| AELIST        | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 6. (Integer > 0)                                                                       |
| SETG          | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                                                           |
| DZ            | Linear attachment flexibility. (Real $\geq$ 0.0; Default = 0.0)                                                                                                                                                        |
| DTOR          | Torsional flexibility ratio (EI/GJ) for the bending in the zy-plane. This value is ignored for slender bodies since they have no torsion; see Remark 7. (Real > 0.0; Default = 1.0; ignored for CAERO2 bodies.)        |
| CID           | Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer $\geq$ 0)                                                             |
| DTHX,<br>DTHY | Rotational attachment flexibility. DTHX is for rotation about the spline's x-axis (the bending rotations). DTHY is for rotation about the spline's y-axis (torsion); however, it is used for bending of bodies. (Real) |
| USAGE         | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 8. (Character; Default = BOTH)                            |
| METH          | Spline method:<br>BEAM          Beam Spline (Default)<br>RIS            Radial Interpolation Spline                                                                                                                    |
| FTYPE         | Selects the radial interpolation function to be used with the RIS option:                                                                                                                                              |

| Describer | Meaning                                                                      |
|-----------|------------------------------------------------------------------------------|
|           | WF0            C0 continuous Wendland function                               |
|           | WF2            C2 continuous Wendland function. (Default)                    |
| RCORE     | Radius of support of radial interpolation function. (Real > 0.0; no Default) |

**Remarks:**

1. The interpolated points (k-set) will be defined by aero boxes.
2. The spline axis for panels is the projection of the y-axis of coordinate system CID, projected onto the plane of the panel. For bodies, the interpolating beam (y-axis) is parallel to the x-axis of the aerodynamic coordinate system; the z-axis is taken from the referenced CID and x is made orthogonal.
3. The flexibilities DZ, DTHX and DTHY are used for smoothing. (Zero attachment flexibility values imply rigid attachment; i.e., no smoothing, whereas negative values of DTHX or DTHY imply infinity, therefore, no attachment.) See the [MSC Nastran Aeroelastic Analysis User's Guide](#) for a discussion of special cases.
4. The continuation entry is required.
5. The SPLINE5 EID must be unique with respect to all SPLINEi entries.
6. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFAC entry). For all other aero methods, the AELIST items are box id's.
7. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily linear deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
8. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
9. FTYPE and RCORE are only used for METHOD=RIS. See Remark 5. on the SPLINE 4 entry for descriptions of the Wendland functions.

## SPLINE6

### 3D Finite Surface Spline

Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.

#### Format:

| 1       | 2      | 3      | 4      | 5      | 6      | 7   | 8       | 9     | 10 |
|---------|--------|--------|--------|--------|--------|-----|---------|-------|----|
| SPLINE6 | EID    | CAERO  | AELIST |        | SETG   | DZ  | METHOD  | USAGE |    |
|         | VSTYPE | VSLIST | I2VNUM | D2VNUM | METHVS | DZR | METHCON | NGRID |    |
|         | ELTOL  | NCYCLE | AUGWEI |        |        |     |         |       |    |

#### Example:

|         |      |   |    |   |     |  |      |      |  |
|---------|------|---|----|---|-----|--|------|------|--|
| SPLINE6 | 5    | 8 | 12 |   | 60  |  | FPS6 | DISP |  |
|         | AERO | 2 | 4  | 4 | VS6 |  |      |      |  |

| Describer | Meaning                                                                                                                                                                                                                                                         |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID       | Unique spline identification number. (Integer > 0)                                                                                                                                                                                                              |
| CAERO     | Aero panel (CAEROi entry ID) that is to be interpolated. See Remarks 2. and 4. (Integer > 0 or blank)                                                                                                                                                           |
| AELIST    | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)                                                                                                                |
| SETG      | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                                                                                                    |
| DZ        | Linear attachment flexibility. (Real $\geq 0.0$ ; Default = 0.0)                                                                                                                                                                                                |
| METHOD    | Method for the spline fit. Either FPS3 or FPS6. See Remark 5. (Character; Default = FPS6)                                                                                                                                                                       |
| USAGE     | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 3. (Character; Default = BOTH)                                                                     |
| VSTYPE    | Virtual surface connectivity type. Either AERO or STRUC. (Character; Default = AERO)                                                                                                                                                                            |
| VSLIST    | Identification number of an AELIST entry listing quadrilateral and/or triangular shell elements of the VSTYPE mesh which define the connectivity of the virtual surface mesh. See Remark 2. (Integer > 0 or can be blank if CAERO is not blank and VSTYPE=AERO) |
| I2VNUM    | The minimum number of structural mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)                                                                                                                        |
| D2VNUM    | The minimum number of aero mesh points to connect to each virtual mesh point. See Remarks 7., 8. and 9. (0 < Integer; Default = 3)                                                                                                                              |



| Describer | Meaning                                                                                                                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| METHVS    | Similar to METHOD, this field chooses whether or not to include the rotational degrees-of-freedom of virtual surface. Either VS6 or VS3. See Remark 5. (Character, Default VS6)                        |
| DZR       | Rotational attachment flexibility. (Real $\geq$ 0.0; Default = 0.0)                                                                                                                                    |
| METHCON   | Method used to determine RBE3 connecting points between the meshes. Either NODEPROX or CIRCBIAS. See Remarks 7., 8. and 9. (Character; Default = NODEPROX)                                             |
| NGRID     | Number of closest grids that are used to determine the element list that is used to define the RBE3 elements. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Integer > 0; Default = 1)        |
| ELTOL     | Tolerance used to determine whether or not a node projects onto an element of the mesh. Specified as % of element size. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real; Default = 100.0) |
| NCYCLE    | Maximum number of cycles used to find elements onto which the nodes project. Only valid for METHCON=CIRCBIAS. See Remark 7. and 9. (Integer > 0; Default = 3)                                          |
| AUGWEI    | RBE3 weighting factor augmentation parameter. Only valid for METHCON=CIRCBIAS. See Remarks 7. and 9. (Real $\geq$ 0.0; Default = 0.0)                                                                  |

**Remarks:**

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating surface and the connections to the dependent and independent grids.
2. If an aerodynamic mesh is entered using AEGRID/AETRIA3/AEQUAD4 entries, the CAERO ID is not required and the AELIST refers to the element ID's of the aero mesh. In this scenario, if VSTYPE=AERO, VSLIST must point to an AELIST that defines the virtual surface mesh by identifying the aerodynamic elements that make up the surface. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFAC entry). For all other aero methods, the AELIST items are box id's.
3. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
4. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
5. The METHOD option provides a choice in using all 6 degrees of freedom (FPS6) on the independent points or only the translational degrees of freedom (FPS3) in connecting between the virtual surface and the independent points. Similarly, there is a choice in connecting the virtual surface to the dependent points (METHVS).

6. The connection between the independent points (structural) and the dependent points (aero) is made through a virtual surface whose mesh is defined by elements listed in the VSLIST (these are either AERO box id's or STRUCtural shell elements (CQUAD4, CQUADR, CTRIA3 ,CTRIAR).
7. To bind the points to the virtual surface, a connection is made between the points and the surface using automatically generated virtual RBE3 elements. Two methods exist to choose which independent mesh points are connected to each dependent mesh point: nodal-proximity (NODEPROX) and circular bias (CIRCBIAS).
8. The nodal proximity method selects the closest independent mesh points to each dependent mesh point. The actual number of points will depend on the user inputs I2VNUM and D2VNUM as well as collinearity checks. Larger values will spread the connectivity (smearing). Smaller values allow for more concentration (with additional points added as necessary for collinearity).
9. The circular bias method uses elements of the virtual mesh in an attempt to select independent mesh points that encircle each dependent mesh point. This method will be restricted to the case where the virtual mesh is the target mesh.

This method will do the following:

- For each splined dependent mesh node, find the closest NGRID splined independent mesh node(s).
- Assemble the list of virtual mesh elements that use the closest node(s).
- Check each of these elements to see if the dependent node projects onto the element in the element's mean plane normal direction. Note that this check may be computationally expensive, so it is performed only to the "possible" elements, not the entire virtual mesh. The projection check will contain a user-defined tolerance, ELTOL, to expand the area of the element that is acceptable for a match.
- If the dependent node does not project onto any element, use the candidate element's nodes to expand the list of elements to check. Repeat the projection check (the original elements will not be rechecked). Repeat this process up to NCYCLE times.
- All elements that are found to encompass the dependent node (and there may be more than one due to curvature) will be selected to move forward.
- Assemble the list of all splined nodes that connect the selected elements.
- Generate RBE3 elements based on this node list. An optional user-defined input parameter, AUGWEI, will be used to augment the RBE3 weighting factors with the following formula:  
$$\text{weight} = (\text{NE} - 1) \cdot \text{AUGWEI} + 1$$

where NE is the number of elements that are connected with the RBE3 node.

## SPLINE7 Finite Beam Spline

Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.

### Format:

| 1       | 2   | 3     | 4      | 5     | 6      | 7   | 8    | 9     | 10 |
|---------|-----|-------|--------|-------|--------|-----|------|-------|----|
| SPLINE7 | EID | CAERO | AELIST |       | SETG   | DZ  | DTOR | CID   |    |
|         |     |       |        | USAGE | METHOD | DZR | IA2  | EPSBM |    |

### Example:

|         |   |   |    |      |      |  |  |   |  |
|---------|---|---|----|------|------|--|--|---|--|
| SPLINE7 | 5 | 8 | 12 |      | 60   |  |  | 3 |  |
|         |   |   |    | BOTH | FBS6 |  |  |   |  |

| Describer | Meaning                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID       | Unique spline identification number. (Integer > 0)                                                                                                                                          |
| CAERO     | Aero panel (CAEROi entry ID) that is to be interpolated. See Remark 6. (Integer > 0 or blank)                                                                                               |
| AELIST    | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 2. (Integer > 0)                                            |
| SETG      | Refers to an SETi entry that lists the structural grid points to which the spline is attached. (Integer > 0)                                                                                |
| DZ        | Linear attachment flexibility. (Real $\geq$ 0.0; Default = 0.0)                                                                                                                             |
| DTOR      | Ratio of the beam bending stiffness to the beam torsional stiffness. See Remark 3.. (Real > 0.0; Default = 1.0)                                                                             |
| USAGE     | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 4. (Character, Default = BOTH) |
| CID       | Rectangular coordinate system that defines the y-axis of the spline and the xy- and yz-planes for bending. Not used for bodies, CAERO2. (Integer $\geq$ 0)                                  |
| METHOD    | Method for the spline fit. Either FBS3 or FBS6. See Remark 5. (Character; Default = FBS6)                                                                                                   |
| DZR       | Rotational attachment flexibility. (Real $\geq$ 0.0; Default = 0.0)                                                                                                                         |
| IA2       | Ratio of the beam bending stiffness to the beam extensional stiffness. (Real > 0.0; Default = 1.0)                                                                                          |
| EPSBM     | Ratio of the minimum beam length to the total beam length. See Remark 7. (Real > 0.0; Default = 0.01)                                                                                       |

**Remarks:**

1. The flexibilities DZ and DZR are used for smoothing. (Zero attachment flexibility values will imply rigid attachment (i.e., no smoothing). The DZ and DZR values are used to derive stiffness of the translational and rotational (respectively) bushing stiffnesses. Bushing elements are placed between the interpolating beam and the connections to the dependent and independent grids.
2. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFAC entry). For all other aero methods, the AELIST items are box id's.
3. DTOR is the ratio of axial rotational to bending deflection and, in lieu of a more accurate estimate, a value of 1.0 is recommended. A different value may be used; e.g., if DTOR is much greater than 1.0, primarily rotational deflection will occur; if DTOR is much less than 1.0, primarily translational deflection will occur. The values will affect the results only if the structural grids over constrain the motion of the interpolating beam. Slender bodies have no torsional motion, so these values will not be used for CAERO2 entries.
4. The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH). See Remark 5. of the SPLINE1 Bulk Data entry.
5. The FBS3 method will map only the three translational degrees of freedom. The FBS6 method will map all six degrees of freedom.
6. CAERO2 entries are not supported. The CAERO entry may be blank unless points from a CAERO4 or a CAERO5 are specified. The list of splined points may span multiple aerodynamic components. If the spline defines points from multiple CAERO4 or CAERO5 entries, then any one of the referenced CAERO IDs is valid input.
7. EPSBM is the minimum length of a SPLINE7 beam as a fraction of the total SPLINE7 FEM length. Thus, an EPSBM value of 0.01 (Default), would produce a minimum beam length that is 1% of the total FEM length. Defining a value of EPSBM that is smaller than 0.01 will decrease the minimum length and potentially increase the number of beams used to define the SPLINE7 FEM.

## SPLINEX

## Externally-Evaluated Spline

Defines the input for a spline that will be evaluated with a user-supplied procedure.

### Format:

| 1       | 2      | 3      | 4       | 5      | 6      | 7      | 8 | 9     | 10 |
|---------|--------|--------|---------|--------|--------|--------|---|-------|----|
| SPLINEX | EID    | GROUP  | DGCOMP  | IGCOMP | DECOMP | IECOMP |   | USAGE |    |
|         | AELIST | AEFACT | AELISTC |        |        |        |   |       |    |

### Example:

|         |     |          |        |        |  |  |  |      |  |
|---------|-----|----------|--------|--------|--|--|--|------|--|
| SPLINEX | 3   | SPLNGRP4 | GWNG1A | GWNG1S |  |  |  | BOTH |  |
|         | 101 | 201      | 301    |        |  |  |  |      |  |

### Describer

### Meaning

|         |                                                                                                                                                                                             |
|---------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EID     | Element identification number. (Integer > 0)                                                                                                                                                |
| GROUP   | Group name to which the external spline type belongs. (Character; no Default)                                                                                                               |
| DGCOMP  | The name of an AECOMP or AECOMPL entry that defines the set of points for the dependent mesh. See Remarks 3. and 4. (Character; Default = Blank).                                           |
| IGCOMP  | The name of an AECOMP or AECOMPL entry that defines the set of points for the independent mesh. See Remarks 3. and 4. (Character; Default = Blank).                                         |
| DECOMP  | The name of an AECOMP or AECOMPL entry that defines the set of elements for the dependent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).                                     |
| IECOMP  | The name of an AECOMP or AECOMPL entry that defines the set of elements for the independent mesh. See Remarks 3., 4. and 5. (Character; Default = Blank).                                   |
| USAGE   | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. FORCE, DISP or BOTH. See Remark 2. (Character; Default = BOTH) |
| AELIST  | ID of an AELIST that contains a list of user-defined integer data. See Remark 6. (Integer; no Default).                                                                                     |
| AEFACT  | ID of an AEFACT that contains a list of user-defined real data. See Remark 6. (Integer; no Default).                                                                                        |
| AELISTC | ID of an AELISTC that contains a list of user-defined character data. See Remark 6. (Integer; no Default).                                                                                  |

### Remarks:

1. The SPLINEX EID must be unique with respect to all SPLINEi entries.

- The USAGE field allows you to specify that the particular spline interpolant defined in this entry is to be used for just the force transformation (FORCE) or just the displacement transformation (DISP) or for both (BOTH).

$$Fg = [GPkg]^T \{Pk\} \quad (\text{FORCE/BOTH splines are in the transform})$$

$$Uk = [GDkg] \{Ug\} \quad (\text{DISP/BOTH splines are in the transform})$$

In general, the two transforms are done with distinct matrices. Only when ALL splines are of type BOTH is the familiar transpose relationship  $[GPkg]^T = [GDkg]$  satisfied. The default behavior (BOTH for all splines) is compatible with version of MSC Nastran prior to Version 70.5.

In general, the USAGE field can be used to apply aerodynamic forces to the structure from aerodynamic panels that are intended NOT to move (USAGE=FORCE) or to apply structural displacements to aerodynamic grids whose forces are not to be applied to the structure (USAGE=DISP). The DISP option is somewhat esoteric in that you are then suggesting that the aeroelastic effect of the surface is important while its forces are not. (In other words, only the forces arising from its effects on other surfaces is important.) While there may be circumstances where this is true, it is unlikely. Take care that you included all the FORCES from aerodynamic panels that are important by including them in either FORCE or BOTH spline(s). Nastran will NOT issue a warning unless ALL forces are omitted. All displacements may be omitted without warning (and is a means to perform “rigid aerodynamic” analyses).

- Typically, for aero-to-structure splines “dependent” means “aerodynamic”, and “independent” means “structural”.
- If the component defines a structural mesh, then the grid component may be left blank and the list of grids will be obtained from the element component member’s connectivity. Both may not be left blank.
- Structural elements referenced by DECOMP and IECOMP are limited to the following element types: CQUAD4, CQUADR, CTRIA3, CTRIAR. In one list, elements from the different types may not share the same ID.
- The data that are defined on the AELIST, AEFACT, and AELISTC have no meaning to Nastran. These lists are generic containers for data that has meaning to the spline server. Note that the AELIST is limited to numbers greater than zero.

## SPLINRB Rigid Body Spline

Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

### Format:

|         |     |       |        |       |    |    |    |    |    |
|---------|-----|-------|--------|-------|----|----|----|----|----|
| 1       | 2   | 3     | 4      | 5     | 6  | 7  | 8  | 9  | 10 |
| SPLINRB | SID | CAERO | AELIST | USAGE | G1 | C1 | G2 | C2 |    |
|         | G3  | C3    | G4     | C4    | G5 | C5 | G6 | C6 |    |

### Example:

|         |     |    |   |      |      |        |  |  |  |
|---------|-----|----|---|------|------|--------|--|--|--|
| SPLINRB | 110 | 20 | 2 | BOTH | 1093 | 123456 |  |  |  |
|---------|-----|----|---|------|------|--------|--|--|--|

| Describer | Meaning                                                                                                                                                                                        |
|-----------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Unique spline identification number. (Integer > 0)                                                                                                                                             |
| CAERO     | Identification number of aerodynamic component that defines the interpolation surface. (Integer > 0 or blank)                                                                                  |
| AELIST    | Identification of an AELIST entry listing the boxes or aerodynamic grid points to be interpolated using this spline. See Remark 3. (Integer > 0)                                               |
| USAGE     | Spline usage flag to determine whether this spline applies to the force transformation, displacement transformation or both. Legal values are FORCE, DISP or BOTH. (Character; Default = BOTH) |
| Gi        | Identification number of a structural grid point. (Integer > 0)                                                                                                                                |
| Ci        | Component numbers: Any unique combination of the Integers 1 through 6 with no embedded blanks.                                                                                                 |

### Remarks:

1. Up to six structural grid points can be used to select exactly 6 structural degrees-of-freedom that define the motion of the rigid body.
2. The selected degrees-of-freedom must define statically determinate supports of the rigid body.
3. For aerodynamic meshes input using AEGRID/AEQUAD4/AETRIA3 entries, the AELIST items are AEGRIDS. For the Mach Box method, the AELIST refers to the aerodynamic grids (x,y pairs on the AEFACT entry). For all other aero methods, the AELIST items are box id's.

## SPOINT

### Scalar Point Definition

Defines scalar points.

#### Format:

|        |     |     |     |     |     |     |     |     |    |
|--------|-----|-----|-----|-----|-----|-----|-----|-----|----|
| 1      | 2   | 3   | 4   | 5   | 6   | 7   | 8   | 9   | 10 |
| SPOINT | ID1 | ID2 | ID3 | ID4 | ID5 | ID6 | ID7 | ID8 |    |

#### Example:

|        |   |    |   |   |    |   |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|
| SPOINT | 3 | 18 | 1 | 4 | 16 | 2 |  |  |  |
|--------|---|----|---|---|----|---|--|--|--|

#### Alternate Format and Example:

|        |     |        |     |  |  |  |  |  |  |
|--------|-----|--------|-----|--|--|--|--|--|--|
| SPOINT | ID1 | “THRU” | ID2 |  |  |  |  |  |  |
| SPOINT | 5   | THRU   | 649 |  |  |  |  |  |  |

| Describer | Meaning                                                                                       |
|-----------|-----------------------------------------------------------------------------------------------|
| IDi       | Scalar point identification number. (0 < Integer < 100,000,000; For “THRU” option, ID1 < ID2) |

#### Remarks:

1. A scalar point defined by its appearance on the connection entry for a scalar element (see the CELASi, CMASSi, and CDAMPi entries) need not appear on an SPOINT entry.
2. All scalar point identification numbers must be unique with respect to all other structural, scalar, fluid and extra (EPOINT) points. However, duplicate scalar point identification numbers are allowed in the input.
3. This entry is used primarily to define scalar points appearing in single-point or multipoint constraint equations to which no scalar elements are connected.
4. If the alternate format is used, all scalar points ID1 through ID2 are defined.
5. For a discussion of scalar points, see [Scalar Elements \(CELASi, CMASSi, CDAMPi\)](#) in the *MSC Nastran Reference Guide*.



## SPRELAX Spline Relaxation

Defines relaxation of a spline based on an adjacent spline.

### Format:

| 1       | 2    | 3    | 4     | 5    | 6     | 7 | 8 | 9 | 10 |
|---------|------|------|-------|------|-------|---|---|---|----|
| SPRELAX | SID1 | SID2 | LIST2 | DREF | LIST1 |   |   |   |    |

### Example:

|         |     |     |    |     |  |  |  |  |  |
|---------|-----|-----|----|-----|--|--|--|--|--|
| SPRELAX | 140 | 130 | 50 | 5.0 |  |  |  |  |  |
|---------|-----|-----|----|-----|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID1      | Identification number of spline to be modified (may be a blended spline and need not be unique). (Integer > 0)                                               |
| SID2      | Identification number of adjacent spline (may be a blended spline). (Integer > 0)                                                                            |
| LIST2     | Identification of an AELIST entry listing aerodynamic grid points that define the curve used in the relaxation . (Integer > 0) (See Remark 1.)               |
| DREF      | Reference Distance (Real > 0.) (See Remark 2.)                                                                                                               |
| LIST1     | Identification number of an AELIST entry listing aerodynamic grid points of the spline to be modified (optional). (Integer > 0; Default = 0) (See Remark 3.) |

### Remarks:

1. The aerodynamic grid points referenced by LIST2 have to define a curve. The curve need not be contiguous, i.e., coincident grid points are allowed. The order of the grid points is arbitrary.
2. Displacements of spline 1 are modified according to

$$u_{1mod} = u_1 + f\left(\frac{r}{DREF}\right)(u_2 - u_1)$$

where  $r$  is the shortest distance of the aerodynamic grid point considered from the curve,  $u_2$  is the displacement from spline 2, interpolated to the position on the curve which is closest to the aerodynamic grid point considered, and function  $f$  is defined by

$$f(x) = \begin{cases} 1 - x & \text{if } x \leq 1 \\ 0 & \text{if } x > 1 \end{cases}$$

3. If LIST1 is defined, only aerodynamic grid points contained in the referenced list are processed. Otherwise, all aerodynamic grid points of spline 1 are processed.

## STOCHAS

## Randomization of Model Parameters

Specifies statistics used in randomization selected model parameters.

### Format:

| 1       | 2   | 3      | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|---------|-----|--------|-----|-----|---|---|---|---|----|
| STOCHAS | SID | PENTRY | CDF | CoV | m |   |   |   |    |
|         |     | MENTRY | CDF | CoV | m |   |   |   |    |
|         |     | CENTRY | CDF | CoV | m |   |   |   |    |
|         |     | LOADS  | CDF | CoV | m |   |   |   |    |
|         |     | SPCD   | CDF | CoV | m |   |   |   |    |

### Example 1: (Randomize all element and material properties with the default settings.)

|         |     |        |  |  |  |  |  |  |  |
|---------|-----|--------|--|--|--|--|--|--|--|
| STOCHAS | 100 | PENTRY |  |  |  |  |  |  |  |
|         |     | MENTRY |  |  |  |  |  |  |  |

### Example 2: (Randomize material properties with CoV = 0.1 and loadings with CoV = 0.3 and default multipliers of standard deviations.)

|         |     |        |       |     |  |  |  |  |  |
|---------|-----|--------|-------|-----|--|--|--|--|--|
| STOCHAS | 200 | LOADS  | GAUSS | 0.3 |  |  |  |  |  |
|         |     | MENTRY | GAUSS | 0.1 |  |  |  |  |  |

| Describer | Meaning                                                                                              |
|-----------|------------------------------------------------------------------------------------------------------|
| SID       | Unique identification number that is selected by the STOCHASTICS Case Control command. (Integer > 0) |
| “PENTRY”  | Flag for randomizing real values on all the element property entries. (Character)                    |
| “MENTRY”  | Flag for randomizing real values on all the independent material property entries. (Character)       |
| “CENTRY”  | Flag for randomizing real values on all the connectivity entries. (Character)                        |
| “LOADS”   | Flag for randomizing real values on all the load entries. (Character)                                |
| “SPCD”    | Flag for randomizing real values on all the SPCD entries. (Character)                                |
| CDF       | Name of a cumulative distribution function. See Remark 2. (Character; Default = GAUSSIANS or blank). |
| CoV       | Coefficient of variance. (Real > 0; Default = 0.05)                                                  |
| m         | Number of standard deviations. See Remark 3. (Real > 0; Default = 3.0)                               |

### Remarks:

1. At least one flag must exist but they can be placed in any order.
2. Currently, only Gaussian distributions are supported.

3. The range of a random variable is defined as  $(\mu - m \cdot \sigma, \mu + m \cdot \sigma)$  where  $\mu$  is the mean of the random variable (or the value of one analysis model parameter on a Bulk Data entry),  $\sigma$  is the standard deviation that is related to  $\mu$ , CoV by  $\sigma = \text{CoV} \cdot \mu$  and  $m$  is the multiplier of the standard deviations.
4.  $m \cdot \text{CoV}$  must be  $< 1.0$ .

## SUPAX

## Conical Shell Fictitious Support

Defines determinate reaction degrees-of-freedom in free bodies for conical shell analysis.

### Format:

| 1     | 2    | 3    | 4  | 5    | 6    | 7  | 8 | 9 | 10 |
|-------|------|------|----|------|------|----|---|---|----|
| SUPAX | RID1 | HID1 | C1 | RID2 | HID2 | C2 |   |   |    |

### Example:

|       |   |   |   |  |  |  |  |  |  |
|-------|---|---|---|--|--|--|--|--|--|
| SUPAX | 4 | 3 | 2 |  |  |  |  |  |  |
|-------|---|---|---|--|--|--|--|--|--|

| Describer | Meaning                                                                                        |
|-----------|------------------------------------------------------------------------------------------------|
| RIDi      | Ring identification number. (Integer > 0)                                                      |
| HIDi      | Harmonic identification number. (Integer ≥ 0)                                                  |
| Ci        | Conical shell degree-of-freedom numbers. (Any unique combination of the Integers 1 through 6.) |

### Remarks:

1. SUPAX is allowed only if an AXIC entry is also present.
2. Up to 12 degrees-of-freedom may appear on a single entry.
3. Degrees-of-freedom appearing on SUPAX entries may not appear on MPCAX, SPCAX, or OMITAX entries.
4. For a discussion of conical shell analysis, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.

## SUPPORT Fictitious Support

Defines determinate reaction degrees-of-freedom in a free body.

### Format:

| 1       | 2   | 3  | 4   | 5  | 6   | 7  | 8   | 9  | 10 |
|---------|-----|----|-----|----|-----|----|-----|----|----|
| SUPPORT | ID1 | C1 | ID2 | C2 | ID3 | C3 | ID4 | C4 |    |

### Example:

|         |    |     |  |  |  |  |  |  |  |
|---------|----|-----|--|--|--|--|--|--|--|
| SUPPORT | 16 | 215 |  |  |  |  |  |  |  |
|---------|----|-----|--|--|--|--|--|--|--|

| Describer       | Meaning                                                                                                                                                |
|-----------------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID <sub>i</sub> | Grid or scalar point identification number. (Integer > 0)                                                                                              |
| C <sub>i</sub>  | Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

### Remarks:

1. The SUPPORT entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint (i.e., SPC<sub>i</sub> entry or PS on the GRID entry).
2. SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.
3. Be careful not to spell SUPPORT with two Ps.
4. Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
5. From 1 to 24 support degrees-of-freedom may be defined on a single entry.
6. See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
7. An alternative to SUPPORT is the SUPPORT1 entry, which is requested by the SUPPORT1 Case Control command.
8. The SUPPORT entry is not allowed in SOLs 106 and 129. SOL 400 does not support ASET<sub>i</sub>, OMIT<sub>i</sub>, BSET<sub>i</sub>, CSET<sub>i</sub>, SUPORT<sub>i</sub>, and QSET<sub>i</sub> except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding "Standard linear physics". This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:

- i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.
9. The SUPPORT entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD. Therefore, SOL101 runs with contact should be converted over to SOL400.

## SUPPORT1 Fictitious Support, Alternate Form

Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPPORT1 must be requested by the SUPPORT1 Case Control command.

### Format:

| 1        | 2   | 3   | 4      | 5   | 6  | 7   | 8  | 9 | 10 |
|----------|-----|-----|--------|-----|----|-----|----|---|----|
| SUPPORT1 | SID | ID1 | C1     | ID2 | C2 | ID3 | C3 |   |    |
|          | ID4 | C4  | -etc.- |     |    |     |    |   |    |

### Example:

|          |   |    |     |  |  |  |  |  |  |
|----------|---|----|-----|--|--|--|--|--|--|
| SUPPORT1 | 5 | 16 | 215 |  |  |  |  |  |  |
|----------|---|----|-----|--|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Identification number of the support set. See Remark 1. (Integer > 0)                                                                                  |
| Idi       | Grid or scalar point identification number. (Integer > 0)                                                                                              |
| Ci        | Component numbers. (Integer 0 or blank for scalar points. Any unique combination of the Integers 1 through 6 for grid points with no embedded blanks.) |

### Remarks:

- The SUPPORT1 entry specifies reference degrees-of-freedom for rigid body motion. It is not intended to be used in place of a constraint; (i.e., SPCi entry or PS on the GRID entry).
- SUPPORT and/or SUPPORT1 entries are required to perform inertia relief in static analysis (SOL 101) if PARAM,INREL,-1 is specified. But if PARAM,INREL,-2 is specified, then SUPPORT and/or SUPPORT1 entries are not required.  
In SOL 101, PARAM,INREL,-1 must also be specified or the SUPPORTi entries will be treated as constraints.
- SUPPORT1 must be requested by the SUPPORT1 Case Control command. The degrees-of-freedom specified on SUPPORT1 will be combined with those on the SUPPORT entry.
- Be careful not to spell SUPPORT with two Ps.
- Degrees-of-freedom specified on this entry form members of the mutually exclusive r-set. They may not be specified on other entries that define mutually exclusive sets. See [Degree-of-Freedom Sets](#) for a list of these entries.
- From 1 to 18 support degrees-of-freedom may be defined on a single entry.
- See [Rigid Body Supports](#) in the *MSC Nastran Reference Guide* for a discussion of supported degrees-of-freedom (members of the r-set).
- In superelement analysis, SUPPORT1 may be specified for points belonging to the residual structure only.

9. The SUPORT entry is not allowed in SOLs 106 and 129. SOL 400 does not support ASETi, OMITi, BSETi, CSETi, SUPORTi, and QSETi except in the following situations:
  - a. Multidisciplinary (linear) analysis. See Remark 3-e. under the ANALYSIS Case Control command regarding “Standard linear physics”. This means there are no subcases for nonlinear analysis using ANALYSIS=NLSTATICS, NLTRAN, HSTAT or HTRAN.
  - b. Linear perturbation with:
    - i. EXTSEOUT Case Control command for external superelement creation. This includes runs with AVLEXB Case Control command.
    - ii. ADAMSMNF Case Control command. These entries must be specified in the BEGIN BULK FLXBDY section. See Remark 21. under the ADAMSMNF Case Control command.
  - c. Superelements defined with BEGIN SUPER may contain ASETi, OMITi, BSETi, CSETi, and QSETi entries.
10. The SUPORT1 entry may not be used in contact in SOL 101 as this is tantamount to executing SOL 400. Additionally, INREL, -2 will not in general converge in SOL101 if run with contact. *Inertial Relief is supported in SOL400 and invoked with the Case Control Command IRLOAD.* Therefore, SOL101 runs with contact should be converted over to SOL400.



## SUPPORT6

## Inertia Relief for SOL 600

Inertia relief used in SOL 600 only.

### Format:

| 1        | 2   | 3    | 4      | 5 | 6 | 7 | 8    | 9 | 10 |
|----------|-----|------|--------|---|---|---|------|---|----|
| SUPPORT6 | SID | METH | IREMOV |   |   |   | IDS1 |   |    |

### Example:

|          |   |   |    |  |  |  |     |  |  |
|----------|---|---|----|--|--|--|-----|--|--|
| SUPPORT6 | 0 | 3 | 1  |  |  |  | 101 |  |  |
| SUPPORT6 | 4 | 3 | -2 |  |  |  |     |  |  |

| Describer | Meaning                                                                                                                                                                                           |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Set ID corresponding to a Case Control SUPPORT1 entry or zero. (Integer; Default = 0)                                                                                                             |
|           | 0 If this is the only SUPPORT6 entry, use this SUPPORT6 entry for all subcases. If there are multiple SUPPORT6 entries, use the one with SID=0 for Marc increment zero.                           |
|           | N Use this SUPPORT6 entry for the subcase specified by Case Control SUPPORT1=N.                                                                                                                   |
|           | Different SUPPORT6 entries can be used for each subcase if desired and different subcases can use different methods.                                                                              |
|           | If there is only one SUPPORT6 entry (with SID=0), no Case Control SUPPORT1 entries are necessary.                                                                                                 |
| METH      | Method to use (Integer; Default = 0)                                                                                                                                                              |
|           | 0 Inertia relief is not active for this subcase.                                                                                                                                                  |
|           | 3 Use the "Support Method", usually specified using param,inrel,-1 for other solution sequences. (See Remark 1.) Input will come from all SUPPORT entries and those SUPPORT1 entries with ID=SID. |
| IREMOV    | Method to retain or remove inertia relief from a previous subcase (Integer; Default = 1).                                                                                                         |
|           | 1 Retain inertia relief conditions from previous subcase.                                                                                                                                         |
|           | -1 Remove inertia relief loads immediately.                                                                                                                                                       |
|           | -2 Remove inertia relief loads gradually                                                                                                                                                          |

| Describer | Meaning                                                                                                                                                                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| IDS1      | <p>IREMOV should be blank or 1 unless METH is 0.</p> <p>ID of SUPORT1 entries to be used if METH=3 and SID=0 (Integer; no Default).</p> <p>For METH=3, only SUPORT1 entries with ID=IDS1 will be used in Marc increment zero. All SUPORT entries will be used.</p> <p>(Used for METH=3 when SID=0 ONLY.)</p> |

**Remark:**

1. The parameter INREL is ignored by SOL 600.

## SURFINI

## Eulerian Initialization Surface

Defines a surface that is used for initialization of regions of an Eulerian mesh. Used in SOL 700 only.

### Format:

|         |     |      |       |         |       |   |   |   |    |
|---------|-----|------|-------|---------|-------|---|---|---|----|
| 1       | 2   | 3    | 4     | 5       | 6     | 7 | 8 | 9 | 10 |
| SURFINI | VID | BSID | COVER | REVERSE | CHECK |   |   |   |    |

### Example:

|         |     |    |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|
| SURFINI | 100 | 37 |  |  |  |  |  |  |  |
|---------|-----|----|--|--|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| VID       | Unique number of an SURFINI region entry. Referenced from TICEUL1. (Integer > 0; Required)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |
| BSID      | ID of a BSURF entry defining the initialization surface. (Integer > 0; Required)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
| COVER     | The processing strategy for Eulerian elements inside and outside of the initialization surface. (Character; Default = INSIDE)<br><br><div style="display: flex; justify-content: space-between;"> <div style="width: 15%;">INSIDE</div> <div>The part of the Eulerian elements that lie inside the closed volume of the initialization surface will obtain the initial conditions belonging to that surface.</div> </div> <div style="display: flex; justify-content: space-between; margin-top: 10px;"> <div style="width: 15%;">OUTSIDE</div> <div>The part of the Eulerian elements that lie outside the closed volume of the initialization surface will obtain the initial conditions that belong to that surface</div> </div> |
| REVERSE   | Auto reverse switch for SURFINI surface segments. (Character; Default = ON)<br><br><div style="display: flex; justify-content: space-between;"> <div style="width: 15%;">ON</div> <div>If necessary, the normals of the SURFINI surface segments are automatically reversed so that they all point in the same general direction and give a positive closed volume.</div> </div> <div style="display: flex; justify-content: space-between; margin-top: 10px;"> <div style="width: 15%;">OFF</div> <div>The segments normals are not automatically reversed.</div> </div>                                                                                                                                                           |
| CHECK     | Checking switch for SURFINI surface segments. (Character; Default = ON)<br><br><div style="display: flex; justify-content: space-between;"> <div style="width: 15%;">ON</div> <div>The normals of the segments are checked to see whether they all point in the same general direction and give a positive closed volume.</div> </div> <div style="display: flex; justify-content: space-between; margin-top: 10px;"> <div style="width: 15%;">OFF</div> <div>The segment normals are not checked.</div> </div> <p>When “REVERSE” is set to “ON”, “CHECK” is automatically set to “ON”.</p>                                                                                                                                         |

### Remarks:

1. All initialization surfaces must form a multifaceted closed volume.

2. An initialization surface can only be used to initialize regions in a Eulerian mesh with appropriate initial conditions. An initialization surface cannot be used as a coupling surface, contact surface or rigid surface.
3. The normal vectors of all segments that form the initialization surface must point in the same general direction, and result in a positive, closed volume. Setting the “REVERSE” option to “ON” ensures that this condition is satisfied, regardless of how segments are initially defined.
4. The “COVER” option determines how Eulerian elements that are (partially) inside or outside of the initialization surface are processed.

## SWLDPRM

### Parameters for CFAST, CWELD, and CSEAM Connector Elements

Overrides default values of parameters for CFAST, CWELD, and CSEAM connectivity search.

**Format:**

| 1       | 2      | 3    | 4      | 5    | 6      | 7    | 8      | 9    | 10 |
|---------|--------|------|--------|------|--------|------|--------|------|----|
| SWLDPRM | PARAM1 | VAL1 | PARAM2 | VAL2 | PARAM3 | VAL3 | PARAM4 | VAL4 |    |
|         | PARAM5 | VAL5 | -etc.- |      |        |      |        |      |    |

**Example:**

|         |        |      |        |   |       |   |  |  |  |
|---------|--------|------|--------|---|-------|---|--|--|--|
| SWLDPRM | GSPROJ | 15.0 | GSMOVE | 2 | PRTSW | 1 |  |  |  |
|---------|--------|------|--------|---|-------|---|--|--|--|

**Alternate Format and Examples:**

| 1       | 2      | 3     | 4      | 5     | 6      | 7     | 8      | 9    | 10 |
|---------|--------|-------|--------|-------|--------|-------|--------|------|----|
| SWLDPRM | PARAM1 | VAL1  | PARAM2 | VAL2  | etc.   | CFAST | PARAM1 | VAL1 |    |
|         | PARAM2 | VAL2  | etc.   | CWELD | PARAM1 | VAL1  | PARAM2 | VAL2 |    |
|         | etc.   | CSEAM | PARAM1 | VAL1  | PARAM2 | VAL2  | etc.   |      |    |

|         |         |        |       |         |     |       |   |       |  |
|---------|---------|--------|-------|---------|-----|-------|---|-------|--|
| SWLDPRM | CHKRUN  | 2      |       |         |     |       |   |       |  |
|         | CWELD   | GSMOVE | 2     | PROJTOL | .04 | PRTSW | 1 | CSEAM |  |
|         | PROJTOL | 0.06   | GMCHK | 1       |     |       |   |       |  |

|         |        |         |      |         |     |       |   |  |  |
|---------|--------|---------|------|---------|-----|-------|---|--|--|
| SWLDPRM | CHKRUN | 2       |      |         |     |       |   |  |  |
|         | CWELD  | GSMOVE  | 2    | PROJTOL | .04 | PRTSW | 1 |  |  |
|         | CSEAM  | PROJTOL | 0.06 | GMCHK   | 1   |       |   |  |  |

| Describer                 | Meaning                                                                                                                                                        |
|---------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------|
| PARAMi                    | Name of the connector parameter. Allowable names are listed in <a href="#">Table 9-39</a> . (Real or Integer)                                                  |
| VALi                      | Value of the parameter. See <a href="#">Table 9-39</a> (Real or Integer)                                                                                       |
| CFAST,<br>CWELD,<br>CSEAM | Keywords to control element type specific parameters. Any parameter following a keyword is applied only to that element type. See Remarks 2 and 3. (Character) |

Table 9-39 PARAMi Names and Descriptions

| Name                 | Type                                          | Default | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |
|----------------------|-----------------------------------------------|---------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ACTVTOL<br>(SOL 600) | Integer $\geq 0$<br>Integer $\leq 2211$       | 1111    | Parameter controlling the behavior of PROJTOL for the different CWELD connection methods. This parameter is entered as an integer and is converted to a four-character string. If its value is less than 1000, the string will be prepended with zeros. The first character (from the left) controls the behavior when method PARTPAT is used. The second controls the behavior when method ELPAT is used. The third controls the behavior when method ELEMID is used and the fourth controls the behavior when method GRIDID is used. For ALIGN the PROJTOL tolerance has no significance. Each digit ( $d_i$ ) in the string can have the value 0 or 1 or 2, where the value 2 only has significance for the methods ELPAT or PARTPAT. The values have the following meaning:<br>0 = PROJTOL is completely deactivated<br>1 = PROJTOL is activated for ELEMID and GRIDID, PROJTOL is activated in initial projections for ELPAT, PROJTOL is only activated over free edges of the patch in auxiliary projections for ELPAT and in initial and auxiliary projections for PARTPAT. Free edges have no neighbors within the set that defines the complete surface.<br>2 = PROJTOL is always activated |
| CHKRUN               | Integer 0, 1, 2                               | 0       | Stop run or allow run to continue after the connectivity elements are generated.<br>0=abort on first error;<br>1=stop after connectivity has been checked; 2=continue run if no errors are found.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
| CNRAGLI              | $90.0 \leq \text{Real} \leq 180.0$<br>or -1.0 | 160.0   | CSEAM only. Minimum angle allowed between the free edges of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is less than the value of CNRAGLI. If set to -1.0, the check will be skipped.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                             |
| CNRAGLO              | $0.0 \leq \text{Real} \leq 90.0$ or<br>-1.0   | 20.0    | CSEAM only. Maximum angle allowed between the normal vectors of shell elements EIDSA and EIDEA or shell elements EIDSB and EIDEB. The CSEAM will not be generated if the angle is greater than the value of CNRAGLO. If set to -1.0, the check will be skipped.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |

Table 9-39 PARAMi Names and Descriptions

| Name                | Type             | Default | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                  |
|---------------------|------------------|---------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| CSVOUT              | Integer $\geq 0$ | 0       | <p>Print diagnostic output in a comma separated .csv file useful for reports. The users may view or manipulate data using Microsoft Excel, or other spread sheet programs that can process CSV format.</p> <p>0        no .csv output.</p> <p><i>unitnum</i> &gt; 0 unit number of the .csv file assigned via the ASSIGN statement in the File Management Section, for example,</p> <p>ASSIGN USERFILE=myfile.csv UNIT=<i>unitnum</i><br/>FORM=FORMATTED DELETE STATUS=NEW</p>                                                                                                                                               |
| CWSETS<br>(SOL 600) | Integer > 0      | 0       | <p>Parameter to control the automatic creation of four element sets with the elements involved in the CWELD connections.</p> <p>0 = the sets will not be created<br/>1 = four sets will be created automatically<br/>“fastener_all_beams_inc0000”, the set containing all connector beam elements.<br/>“fastener_all_faces_sidea_inc0000”, the set containing all elements with patches on side B of the connection.<br/>“all_fastener_warnings_inc0000”, the set containing all elements involved in CWELD warning messages.<br/>Defining sets with any of these names must be avoided and will be considered an error.</p> |
| CWSPOT<br>(SOL 600) | 0 < Integer < 3  | 1       | <p>Parameter to choose the method for modifying the beam length.</p> <p>1 = scale the stiffness of the beam<br/>2 = reposition the end nodes of the beam<br/>3 = reposition the auxiliary patch nodes and dthe end nodes of the beam.</p>                                                                                                                                                                                                                                                                                                                                                                                    |
| DELMAX<br>(SOL 600) | Real             | 0.1     | <p>Maximum allowable parametric coordinate change during the iteration process for finding the projection on a patch. At first DELMAX is not activated, i.e., the parametric coordinate change is not limited during the iteration process. The parameter is only activated when the full Newton Raphson iteration process for a projection did not converge. In that case the iteration process is restarted with DELMAX activated.</p>                                                                                                                                                                                     |
| DLDMAX<br>(SOL 600) | Real > 0.0       | 5.0     | <p>Default value for LDMAX, the largest ratio of length to characteristic diameter.</p>                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |

Table 9-39 PARAMi Names and Descriptions

| Name                | Type                             | Default | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |
|---------------------|----------------------------------|---------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| DLDMIN<br>(SOL 600) | Real > 0.0                       | 0.2     | Default value for LDMIN, the smallest ratio of length to characteristic diameter.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                    |
| DRATIO              | $1.0 \leq \text{Real} \leq 10.0$ | 1.0     | CFAST or CWELD (PARTPAT and ELPAT) only. Increases the Connector patch diameter and is used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. See Remark 10.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                            |
| EPSITR<br>(SOL 600) | Real > 0.0                       | 1.0E-5  | Tolerance to terminate the iteration process for finding the projection on a patch. If the parametric coordinate change in an iteration is less than EPSITR the projection is accepted as converged.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |
| GMCHK               | Integer 0, 1, 2, 3               | 0       | <p>For CSEAM, CWELD with ELPAT or PARTPAT format, and CFAST only.</p> <p>0=no geometry error checks;</p> <p>1=check errors of the CSEAM across a cutout or over a corner with patch elements in plane or out of plane;</p> <p>2=check errors of CSEAM and output all candidate shell elements if an error is encountered.</p> <p>If GMCHK=1 or 2 and an error is detected, the program will loop back to search for next candidate element until a good pair of connection is found or all adjacent elements have been checked. In the latter case, a user fatal message 7595, 7638, or 7667 will be issued. A UFM 7595 is issued if the normal angles between the patches at end GS or the patches at end GE exceed the value of GSPROJ; a UFM 7638 is issued if either the length of the seam spans more than three elements or the seam spans a cutout; a UFM 7667 is issued if the normal angles between the top patches at GS and GE or the normal angles between the bottom patches at GS and GE exceeds CNRAGLO or if the angle between the free edges of the shell elements onto which GS and GE are projected is less than CNRAGLI.</p> <p>3=check backward projections for CWELD with ELPAT or PARTPAT format and CFAST. See Remark 8.</p> |



Table 9-39 PARAMi Names and Descriptions

| Name                | Type                                        | Default                     | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                |
|---------------------|---------------------------------------------|-----------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| GSMOVE              | Integer                                     | 0                           | Maximum number of times GS for the CFAST or CWELD (PARTPAT or ELPAT options only) or GS/GE for the CSEAM is moved in case a complete projection of all auxiliary points has not been found. See Remark 9.                                                                                                                                                                                                                                                                                                                                                                                  |
| MOVGAB              | Integer 0, 1                                | 0                           | Option to correct the locations of user defined GA/GB for CFAST and CWELD elements.<br><br>0 = keep the locations of the user specified GA/GB and connect them to shell elements directly.<br><br>1 = generate new grids with corrected locations to connect shell elements.                                                                                                                                                                                                                                                                                                               |
| GSPROJ              | $0.0 \leq \text{Real} \leq 90.0$<br>or -1.0 | 20.0<br>(-89.0 for SOL 600) | Maximum angle allowed between the normal vectors of shell A and shell B. The connector element will not be generated if the angle between these two normal vectors is greater than the value of GSPROJ. For CSEAM, see also GMCHK for additional error checks using GSPROJ. If GSPROJ is set to -1.0 (or -89.0 for SOL 600), the program will skip the checking of GSPROJ.                                                                                                                                                                                                                 |
| GSTOL               | $\text{Real} \geq 0.0$                      | 0.0                         | For CFAST or CWELD (PARTPAT and ELPAT only), if $\text{GSTOL} > 0.0$ and the distance between GS and the projected point GA or GB is greater the GSTOL, a UFM 7549 is issued and the CFAST or CWELD is rejected. For CSEAM, if $\text{GSTOL} > 0.0$ and the distance between GS and the projected point GSA or GSB or the distance between GE and the projected point GEA or GEB is greater than the GSTOL, a UFM 7549 is issued and the CSEAM is rejected.                                                                                                                                |
| MAXEXP<br>(SOL 600) | Integer > 0                                 | 2                           | Parameter to control the maximum number of expansions in the search for projections of the auxiliary nodes. First the master patch will be searched. If no projection is found on the master patch a first expansion will be made including all neighboring patches of the master patch. If no projection is found on any of the new patches a second expansion will be made including all neighbors of the patches tried so far. This process continues until the number of expansions exceeds MAXEXP. Two patches are neighbors if they share at least one node in their connectivities. |

Table 9-39 PARAMi Names and Descriptions

| Name                | Type                              | Default                     | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                            |
|---------------------|-----------------------------------|-----------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| MAXITR<br>(SOL 600) | Integer > 0                       | 20                          | The maximum number of iterations allowed in the iteration process for finding the projection on a patch.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                               |
| NREDIA              | Integer 0, 1, 2, 3, 4, 5, 6, 7, 8 | 0                           | CFAST or CWELD (PARTPAT and ELPAT) only. Maximum number of times the diameter D is reduced in half in case a complete projection of all points has not been found.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                     |
| PROJTOL             | $0.0 \leq  \text{Real}  \leq 1.0$ | 0.02<br>(0.199 for SOL 600) | <p>For CFAST or CWELD, tolerance to accept the projected point GA or GB if the computed coordinates of the projection point lie outside the shell element but is located within <math> \text{PROJTOL} </math>*(dimension of the shell element forming the patch). For the CSEAM, a projection from GS/GE will always be attempted as if <math>\text{PROJTOL}=0.0</math> and if one cannot be found then the non-zero value of <math> \text{PROJTOL} </math> will be used.</p> <p>Regardless of the value of SWLDPRM PROJTOL, the algorithm starts by assuming a zero projection tolerance for the projections of GA/GB for the CWELD option "PARTPAT" or the CFAST option "PROP" and for GAHi/GBHi for the CWELD options "PARTPAT" and "ELPAT" and any CFAST option. The tolerance is increased by 0.02 until a projection is found or the PROJTOL value is reached.</p> <p>This can be turned off while computing the auxiliary grid projection onto EIDA/EIDB or onto the adjacent elements of EIDA/EIDB by setting <math>\text{PROJTOL} = -\text{value}</math> where <math>0.0 \leq \text{value} \leq 1.0</math>. In this case, the projection calculation starts at tolerance = <math> \text{PROJTOL} </math>. For the rest of the projection search, the algorithm reverts back to the iteration starting at tolerance = 0.0.</p> |
| WMASS               | Integer 0, 1                      | 0                           | For CWELD to react to a non-zero density 1 value is required.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                          |

Table 9-39 PARAMi Names and Descriptions

| Name                | Type                                          | Default              | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                     |
|---------------------|-----------------------------------------------|----------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| PRTSW               | Integer 0, 1, 2 ,<br>11, 12, 100, 101,<br>111 | 0 (2 for<br>SOL 600) | Print diagnostic in output or punch out internally generated RBE3 elements and auxiliary grids in Bulk Data format for the connector elements.<br>0=no diagnostic output (appropriate USER MESSAGES such as 'USER WARNING MESSAGE 7636 will continue to be issued);<br>1=print diagnostic output in exponential format to f06 file;<br>2=punch diagnostic output in exponential format to .pch file;<br>11=print diagnostic output in real format to .f06 file;<br>12=punch diagnostic output in real format to .pch file.<br>100=punch out Bulk Data without diagnostic output.<br>101= punch out Bulk Data and print diagnostic output in exponential format to f06 file.<br>111= punch out Bulk Data and print diagnostic output in real format to f06 file. |
| RBE3WT<br>(SOL 600) | Real                                          | 0.0                  | Default RBE3 distance weighting exponent.<br>The weight factor for each retained node in a RBE3 involved in a CWELD connection is: $f_i = 1/d_i^n$ , where:<br>$f_i$ is the weighting factor for retained node $i$ .<br>$d_i$ is the distance from the tied node to retained node $i$<br>$n$ is the weighting exponent RBE2WT<br>Negative values for RBE3WT are not recommended, since they will result in heavier weighting for nodes further away. The default results in uniform weighting ( $f_1 = 1$ ).                                                                                                                                                                                                                                                    |

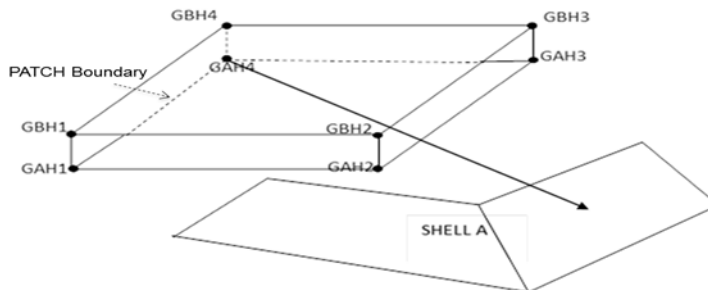
Table 9-39 PARAMi Names and Descriptions

| Name                | Type                     | Default | Description                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                   |
|---------------------|--------------------------|---------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| GSCURV<br>(SOL 600) | $-90 < \text{Real} < 90$ | 20.0    | <p>Maximum angle allowed between the normal vectors of a patch to which an auxiliary node projects and its corresponding auxiliary and master patches. It provides a measure to monitor the curvature of a surface and to recognize patches that belong to, for example, stiffeners. A connection is not generated if the angle between the normal vectors is greater than <math>90 - \text{GSCURVE}</math> meaning that the patches are almost normal to each other. In that case, the patch is reflected and the search proceeds to the next patch in the list. If the angle is between zero and GSCURV, no message is displayed. If the angle is between GSCURV and <math>90 - \text{GSCURV}</math>, a large angle warning is displayed. The following three tests are performed in the order given below when GSCURV is positive:</p> <p>If <math>0 &lt; \text{angle} &lt; \text{GSCURVE} \geq \text{OK}</math><br/> If <math>\text{GSCURVE} &lt; \text{angle} &lt; 90 - \text{GSCURV} &gt;</math> trigger a warning.<br/> If <math>\text{angle} &gt; 90 - \text{GSCURV} \geq \text{reject}</math>.</p> <p>Note that the warning condition is never triggered when <math>\text{GSCURV} &gt; 45</math> as it is overruled by the reject condition.</p> <p>If GSCURV is negative, the projection is always accepted and a warning is issued when the angle is larger than <math> \text{GSCURV} </math>.</p> |
| SCLSKIN             | $\text{Real} \geq 0.0$   | 0.0     | <p>CFAST or CWELD (PARTPAT and ELPAT) only. If set to a value <math>&gt; 0.0</math> the CFAST or CWELD connector will be stiffened by increasing the bending moment of inertia ratio by SCLSKIN of the shell elements involved in the connector patch. SCLSKIN is Factor used to stiffen the connector patch when a loss of stiffness occurs for the FEM model as mesh size is made smaller. An initial recommended value is 0.1; See Remark 10..</p>                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                         |

**Remarks:**

1. This entry changes the default settings of control variables for the CFAST, CWELD, and CSEAM connector elements. None of the parameters of this entry are required. Only one SWLDPRM entry is allowed in the Bulk Data Section.

2. If any of the key words CFAST, CWELD, and CSEAM does not appear on this entry, then a parameter set on this entry is considered “global” and applies to all the connector elements in the model. Any parameter set on this entry that comes before a key word CFAST, CWELD, or CSEAM is considered global.
3. Any parameter set on the entry that comes after a key word such as CFAST will only apply to that connector element type until another key word such as CSEAM is encountered. If a parameter is defined for a specific connector type that does not apply to that connector type then it will be ignored.
4. Blank fields are allowed for readability. However, a parameter name must be followed in the immediately following field by the corresponding parameter value. If the parameter name falls in the field just before a continuation field, then its parameter value must be placed in the first field after the continuation marker of the continuation entry.
5. Connectivity information is generated for the CFAST and CSEAM elements. For the CWELD elements, connectivity information is only generated for the PARTPAT, ELPAT, ELEMID, and GRIDID options.
6. The details of individual connector connectivity can be found on the appropriate CFAST, CWELD, and CSEAM Bulk Data entries.
7. The CHKRUN parameter must be global.
8. Backward connections sometimes occur if the patch is near the boundary of a structure and there is a "vertical" flange associated with the patch elements. In this case GMCHK=3 may be used to prevent backward projection. See the figure below. If GMCHK=3 and a backward projection is detected, the program will not connect this projected shell element. Instead, it will continue searching iterations until a satisfying connection is reached.

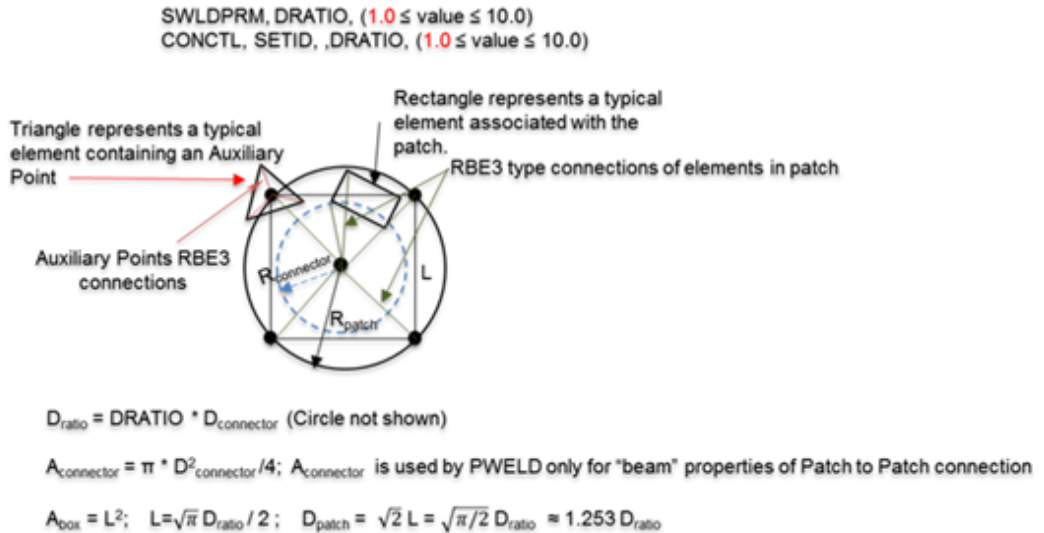


9. If the GSMOVE specification limit is reached for the CFAST or the CWELD with options "PARTPAT" and "ELPAT" and SWLDPRM NREDIA  $\geq 0$ ; then the diameter of the connector will be reduced by half to compute new locations of auxiliary grids. If necessary this is repeated until the NREDIA specified value is reached.
  - a. When the NREDIA  $\neq 0$  is initiated, the GS at its current location is used for GSMOVE  $\geq 0$ .
  - b. When the NREDIA  $\neq 0$  is initiated, the GS at its original location is used for the new option GSMOVE  $< 0$ .

- Connector contribution to a structural model's overall stiffness is sensitive to the model's mesh size and the orientation of the connector relative to the mesh. Thus, the discretization process itself may cause, for example, a model using a fine mesh to be stiffer in torsion than a corresponding model using a coarse mesh. Also for production models that correlate well with test, refining the mesh may cause an inherent overall loss of stiffness due to mesh refinement and hence loss of correlation.

To allow the user some control over model stiffness, the new connectors (CWELD with ELPAT or PARTPAT or CFAST) are provided with two options to provide additional connector stiffness. The two options may be used individually or in combination.

The first stiffening technique is activated by "SWLDPRM, DRATIO, (1.0 ≤ value ≤ 10.0)" or "CONCTL, SETID, ,DRATIO, (1.0 ≤ value ≤ 10.0)". For this option the diameter,  $D_{ratio}$ , is defined as  $D_{ratio} = DRATIO * D_{connector}$ . This results in the diameter of the patch taking a value of  $D_{patch} = \sqrt{\pi/2} D_{ratio}$ . The default of DRATIO is a value=1.0 which implies the diameter of the patch is computed in the standard fashion. For the patch to patch connection for the "beam" properties of the CWELD, the area is still computed as  $A_{connector} = \pi D_{connector}^2 / 4$  as defined in the PWELD entry.



A *disadvantage* of this method is that as DRATIO is increased using the global command SWLDPRM, DRATIO, *value*?; some connector elements may begin to fail because they may no longer be able to find a patch projection.

To overcome this, the "SWLDPRM, NREDIA, *Integer\_value*" can be increased to a value as high as *Integer\_value* = 8 to allow failing welds to halve their patch diameters up to eight times. Whenever a connector has its diameter reduced a message such as: "DA IS REDUCED BY HALF TO 8.8623E+00" when PRTSW diagnostic output is requested. In this message the DA is computed as : DA = L/2.

If the "SWLDPRM, NREDIA, *Integer\_value*" is not an approach the user wishes to pursue, then for these failing elements, the bulk data entry CONCTL, SETID, ,DRATIO, value can be used to define a set for failing connectors and set a value of DRATIO for these connectors that allows them to find a projection.

The second stiffening algorithm attempts, based on the diameter of the connector, to determine a measure of the mesh discretization.

This feature is activated by setting "SWLDPRM, SCLSKIN > 0.0" or "CONCTL, SETID, SCLSKIN > 0.0". The default is a value = 0.0 which implies no stiffening. When activating the "SWLDPRM, SCLSKIN, real value" option for the first time with a new FEM, it is recommended that an initial value = 0.1 be used.

Depending on the complexity of the model and the overall mesh size and the number of connectors within the model and the diameter of the connectors relative to the mesh, the *default value* tends to stiffen a structural model from about 0.4% to about 4%. A value of SCLSKIN=10.0 stiffens coarser mesh models by about 10% to 11% and finer mesh models by about 2% to 6%.

The contribution of the stiffening algorithm to the overall stiffness of the FEM model eventually reaches a limit. For example, a very large value SCLSKIN=100 increases the stiffness of the models overall by only about 0.1% to 2% over the stiffness obtained for SCLSKIN=10.

For a correlated structural model evaluated at a specific mesh size, *with an aim to refine the mesh for some portion of this model containing connectors*, while leaving other portions containing connectors with an unmodified mesh, it is recommended that the "SCLSKIN, *real value*" be entered on the CONCTL bulk data entry referring to the connectors within the area of the refined mesh. Different refined mesh areas within the structural model can have different values of SCLSKIN associated to the specific connectors in each refined region.

For post processing affected shell elements, an updated EPT table is available after module MODGM2. It contains the PSKNSHL record that correlates the property data of the shells involved and a list of shell elements for each patch modified.

Neither of these stiffening methods have any effect on the mass computations of the model.

11. This entry is ignored in part super-elements. Inside the part super-elements, the default settings will be applied.
12. If Modules are present then this entry may only be specified in the main Bulk Data section.

## Entries T-Y

### TABD1MD TABLED1 entries internal modification in SOL 600

Defines how TABLED1 entries are internally modified in SOL 600.

SOL 600 usually requires that the first point of all TABLED1 entries used to describe time histories start with time/amplitude of (0.0, 0.0). If the user omits this entry, all TABLED1 entries that do not start with (0.0, 0.0) will be modified internally to add two points at the start, the first at (0.0, 0.0) and the second at (0.001, V1) where V1 is the first amplitude of the original TABLED1 entry for that curve.

#### Format

| 1       | 2    | 3    | 4   | 5   | 6  | 7  | 8  | 9 | 10 |
|---------|------|------|-----|-----|----|----|----|---|----|
| TABD1MD | TID1 | THRU | ID2 | MOD | T2 | A2 | T3 |   |    |

#### Example:

|         |   |      |   |   |      |     |      |  |  |
|---------|---|------|---|---|------|-----|------|--|--|
| TABD1MD | 1 | THRU | 5 | 1 | .002 | 1.0 | .003 |  |  |
|---------|---|------|---|---|------|-----|------|--|--|

| Descriptor | Meaning                                                                                                                                                             |
|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID1        | First TABLED1 ID. (Integer > 0; no Default required)                                                                                                                |
| ID2        | Last TABLED1 ID to which MOD, T1 and A1 apply. See Remark 2. (Integer; Default = ID1)                                                                               |
| MOD        | Flag indicating whether or not to modify TABLED1 entries with ID in the range ID1 to ID2. (Integer; Default = 1)<br>0 Do not modify the table<br>1 Modify the table |
| T1         | New time for the second point. See Remark 1. (Real; Default = 0.001)                                                                                                |
| A1         | New amplitude of the second point. See Remark 2. (Real; Default = 1.0)                                                                                              |
| T2         | New time for the third point. See Remark 1. (Real; Default = 0.002)                                                                                                 |

#### Remarks:

1. If MOD=1, all TABLED1 entries with ID's in described by ID1 to ID2 will be modified to add two points to the beginning of the each table. The first point will be at (0.0, 0.0) the second point will be at (T1,A1). The original first point will be modified to be at time T3.
2. All TABLED1 entries in the range ID1 to ID2 will be modified as indicated. Repeat this entry as many times as necessary to specify all TABLED1 ID's that should either be modified or not be modified.
3. Make sure to set MOD=0 to all TABLED1 entries that are not used to describe time history loading.
4. The ID1 to ID2 range may include values that do not have any TABLED1 ID's in the model.



5. If all TABLED1 entries are to be modified or if they do not all start with the same T2, A2, T3 values, PARAM,MTALBD1M PARAM,MTALBD1T may be entered.

## TABDMP1 Modal Damping Table

Defines modal damping as a tabular function of natural frequency.

Format:

| 1       | 2   | 3    | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|------|----|----|----|----|--------|---|----|
| TABDMP1 | TID | TYPE |    |    |    |    |        |   |    |
|         | f1  | g1   | f2 | g2 | f3 | g3 | -etc.- |   |    |

Example:

|         |     |        |     |        |      |  |  |  |  |
|---------|-----|--------|-----|--------|------|--|--|--|--|
| TABDMP1 | 2   |        |     |        |      |  |  |  |  |
|         | 2.5 | .01057 | 2.6 | .01362 | ENDT |  |  |  |  |

| Describer      | Meaning                                                                 |
|----------------|-------------------------------------------------------------------------|
| TID            | Table identification number. (Integer > 0)                              |
| TYPE           | Type of damping units. (Character: “G”, “CRIT”, or “Q”; Default is “G”) |
| f <sub>i</sub> | Natural frequency value in cycles per unit time. (Real ≥ 0.0)           |
| g <sub>i</sub> | Damping value. (Real)                                                   |

Remarks:

1. Modal damping tables must be selected with the Case Control command SDAMPING = TID.
2. The frequency values, f<sub>i</sub>, must be specified in either ascending or descending order, but not both.
3. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-146](#) discontinuities are allowed only between points f<sub>2</sub> through f<sub>7</sub>. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-146](#), the value of g at f = f<sub>3</sub> is  $g = (g_3 + g_4)/2$ .
4. At least one continuation entry must be specified.
5. Any f<sub>i</sub> or g<sub>i</sub> entry may be ignored by placing “SKIP” in either of the two fields used for that entry.
6. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
7. The TABDMP1 uses the algorithm

$$g = g_T(f)$$

where  $f$  is input to the table and  $g$  is returned. The table look-up  $g_T(f)$  is performed using linear interpolation within the table and linear extrapolation outside the table using the last two end points. See [Figure 9-146](#). No warning messages are issued if table data is input incorrectly. See Remark 11.

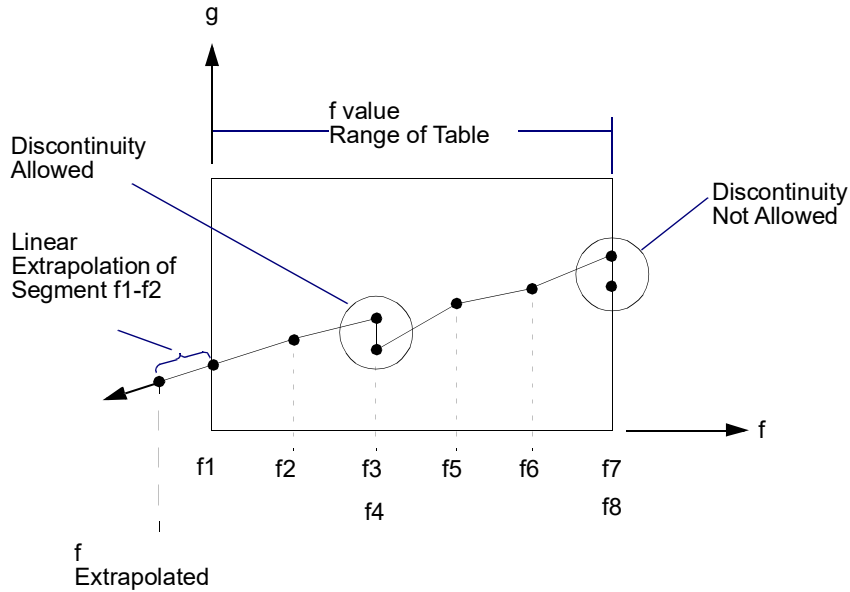


Figure 9-146 Example of Table Extrapolation and Discontinuity

8. This form of damping is used only in modal formulations of complex eigenvalue analysis, frequency response analysis, or transient response analysis. The type of damping used depends on the solution sequence (structural damping is displacement-dependent, and viscous damping is velocity-dependent). See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for the equations used.
9. PARAM,KDAMP may be used in solution sequences that perform modal frequency and modal complex analysis, to select the type of damping.

| KDAMP       | Result      |
|-------------|-------------|
| 1 (Default) | $B$ Matrix  |
| -1          | $(1 + ig)K$ |

See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide* for a full explanation.

10. If TYPE is “G” or blank, the damping values  $g_i$ , etc., are in units of equivalent viscous dampers, as follows:

$$b_i = \frac{g_i}{\omega_i} K_i$$

(See [Formulation of Dynamic Equations in SubDMAP GMA](#) in the *MSC Nastran Reference Guide*) If TYPE is “CRIT”, the damping values  $g_i$ , etc., are in the units of fraction of critical damping  $C/C_0$ . If TYPE is “Q”, the damping values  $g_i$  are in the units of the amplification or quality factor,  $Q$ . These constants are related by the following equations:

$$C/C_0 = g/2$$

$$Q = \begin{cases} 1/(2C/C_0) \\ 1/g \end{cases}$$

11. A user warning message is used if either of the following conditions is satisfied:
  - a. The modal damping value is computed as a result of extrapolation.
  - b. The computed modal damping value is negative.

For any modal damping value that satisfies condition a or b, the program lists the cyclic frequency and the corresponding modal damping value and indicates whether this value was computed as a result of interpolation or extrapolation. For the latter case, it also indicates whether the extrapolation was beyond the left end of the table or beyond the right end of the table.

If a modal damping value satisfies *both* of the conditions, a and b above (that is, the modal damping value is computed as a result of extrapolation *and* it is negative), the program terminates the job with a user fatal message.

The user can prevent the program from terminating the job as above by specifying MDAMPEXT=1 [or SYSTEM(426)=1] on the NASTRAN statement. The user fatal message mentioned above does inform the user of this avoidance scheme.

12. The modal damping matrix generated by this approach is not affected by the presence of rotors in the model. Also, the circulation terms are not generated for damping corresponding to rotor degrees of freedom.
13. If Modules are present then this entry may only be specified in the main Bulk Data section.

TABL3D

Multi-Dimensional Table

Specifies a table where an entry can be a function of up to 4 variables such as strain, temperature, strain rate, etc for SOL 600 and SOL 400.

Format 0: Simple Table Type 0 - entry is a function of only one variable:

| 1       | 2    | 3  | 4      | 5     | 6     | 7     | 8  | 9  | 10 |
|---------|------|----|--------|-------|-------|-------|----|----|----|
| TABL3D0 | ITID |    | KIND   | EXTRP | ITIDS | ITIDB | SM |    |    |
|         | X1   | Y1 | X2     | Y2    | X3    | Y3    | X4 | Y4 |    |
|         | X5   | Y5 | -etc.- |       |       |       |    |    |    |

Format 1: Multi-Dimensional Table Type 1 - entry is a function of 2, 3, or 4 variables, data entered one row at a time

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10                |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|-------------------|
| TABL3D1 | ITID   | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |                   |
|         | NW3    | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |                   |
|         | ITIDS1 | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |                   |
|         | SM1    | SM2    | SM3    | SM4    |        |        |        |        |                   |
|         | X11    | X12    | X13    | X14    | X15    | X16    | -etc.- |        |                   |
|         | X21    | X22    | X23    | X24    | X25    | X26    | -etc.- |        | Enter if<br>NW2>0 |
|         | X31    | X32    | X33    | X34    | X35    | X36    | -etc.- |        | Enter if<br>NW3>0 |
|         | X41    | X42    | X43    | X44    | X45    | X46    | -etc.- |        | Enter if<br>NW4>0 |
|         | Y1     | Y2     | Y3     | Y4     | Y5     | Y6     | -etc.- |        | See<br>Remark 1.  |

Format 2: Multi-Dimensional Table Type 2- entry is a function of 2, 3, or 4 variables, data entered one point at a time

| 1       | 2      | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10                |
|---------|--------|--------|--------|--------|--------|--------|--------|--------|-------------------|
| TABL3D2 | ITID   | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |                   |
|         | NW3    | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |                   |
|         | ITIDS1 | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |                   |
|         | SM1    | SM2    | SM3    | SM4    |        |        |        |        |                   |
|         | X11    | X12    | X13    | X14    | X15    | X16    | -etc.- |        |                   |
|         | X21    | X22    | X23    | X24    | X25    | X26    | -etc.- |        | Enter if<br>NW2>0 |
|         | X31    | X32    | X33    | X34    | X35    | X36    | -etc.- |        | Enter if<br>NW3>0 |

|  |     |     |     |     |     |     |        |  |                   |
|--|-----|-----|-----|-----|-----|-----|--------|--|-------------------|
|  | X41 | X42 | X43 | X44 | X45 | X46 | -etc.- |  | Enter if<br>NW4>0 |
|  | Y1  | Y2  | Y3  | Y4  | Y5  | Y6  | -etc.- |  | See<br>Remark 2.  |

**Format 3: Multi-Dimensional Table Type 3- entry is specified by a formula (SOL 600 only)**

| 1       | 2       | 3      | 4      | 5      | 6      | 7      | 8      | 9      | 10               |
|---------|---------|--------|--------|--------|--------|--------|--------|--------|------------------|
| TABL3D3 | ITID    | NV     | KIND1  | KIND2  | KIND3  | KIND4  | NW1    | NW2    |                  |
|         | NW3     | NW4    | EXTRP1 | EXTRP2 | EXTRP3 | EXTRP4 |        |        |                  |
|         | ITIDS1  | ITIDB1 | ITIDS2 | ITIDB2 | ITIDS3 | ITIDB3 | ITIDS4 | ITIDB4 |                  |
|         | SM1     | SM2    | SM3    | SM4    |        |        |        |        |                  |
|         | Formula |        |        |        |        |        |        |        | See<br>Remark 3. |

| Describer                                 | Meaning                                                                                                                                                                       |
|-------------------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ITID                                      | Table identification number. (Integer > 0; no Default)                                                                                                                        |
| SM or SMi                                 | Flag to indicate smoothing of the table data. (Integer; Default = 0)<br>0 Do not smooth the data<br>1 Smooth the data                                                         |
| NV                                        | Number of variables the entry is a function of (Integer, 1, 2, 3, or 4; no Default)                                                                                           |
| KIND or<br>KINDi                          | “Independent” variable type (such as strain, temperature, Integer > 0; no Default; see <a href="#">Table 9-40</a> for application values)                                     |
| EXTRP or<br>EXTRPi                        | Extrapolation flag. (Integer; Default = 2)<br>1 Do not allow extrapolation<br>2 Allow extrapolation (both ends of curve)                                                      |
| ITIDS or<br>ITIDSi,<br>ITIDB or<br>ITIDBi | Table IDs meant for future expansion to reference other tables from this table when trying to evaluate this table outside its defined range. Currently not used, leave blank. |
| Nwi                                       | Number of X values of each variables. (i can range from 1 to 4) (Integer > 0; no Default)                                                                                     |
| Xi or Xij                                 | Value of “independent” variable such as strain, temperature, ... (Real; no Default)                                                                                           |
| Yi                                        | Value of the quantity desired such as stress, Poisson’s ratio, ... (Real; no Default)                                                                                         |

The “Independent” variable(s) should be selected from Table 9-40:

Table 9-40

| Independent Variable Type |                                                                  |    |                                                                                        |    |                                                       |
|---------------------------|------------------------------------------------------------------|----|----------------------------------------------------------------------------------------|----|-------------------------------------------------------|
| 1                         | time                                                             | 26 | $Z_0$ coordinate                                                                       | 51 | wavelength (used in spectral radiation)               |
| 2                         | normalized time                                                  | 27 | $s_0 = \sqrt{x_0^2 + y_0^2 + z_0^2}$                                                   | 52 | creep strain                                          |
| 3                         | increment number                                                 | 28 | contact force $ F $                                                                    | 53 | pressure or primary quantity in diffusion             |
| 4                         | normalized increment time                                        | 29 | not available                                                                          | 54 | equivalent strain rate for non-Newtonian viscosity    |
| 5                         | x coordinate                                                     | 30 | $\sigma_n$ (normal stress)                                                             | 55 | normalized arc distance                               |
| 6                         | y coordinate                                                     | 31 | voltage**                                                                              | 56 | distance to other contact surface (near contact only) |
| 7                         | z coordinate                                                     | 32 | current**                                                                              | 57 | terms of series                                       |
| 8                         | $s = \sqrt{x^2 + y^2 + z^2}$                                     | 33 | $\left(\frac{\text{current radius}}{\text{radius of throat}}\right)^2$<br>(see throat) | 58 | hydrostatic stress                                    |
| 9                         | $\theta$ angle                                                   | 34 | Not available                                                                          | 59 | hydrostatic strain                                    |
| 10                        | mode number                                                      | 35 | Not available                                                                          | 60 | Not available                                         |
| 11                        | frequency                                                        | 36 | Not available                                                                          | 61 | Not available                                         |
| 12                        | temperature                                                      | 37 | gasket closure distance                                                                | 62 | 2nd state variable                                    |
| 13                        | function                                                         | 38 | displacement magnitude                                                                 | 63 | 3rd state variable                                    |
| 14                        | fourier                                                          | 39 | stress rate                                                                            | 64 | 4th state variable                                    |
| 15                        | $\frac{-\dot{\rho}}{\dot{\epsilon}}$ (equivalent plastic strain) | 40 | experimental data                                                                      | 65 | 5th state variable                                    |
| 16                        | $\dot{\epsilon}$ (equivalent strain rate)                        | 41 | porosity                                                                               | 66 | loadcase number*                                      |
| 17                        | Not available                                                    | 42 | $\frac{\dot{v}}{c}$ void ratio                                                         | 67 | degree of cure*                                       |
| 18                        | arc length                                                       | 43 | $\dot{\epsilon}$ (equivalent creep strain rate)                                        | 68 | magnetic field intensity**                            |
| 19                        | relative density (not available for shells)                      | 44 | minor principal strain                                                                 | 69 | equivalent mechanical strain                          |
| 20                        | $\bar{\sigma}$ (equivalent stress)                               | 45 | distance from neutral axis (-1/2, +1/2)                                                | 70 | 1st strain invariant                                  |
| 21                        | magnetic induction**                                             | 46 | normalized distance from neutral axis (-1, +1)                                         | 71 | 2nd strain invariant                                  |

Table 9-40

| Independent Variable Type |                           |    |                                                                   |    |                           |
|---------------------------|---------------------------|----|-------------------------------------------------------------------|----|---------------------------|
| 22                        | velocity                  | 47 | local x-coordinate of layer point for open or closed section beam | 72 | 3rd strain invariant      |
| 23                        | parameter diameter**      | 48 | local y-coordinate of layer point for open or closed section beam | 73 | local strain component    |
| 24                        | X <sub>0</sub> coordinate | 49 | not available                                                     | 74 | damage                    |
| 25                        | Y <sub>0</sub> coordinate | 50 | not available                                                     | 75 | accumulated crack growth  |
|                           |                           |    |                                                                   | 76 | relative sliding velocity |

## Remarks (General):

1. Independent Variable Types marked as 'NOT AVAILABLE' are not available in either SOL 600 or SOL 400. Independent variable types marked with \* are only available in SOL 600. Independent variable types marked with \*\* are only available in SOL 600 through hand-editing of the Marc input file. The rest are available in SOL 600 and SOL 400.
2. Dependent quantities cannot be arbitrary functions of the Independent Variable Types shown in Table 9-40. For e.g., Young's Modulus can be varied with Temperature and Space but cannot be varied with Time. Incorrect tabular settings that are not allowed will be errored out by the program.
3. For time-independent materials, dependent variables (i.e., yield stress) can typically be defined only as a function of temperature, stress/strain data or coordinates. For time-dependent materials, (e.g., creep coefficient), properties can also be defined as a function of time.
4. For contact dependent variables, (i.e., friction coefficient) can typically be defined as a function of temperature, contact body quantities like normal stress, body force, distance from body, relative velocity, etc.
5. If the independent variable is out of range of the table, the user can indicate if the last point in the table should be used or if the table should be extrapolated. Extrapolation means the table is continued with its first or last slope. Care should be taken with extrapolation, particularly for material properties.
6. Independent variable types for coordinates (5, 6, 7) depend on the type of analysis flagged. For a small strain or Total Lagrange or non-mechanical analysis, these coordinates refer to the original coordinates. For Updated Lagrange mechanical analysis, these coordinates refer to the updated
7. At present, options to specify additional tables using ITIDS, ITIDB, ITIDS1, ITIDB1, ... ITIDS4, ITIDB4 are not active and will be ignored if entered.

## Remarks (for SOL 600 only):

1. The function is read by giving NW1 data points (NW4\*NW3\*NW2) times. The program reads the data using the following method.



```
do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      read nw1 values f(X1, K2, K3, K4)
    enddo
  enddo
enddo
```

2. The function is read one value at a time. There are  $NW1 * NW2 * NW3 * NW4$  values. The program uses the values as follows:

```
do k4=1, nw4
  do k3=1, nw3
    do k2=1, nw2
      do k1=1, nw1
        read one value f(K1, K2, K3, K4)
      enddo
    enddo
  enddo
enddo
```

3. The formula can extend from field 2 through field 9 and must be comprised of the items listed previously.
4. This parameter may also be used to control the BOLT entry. For BOLT, the default for  $MMBOLTUS=1$  to achieve the same results as SOL 400. If the results appear to be backwards, set  $MMBOLTUS=-1$ . BOLT and MBOLTUS may not both be entered in the same model.

#### Remarks for SOL 400 only:

1. Independent variable types are typically specified for materials or contact. When material properties are varied through TABL3DX, the tables are honored only for elements with property extensions. For e.g., if TABL3D0 is used in conjunction with MATEP for a CHEXA mesh with PSOLID properties, then the elements should be given the PSLDN1 property extension for the table to be valid. Note that, with default property mapping (NLMOPTS,SPROPMAP,0), the property extensions will be added automatically based on the rules specified in Remark 9. of the NLMOPTS entry.
2. The strains and stresses identified as independent variables depend on the parameters flagged for large displacement.
  - a. For PARAM,LGDISP,-1, the strains and stresses are engineering quantities.
  - b. For PARAM,LGDISP,n ( $n > 0$ ) or with NLMOPTS,LRGSTRN,n ( $n > 0$ ), the strains are typically Logarithmic strains and the stresses are Cauchy Stresses.

**TABLE3D** Tabular Function with Three Variables

Specify a function of three variables for the CORD3G entry only.

Format:

|         |        |      |    |    |    |    |    |    |    |
|---------|--------|------|----|----|----|----|----|----|----|
| 1       | 2      | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
| TABLE3D | TID    | X0   | Y0 | Z0 | F0 |    |    |    |    |
|         | X1     | Y1   | Z1 | F1 | X2 | Y2 | Z2 | F2 |    |
|         | X3     | Y3   | Z3 | F3 | X4 | Y4 | Z4 | F4 |    |
|         | -etc.- | ENDT |    |    |    |    |    |    |    |

Example:

|         |      |     |     |       |      |      |      |       |  |
|---------|------|-----|-----|-------|------|------|------|-------|--|
| TABLE3D | 128  | 0.  | 0.  | 1.    |      |      |      |       |  |
|         | 7.   | 8.  | 9.  | 100.  | 12.  | 14.  | 11.  | 200.  |  |
|         | 17.  | 18. | 19. | 1100. | 112. | 114. | 111. | 1200. |  |
|         | ENDT |     |     |       |      |      |      |       |  |

| Describer | Meaning                              | Type        | Default  |
|-----------|--------------------------------------|-------------|----------|
| TID       | Table identification number.         | Integer > 0 | Required |
| X0,Y0,Z0  | Offset of the independent variables. | Real        | 0.0      |
| F0        | Offset of the dependent variables.   | Real        | 0.0      |
| Xi,Yi,Zi  | Independent variables.               | Real        | 0.0      |
| Fi        | Dependent variable.                  | Real        | 0.0      |

Remarks:

1. At least two continuation entries must be specified.
2. The value of the function at  $(x,y,z)$  is calculated as

$$f = \frac{\sum_{i=1}^4 \frac{Fi - F0}{d_i}}{\sum_{i=1}^4 \frac{1}{d_i}}$$

where  $f$  are the function values at the four points with the lowest value of

$$d_i^2 = (x - X0 - Xi)^2 + (y - Y0 - Yi)^2 + (z - Z0 - Zi)^2$$

## TABLED1      Dynamic Load Tabular Function, Form 1

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

|         |     |       |       |    |    |    |        |        |    |
|---------|-----|-------|-------|----|----|----|--------|--------|----|
| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9      | 10 |
| TABLED1 | TID | XAXIS | YAXIS |    |    |    |        |        |    |
|         | x1  | y1    | x2    | y2 | x3 | y3 | -etc.- | “ENDT” |    |

Example:

|         |      |     |     |     |     |     |      |  |  |
|---------|------|-----|-----|-----|-----|-----|------|--|--|
| TABLED1 | 32   |     |     |     |     |     |      |  |  |
|         | -3.0 | 6.9 | 2.0 | 5.6 | 3.0 | 5.6 | ENDT |  |  |

| Describer | Meaning                                                                                                                                   |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                                                                                                |
| XAXIS     | Specifies a linear or logarithmic interpolation for the x-axis. See Remarks 6. and 10. (Character: “LINEAR” or “LOG”; Default = “LINEAR”) |
| YAXIS     | Specifies a linear or logarithmic interpolation for the y-axis. See Remarks 6. and 10. (Character: “LINEAR” or “LOG”; Default = “LINEAR”) |
| xi, yi    | Tabular values. (Real)                                                                                                                    |
| “ENDT”    | Flag indicating the end of the table.                                                                                                     |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-147](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-147](#), the value of y at  $x = x3$  is  $y = (y3 + y4)/2$ . If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as  $y = \sqrt{y3y4}$ .
3. At least one continuation must be specified.
4. Any xi-yi pair may be ignored by placing the character string “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of the character string “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLED1 uses the algorithm

$$y = y_T(x)$$

where  $x$  is input to the table and  $y$  is returned. The table look-up is performed using interpolation within the table and extrapolation outside the table using the two starting or end points. See Figure 9-147. The algorithms used for interpolation or extrapolation are:

| AXIS   | YAXIS  | $y_f(x)$                                                                                             |
|--------|--------|------------------------------------------------------------------------------------------------------|
| LINEAR | LINEAR | $\frac{x_j - x}{x_j - x_i} y_i + \frac{x - x_i}{x_j - x_i} y_j$                                      |
| LOG    | LINEAR | $\frac{\ln(x_j/x)}{\ln(x_j/x_i)} y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} y_j$                          |
| LINEAR | LOG    | $\exp\left[\frac{x_j - x}{x_j - x_i} \ln y_i + \frac{x - x_i}{x_j - x_i} \ln y_j\right]$             |
| LOG    | LOG    | $\exp\left[\frac{\ln(x_j/x)}{\ln(x_j/x_i)} \ln y_i + \frac{\ln(x/x_i)}{\ln(x_j/x_i)} \ln y_j\right]$ |

where  $x_j$  and  $y_j$  follow  $x_i$  and  $y_i$ .

No warning messages are issued if table data is input incorrectly.

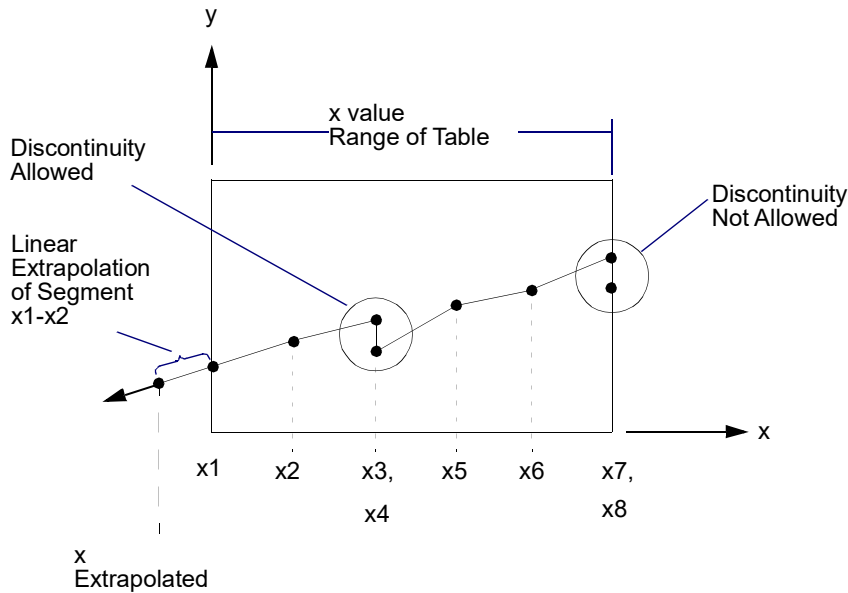


Figure 9-147 Example of Table Extrapolation and Discontinuity

- Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.

8. For frequency-dependent loads, xi is measured in cycles per unit time.
9. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value  $\leq 0$ .
10. LOG is not supported for SOL 600 or SOL 700. Fields 3 and 4 must be blank.
11. RC network solver only supports LINEAR type XAXIS and YAXIS for thermal analysis.
12. The X-Y plot information resulting from a dynamic response solution (like any response versus frequency in SOLs 108/111 or any response versus time in SOLs 109/112) can be generated in TABLED1 format by using the XYTRAN module via DMAP. The 6th parameter of this module, which is an integer value, can be used for this purpose. Details can be obtained by referring to the description of this module in the DMAP Programmer's Guide.

## TABLED2 Dynamic Load Tabular Function, Form 2

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLED2 | TID | X1 |    |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |      |       |     |      |      |     |     |     |  |
|---------|------|-------|-----|------|------|-----|-----|-----|--|
| TABLED2 | 15   | -10.5 |     |      |      |     |     |     |  |
|         | 1.0  | -4.5  | 2.0 | -4.2 | 2.0  | 2.8 | 7.0 | 6.5 |  |
|         | SKIP | SKIP  | 9.0 | 6.5  | ENDT |     |     |     |  |

| Describer | Meaning                                    |
|-----------|--------------------------------------------|
| TID       | Table identification number. (Integer > 0) |
| X1        | Table parameter. See Remark 6. (Real)      |
| xi, yi    | Tabular values. (Real)                     |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-148](#) discontinuities are allowed only between points  $x_2$  and  $x_7$ . Also if  $y$  is evaluated at a discontinuity, then the average value of  $y$  is used. In [Figure 9-148](#), the value of  $y$  at  $x = x_3$  is  $y = (y_3 + y_4)/2$ .
3. At least one continuation entry must be specified.
4. Any  $x_i$ - $y_i$  pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLED2 uses the algorithm

$$y = y_T(x - X1)$$

where  $x$  is input to the table and  $y$  is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-148](#). No warning messages are issued if table data is input incorrectly.

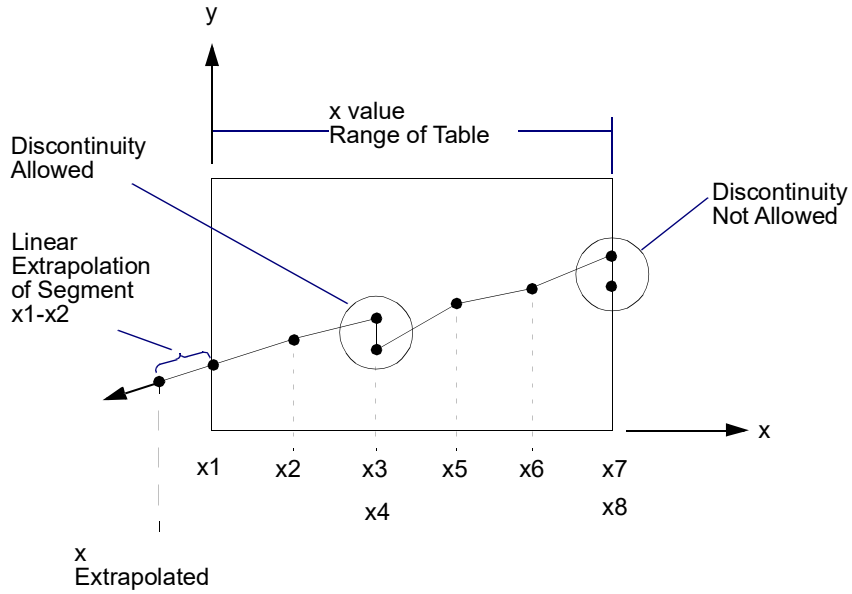


Figure 9-148 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads,  $X1$  and  $x_i$  are measured in cycles per unit time.

## TABLED3 Dynamic Load Tabular Function, Form 3

Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLED3 | TID | X1 | X2 |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |     |       |      |     |     |     |      |  |  |
|---------|-----|-------|------|-----|-----|-----|------|--|--|
| TABLED3 | 62  | 126.9 | 30.0 |     |     |     |      |  |  |
|         | 2.9 | 2.9   | 3.6  | 4.7 | 5.2 | 5.7 | ENDT |  |  |

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

|        |                                            |
|--------|--------------------------------------------|
| TID    | Table identification number. (Integer > 0) |
| X1, X2 | Table parameters. (Real; X2 ≠ 0.0)         |
| xi, yi | Tabular values. (Real)                     |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-149](#) discontinuities are allowed only between points  $x_2$  and  $x_7$ . Also if  $y$  is evaluated at a discontinuity, then the average value of  $y$  is used. In [Figure 9-149](#), the value of  $y$  at  $x = x_3$  is  $y = (y_3 + y_4)/2$ .
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLED3 uses the algorithm

$$y = y_T \left( \frac{x - X1}{X2} \right)$$

where  $x$  is input to the table and  $y$  is returned. The table look-up is performed using interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-149](#). No warning messages are issued if table data is input incorrectly.



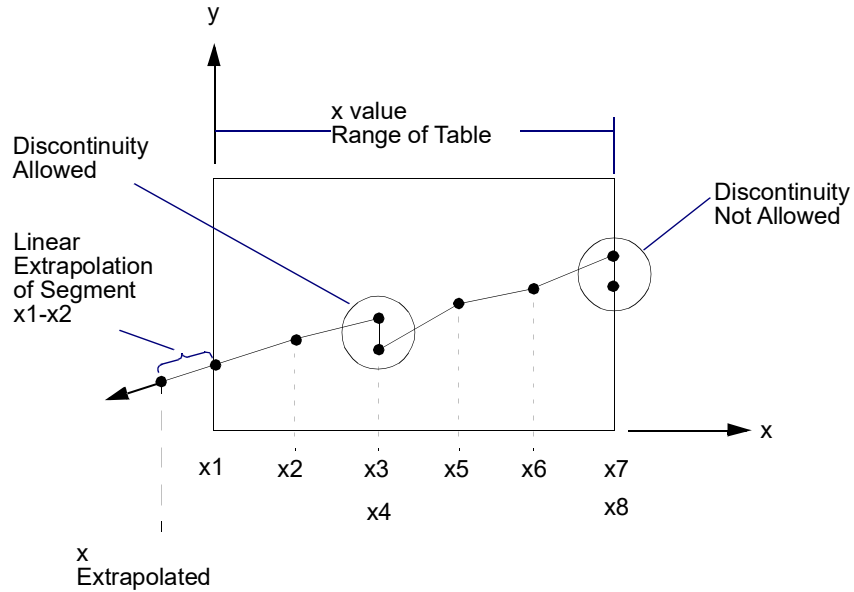


Figure 9-149 Example of Table Extrapolation and Discontinuity

7. Linear extrapolation is not used for Fourier transform methods. The function is zero outside the range of the table.
8. For frequency-dependent loads,  $X_1$ ,  $X_2$ , and  $x_i$  are measured in cycles per unit time.

## TABLED4 Dynamic Load Tabular Function, Form 4

Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table.

Format:

|         |     |    |    |    |    |    |        |   |    |
|---------|-----|----|----|----|----|----|--------|---|----|
| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
| TABLED4 | TID | X1 | X2 | X3 | X4 |    |        |   |    |
|         | A0  | A1 | A2 | A3 | A4 | A5 | -etc.- |   |    |

Example:

|         |      |         |        |     |        |      |  |  |  |
|---------|------|---------|--------|-----|--------|------|--|--|--|
| TABLED4 | 28   | 0.0     | 1.0    | 0.0 | 100.   |      |  |  |  |
|         | 2.91 | -0.0329 | 6.51-5 | 0.0 | -3.4-7 | ENDT |  |  |  |

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

|     |                                             |
|-----|---------------------------------------------|
| TID | Table identification number. (Integer > 0)  |
| Xi  | Table parameters. (Real; X2 ≠ 0.0; X3 < X4) |
| Ai  | Coefficients. (Real)                        |

Remarks:

1. At least one continuation entry must be specified.
2. The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
3. TABLED4 uses the algorithm

$$y = \sum_{i=0}^N A_i \left( \frac{x - X1}{X2} \right)^i$$

where  $x$  is input to the table,  $y$  is returned, and  $N$  is the number of pairs. Whenever  $x < X3$ , use  $X3$  for  $x$ ; whenever  $x > X4$ , use  $X4$  for  $x$ . There are  $N + 1$  entries in the table. There are no error returns from this table look-up procedure.

4. For frequency-dependent loads,  $x_i$  is measured in cycles per unit time.

## TABLED5      Dynamic Load Tabular Function, Form 5

Defines a value as a function of two variables for use in generating frequency-dependent and time-dependent dynamic loads.

Format:

| 1       | 2    | 3      | 4    | 5      | 6    | 7      | 8    | 9      | 10 |
|---------|------|--------|------|--------|------|--------|------|--------|----|
| TABLED5 | TID  |        |      |        |      |        |      |        |    |
|         | X(1) | TID(1) | X(2) | TID(2) | X(3) | TID(3) | X(4) | TID(4) |    |
|         | ...  | ...    | ENDT |        |      |        |      |        |    |

Example:

|         |     |     |      |     |      |     |      |  |  |
|---------|-----|-----|------|-----|------|-----|------|--|--|
| TABLED5 | 52  |     |      |     |      |     |      |  |  |
|         | 0.0 | 101 | 10.0 | 102 | 30.0 | 103 | ENDT |  |  |

| Describer | Meaning                                                                                                                   |
|-----------|---------------------------------------------------------------------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                                                                                |
| X(i)      | X value for the function specified by TID(i) (Real; no Default).                                                          |
| TID(i)    | ID of a TABLED1, TABLED2, TABLED3 or TABLED4 defining the function Y for the given value of X. (Integer > 0; no Default). |

Remarks:

1. This table returns a value that is a function of 2 variables  $f(x, y)$ . The first variable value  $x$  is specified on this entry as  $X(i)$ , the function versus  $y$  for the specified value for  $X(i)$  is specified on the referenced table TID(i).
2. When used with the NLRGAP entry to define a frequency dependent gap force as a function of penetration, the frequency values are input as  $X(i)$  and the variation of force vs. penetration is input on a series of TABLED1 entries referenced by the table values TID(i), one for each required change in properties with respect to frequency.
3. 2D linear interpolation is carried out for values of  $(x,y)$  not specified on the tabular entries.
4.  $X(i)$  values must be in ascending order.
5. TABLED5 Usage for frequency as function of temperature:

TABLED5 usage

|         |     |        |     |     |      |   |     |     |  |
|---------|-----|--------|-----|-----|------|---|-----|-----|--|
|         | ID  | E      | G   | NU  |      |   |     | GE  |  |
| MAT1    | 33  | 7.2+10 |     | .3  |      |   |     | .02 |  |
| MAT1F   | 33  | 110    | 111 | 112 |      |   |     | 200 |  |
| TABLED5 | 200 |        |     |     |      |   |     |     |  |
|         | 0.  | 3      | 40. | 4   | 100. | 5 | ... |     |  |
| TABLED1 | 3   | ...    |     |     |      |   |     |     |  |
| TABLED1 | 4   | ...    |     |     |      |   |     |     |  |
| TABLED1 | 5   | ...    |     |     |      |   |     |     |  |

Usage Example

E is dependent only on frequency, therefore points to a TABLED1 entry.

GE is both temperature and frequency dependent and therefore points to a TABLED5 entry

|                                               |     |        |        |        |      |        |      |      |  |
|-----------------------------------------------|-----|--------|--------|--------|------|--------|------|------|--|
|                                               |     | E      | G      | NU     |      |        |      | GE   |  |
| MAT1                                          | 1   | 7.2+10 | 2.8+10 | .3     |      | 2.22-5 |      | 0.02 |  |
| MAT1F                                         | 1   | 110    | 111    | 112    |      |        |      | 200  |  |
| TABLED1                                       | 110 |        |        |        |      |        |      |      |  |
|                                               | 10. | 7.2+10 | 200.   | 7.1+10 | 300. | 6.9+10 | ENDT |      |  |
| TABLED1                                       | 111 |        |        |        |      |        |      |      |  |
|                                               | 10. | 2.8+10 | 200.   | 2.7+10 |      | 2.6+10 | ENDT |      |  |
| TABLED1                                       | 112 |        |        |        |      |        |      |      |  |
|                                               | 10. | .3     | 200.   | .3     |      | .3     | ENDT |      |  |
| GE has frequency as a function of temperature |     |        |        |        |      |        |      |      |  |
| TABLED5                                       | 200 |        |        |        |      |        |      |      |  |
|                                               | 0.  | 3      | 40.    | 4      | 100. | 5      | ENDT |      |  |
| TABLED1                                       | 3   |        |        |        |      |        |      |      |  |
|                                               | 10. | 0.02   | 200.   |        |      | 0.025  | ENDT |      |  |

|         |     |       |      |      |      |       |      |  |  |
|---------|-----|-------|------|------|------|-------|------|--|--|
| TABLED1 | 4   |       |      |      |      |       |      |  |  |
|         | 10. | 0.025 | 200. |      |      | 0.03  | ENDT |  |  |
| TABLED1 | 5   |       |      |      |      |       |      |  |  |
|         | 10. | 0.03  | 200. | 0.04 | 300. | 0.035 | ENDT |  |  |

TABLED1 units are x=frequency, y=material value

TABLED5 input is numerical temperature - frequency table ID

For an element with average temperature of 15.0 degrees the GE value will be selected from TABLED1 ID=3;

For an element with average temperature of 30.0 degrees the GE value will be selected from TABLED1 ID=4;

For an element with average temperature of 20.0 degrees the GE value will be selected from TABLED1 ID=3;

6. TABLED5 in Solution Sequence SOL108, SOL111, SO200, or SOL400 with ANALYSIS=DFREQ or MFREQ:

Example (1): TABLED5 specification using TEMP(INIT) or TEMP(MATE) entry, no TEMP(LOAD) in Case Control section.

```
SOL 108
...
TEMP (INIT) =5
BEGIN BULK
...
TEMP, 5, ...
...
ENDDATA
```

The temperatures specified by TEMP(INIT) will be used for TABLED5 look up

Example (2): TABLED5 specification using TEMP(LOAD) entry.

```
SOL 108
...
TEMP (INIT) =5
SUBCASE 1
TEMP (LOAD) =12
...
```

```
BEGIN BULK
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up.

User is reminded that for dynamic solutions sequences, the above will not apply thermal loading unless there is a DLOAD entry pointing to an RLOAD1 or RLOAD2 referencing TEMP(LOAD) with ID=12 specified.

Example (3): TABLED5 specification using TEMP(LOAD) entry and thermal loading desired.

```
SOL 108
...
TEMP (INIT) =5
SUBCASE 1
TEMP (LOAD) =12
DLOAD = 50
...
BEGIN BULK
...
RLOAD, 50, 12, , , 1.0, , LOAD
...
TEMP, 5, ...
TEMP, 12, ...
...
ENDDATA
```

The temperatures specified by TEMP(LOAD) will be used for TABLED5 look up and will also be used to compute thermal loads in elements.

## TABLEHT Heat Transfer Coefficient Table or Nusselt Number Table with Two Variables

Specifies a function of two variables for convection heat transfer coefficient or Nusselt number.

Format:

| 1       | 2   | 3    | 4  | 5    | 6  | 7     | 8 | 9 | 10 |
|---------|-----|------|----|------|----|-------|---|---|----|
| TABLEHT | TID |      |    |      |    |       |   |   |    |
|         | x1  | TID1 | x2 | TID2 | x3 | -etc. |   |   |    |

Example:

|         |      |     |      |     |      |     |      |  |  |
|---------|------|-----|------|-----|------|-----|------|--|--|
| TABLEHT | 85   |     |      |     |      |     |      |  |  |
|         | 10.0 | 101 | 25.0 | 102 | 40.0 | 110 | ENDT |  |  |

| Describer | Meaning                                                        |
|-----------|----------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                     |
| xi        | Independent variables. (Real)                                  |
| TIDi      | Table identification numbers of TABLEH1 entries. (Integer > 0) |

Remarks:

1. xi must be listed in ascending order.
2. At least one continuation entry must be present.
3. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by PCONV entries that define free convection boundary condition properties. It computes heat transfer coefficient or Nusselt number, depending on the FTYPE of the associated PCONV entry.
  - If FTYPE=1 in PCONV entry: compute heat transfer coefficient.
  - If FTYPE=2 in PCONV entry: compute Nusselt number.

**TABLEH1** Heat Transfer Coefficient Table or Nusselt Number Table, Form 1

Defines a tabular function referenced by TABLEHT for convection heat transfer coefficient or Nusselt number.

Format:

|         |     |    |    |    |    |        |   |   |    |
|---------|-----|----|----|----|----|--------|---|---|----|
| 1       | 2   | 3  | 4  | 5  | 6  | 7      | 8 | 9 | 10 |
| TABLEH1 | TID |    |    |    |    |        |   |   |    |
|         | y1  | f1 | y2 | f2 | y3 | -etc.- |   |   |    |

Example:

|         |      |      |      |      |       |      |      |  |  |
|---------|------|------|------|------|-------|------|------|--|--|
| TABLEH1 | 123  |      |      |      |       |      |      |  |  |
|         | 50.0 | 5.23 | 75.0 | 3.76 | 110.0 | 0.97 | ENDT |  |  |

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

- |     |                                            |
|-----|--------------------------------------------|
| TID | Table identification number. (Integer > 0) |
| yi  | Independent variables. (Real)              |
| fi  | Dependent variable. (Real)                 |

Remarks:

1. yi must be listed in ascending order.
2. At least one continuation entry must be present.
3. Any yi-fi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
4. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
5. TABLEH1 is used to input a curve in the form of
 
$$f = f(y)$$
 where  $y$  is input to the table and  $f$  is returned. The table look-up is performed using linear interpolation within the table and is evaluated at the starting or end point outside the table. No warning messages are issued if table data is input incorrectly.
6. Discontinuities are not recommended and may lead to unstable results.



## TABLE1 Specifies a Table of Amplitude vs Pseudo-Time for Static Loads Specified in LDTABL Entries

Specifies a table of amplitude vs pseudo-time for static loads specified in LDTABL entries SOL 600 only.

### Format:

| 1      | 2   | 3  | 4  | 5  | 6  | 7  | 8    | 9     | 10 |
|--------|-----|----|----|----|----|----|------|-------|----|
| TABLE1 | TID |    |    |    |    |    |      |       |    |
|        | X1  | Y1 | X2 | Y2 | X3 | Y3 | etc. | "ENT" |    |

### Example:

|        |     |     |     |     |     |     |      |  |  |
|--------|-----|-----|-----|-----|-----|-----|------|--|--|
| TABLE1 | 101 |     |     |     |     |     |      |  |  |
|        | 0.0 | 0.0 | 0.5 | 1.0 | 1.0 | 0.0 | ENDT |  |  |

| Describer | Meaning                                                |
|-----------|--------------------------------------------------------|
| TID       | Table identification number. (Integer > 0; no Default) |
| Xi, Yi    | Tabular values. (Real; no Default)                     |
| "ENDT"    | Flag indicating end of the table. (Character)          |

### Remarks:

1. TABLE1 can only be used in SOL 600.
2. TABLED1 may be used instead of TABLEL1, however, XAXIS and YAXIS must be LINEAR or blank.
3. The TID must be unique among all TABLEL1's and TABLED1's.
4. The string "SKIP" used in TABLED1 may not be used in TABLEL1.
5. Log X and Y is not available in TABLEL1.
6. No blank fields are allowed starting with the second field of the first continuation line until the ENDT string.
7. There must be at least two points in the table.

**TABLEM1** Material Property Table, Form 1

Defines a tabular function for use in generating temperature-dependent material or fatigue related properties.

Format:

|         |     |       |       |    |    |    |        |        |    |
|---------|-----|-------|-------|----|----|----|--------|--------|----|
| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9      | 10 |
| TABLEM1 | TID | XAXIS | YAXIS |    |    |    |        |        |    |
|         | x1  | y1    | x2    | y2 | x3 | y3 | -etc.- | “ENDT” |    |

Example:

|         |      |     |     |     |     |     |      |  |  |
|---------|------|-----|-----|-----|-----|-----|------|--|--|
| TABLEM1 | 32   |     |     |     |     |     |      |  |  |
|         | -3.0 | 6.9 | 2.0 | 5.6 | 3.0 | 5.6 | ENDT |  |  |

| Describer | Meaning                                                                                                                          |
|-----------|----------------------------------------------------------------------------------------------------------------------------------|
| TID       | Table identification number. See Remark 8. (Integer > 0 or Integer < 0)                                                          |
| XAXIS     | Specifies a linear or logarithmic interpolation for the x-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”) See Remark 9. |
| YAXIS     | Specifies a linear or logarithmic interpolation for the y-axis. (Character: “LINEAR” or “LOG”; Default = “LINEAR”) See Remark 9. |
| xi, yi    | Tabular values. (Real)                                                                                                           |
| “ENDT”    | Flag indicating the end of the table.                                                                                            |

Remarks:

- xi must be in either ascending or descending order, but not both.
- Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 9-150 discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In Figure 9-150, the value of y at x = x3 is  $y = (y3 + y4)/2$ .
- At least one continuation entry must be specified.
- Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
- The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLEM1 uses the algorithm  

$$y = y_T(x) \quad (\text{for heat transfer, see Remark 7.})$$

where  $x$  is input to the table and  $y$  is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See Figure 9-150. No warning messages are issued if table data is input incorrectly.

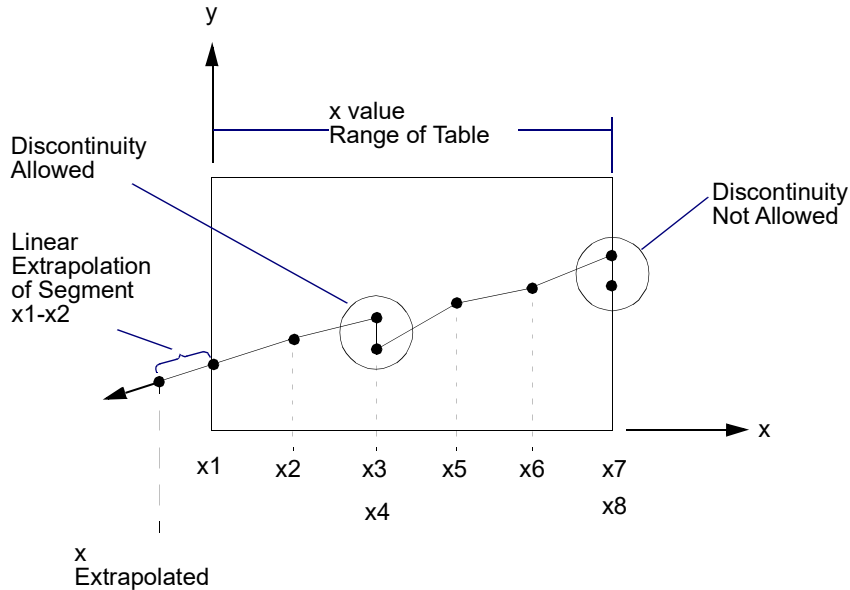


Figure 9-150 Example of Table Extrapolation and Discontinuity

- For Nastran heat transfer, the TABLEM1 assumes

$$y = zy_T(x)$$

where  $x$  is input to the table,  $y$  is returned and  $z$  is supplied from MAT4 or MT5 entries.

- A negative TID is used to associate thermal strain  $\epsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the T(Ai) fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(Ai)}$  value will be changed to  $|ID_{T(Ai)}| + 100000000$ .
- Logarithmic XAXIS and YAXIS is only recognized when specifying S-N curves referenced by MATFTG entries for TYPE=TABLE. All other uses use linear and ignore these fields.
- TABLEM1 in RESTART job will cause recalculation of eigenvalues even in the cases where TABLEM1 does not change stiffness and/or mass matrix. An example is SOL 111 RESTART for fatigue analysis with TABLEM1 referenced on MATFTG.

## TABLEM2 Material Property Table, Form 2

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLEM2 | TID | X1 |    |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |      |       |     |      |      |     |     |     |  |
|---------|------|-------|-----|------|------|-----|-----|-----|--|
| TABLEM2 | 15   | -10.5 |     |      |      |     |     |     |  |
|         | 1.0  | -4.5  | 2.0 | -4.5 | 2.0  | 2.8 | 7.0 | 6.5 |  |
|         | SKIP | SKIP  | 9.0 | 6.5  | ENDT |     |     |     |  |

| Describer | Meaning                                                                 |
|-----------|-------------------------------------------------------------------------|
| TID       | Table identification number. See Remark 7. (Integer > 0 or Integer < 0) |
| X1        | Table parameter. (Real)                                                 |
| xi, yi    | Tabular values. (Real)                                                  |

Remarks:

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-151](#), discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-151](#), the value of y at x = x3 is  $y = (y3 + y4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
6. TABLEM2 uses the algorithm

$$y = zy_T(x - X1)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-151](#). No warning messages are issued if table data is input incorrectly.

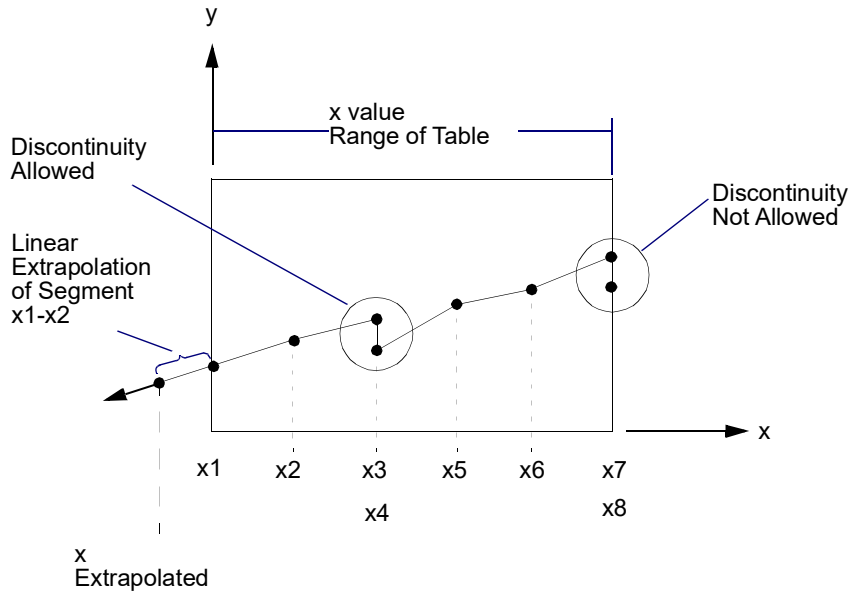


Figure 9-151 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain  $\epsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 100000000$ .

**TABLEM3** Material Property Table, Form 3

Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

|         |     |    |    |    |    |    |        |   |    |
|---------|-----|----|----|----|----|----|--------|---|----|
| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
| TABLEM3 | TID | X1 | X2 |    |    |    |        |   |    |
|         | x1  | y1 | x2 | y2 | x3 | y3 | -etc.- |   |    |

Example:

|         |     |       |      |     |     |     |      |  |  |
|---------|-----|-------|------|-----|-----|-----|------|--|--|
| TABLEM3 | 62  | 126.9 | 30.0 |     |     |     |      |  |  |
|         | 2.9 | 2.9   | 3.6  | 4.7 | 5.2 | 5.7 | ENDT |  |  |

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

- TID Table identification number. See Remark 7. (Integer > 0 or Integer < 0)
- X1, X2 Table parameters. See Remark 6. (Real; X2 ≠ 0.0)
- xi, yi Tabular values. (Real)

Remarks:

1. Tabular values for xi must be specified in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in Figure 9-152 discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In Figure 9-152, the value of y at x = x3 is  $y = (y3 + y4)/2$ .
3. At least one continuation entry must be specified.
4. Any xi-yi pair may be ignored by placing "SKIP" in either of the two fields.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABLEM3 uses the algorithm

$$y = zy_T \left( \frac{x - X1}{X2} \right)$$

where x is input to the table, y is returned and z is supplied from the MATi entry. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See Figure 9-152. No warning messages are issued if table data is input incorrectly.



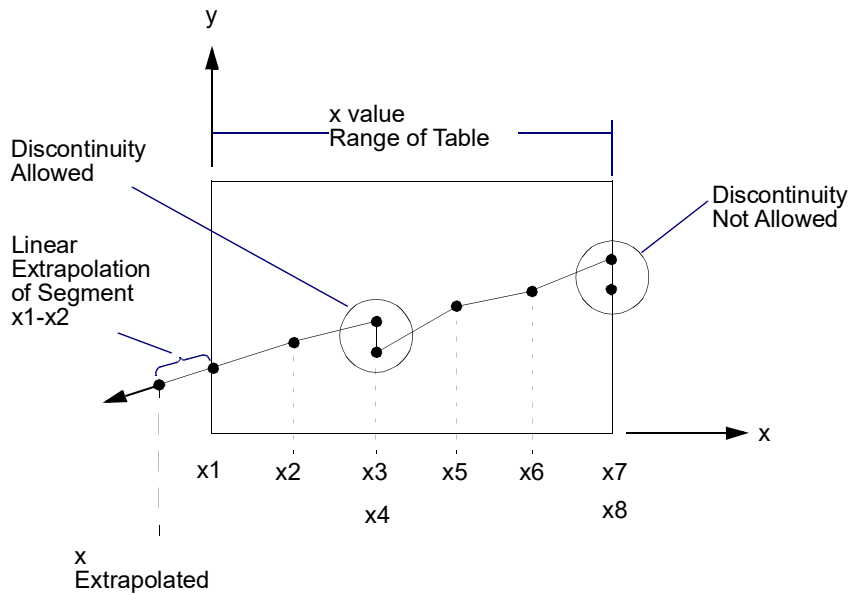


Figure 9-152 Example of Table Extrapolation and Discontinuity

7. A negative TID is used to associate thermal strain  $\epsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the  $T(A_i)$  fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(A_i)}$  value will be changed to  $|ID_{T(A_i)}| + 100000000$ .

## TABLEM4 Material Property Table, Form 4

Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8      | 9 | 10 |
|---------|-----|----|----|----|----|----|--------|---|----|
| TABLEM4 | TID | X1 | X2 | X3 | X4 |    |        |   |    |
|         | A0  | A1 | A2 | A3 | A4 | A5 | -etc.- |   |    |

Example:

|         |      |         |        |     |        |      |  |  |  |
|---------|------|---------|--------|-----|--------|------|--|--|--|
| TABLEM4 | 28   | 0.0     | 1.0    | 0.0 | 100.   |      |  |  |  |
|         | 2.91 | -0.0329 | 6.51-5 | 0.0 | -3.4-7 | ENDT |  |  |  |

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

|     |                                                                         |
|-----|-------------------------------------------------------------------------|
| TID | Table identification number. See Remark 4. (Integer > 0 or Integer < 0) |
| Xi  | Table parameters. (Real; X2 ≠ 0.0; X3 < X4)                             |
| Ai  | Coefficients. (Real)                                                    |

Remarks:

- At least one continuation entry must be specified.
- The end of the table is indicated by the existence of “ENDT” in the field following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag “ENDT”.
- TABLEM4 uses the algorithm

$$y = z \sum_{i=0}^N Ai \left( \frac{x - X1}{X2} \right)^i$$

where  $x$  is input to the table,  $y$  is returned and  $z$  is supplied from the MATi entry. Whenever  $x < X3$ , use  $X3$  for  $x$ ; whenever  $x > X4$ , use  $X4$  for  $x$ . There are  $N + 1$  entries in the table. There are no error returns from this table look-up procedure.

- A negative TID is used to associate thermal strain  $\epsilon(T)$  ordinate values instead of coefficient of thermal expansion ordinate values to the T(Ai) fields of MATT1, MATT2, or MATT8 Bulk Data entries as described in the remarks of those entries. Internally to Nastran, a negative  $ID_{T(Ai)}$  value will be changed to  $|ID_{T(Ai)}| + 100000000$ .



## TABLES1 Material Property Table, Form 1

Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

**Format:**

| 1       | 2   | 3    | 4  | 5  | 6  | 7  | 8      | 9      | 10 |
|---------|-----|------|----|----|----|----|--------|--------|----|
| TABLES1 | TID | TYPE |    |    |    |    |        |        |    |
|         | x1  | y1   | x2 | y2 | x3 | y3 | -etc.- | “ENDT” |    |

**Example:**

|         |     |     |     |        |     |        |      |  |  |
|---------|-----|-----|-----|--------|-----|--------|------|--|--|
| TABLES1 | 32  |     |     |        |     |        |      |  |  |
|         | 0.0 | 0.0 | .01 | 10000. | .02 | 15000. | ENDT |  |  |

| Describer | Meaning                                                                                        |
|-----------|------------------------------------------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                                                     |
| TYPE      | Flag to define type of the stress-strain curve. See Remark 10. (Integer = 1 or 2; Default = 1) |
| xi, yi    | Tabular values. (Real)                                                                         |
| “ENDT”    | Flag indicating the end of the table.                                                          |

**Remarks:**

1. xi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-153](#) discontinuities are allowed only between points x2 through x7. Also, if y is evaluated at a discontinuity, then the average value of y is used. In [Figure 9-153](#), the value of y at x = x3 is  $y = (y3 + y4)/2$ .
3. At least one continuation entry must be present.
4. Any xi-yi pair may be ignored by placing “SKIP” in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of “ENDT” in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
6. TABLES1 is used to input a curve in the form of

$$y = y_T(x)$$

where x is input to the table and y is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See [Figure 9-153](#). No warning messages are issued if table data is input incorrectly.

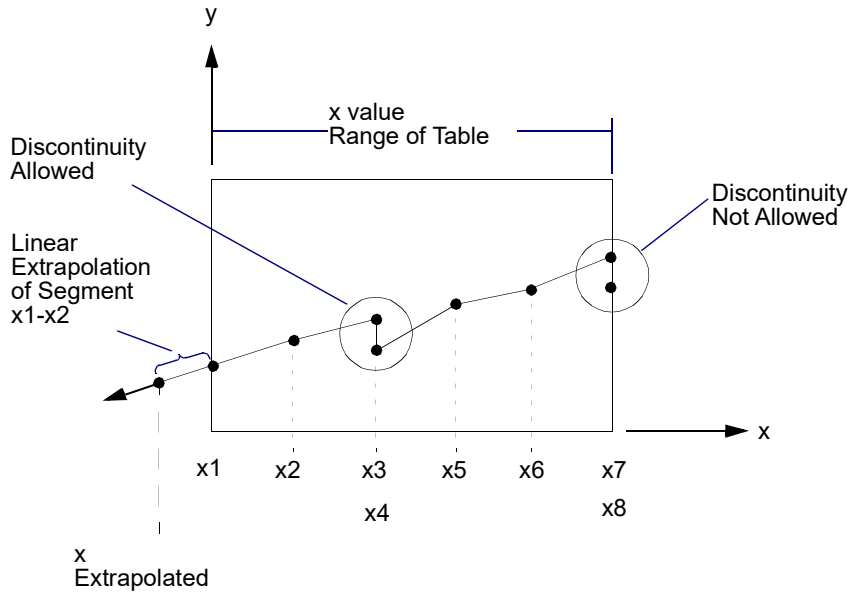


Figure 9-153 Example of Table Extrapolation and Discontinuity

7. Discontinuities are not recommended and may lead to unstable results. Discontinuities are not allowed in nonlinear solution sequences.
8. For SOL 600, general temperature-dependent stress vs. plastic strain curves may be entered using a combination of TABLEST and TABLES1 entries. Each TABLES1 entry is at a constant temperature. All entries must be in the form of stress vs. plastic strain using the stress and strain measures to be incorporated into the analysis. All sets of stress-strain values for a particular TABLES1 entry must be at the same temperature. One set is required for the lowest temperature in the model and another at or above the highest temperature in the model.
9. For SOL 600, the stress and strain values entered here depend on the stress and strain measures selected for the analysis. In addition, the strain is controlled using PARAM,MRTABLS1 which provides several methods of converting an engineering stress-strain curve to a stress vs. plastic strain curve (see MRTABLS1 in the Parameters Section).
10. For SOL 400, TYPE denotes the type of stress-strain curve; 1 - Cauchy (true) stress vs. total true strain; and 2 - Cauchy (true) stress vs. plastic true strain. For MATS1 Bulk Data entry, only TYPE = 1 can be used. A user fatal error will be issued if TYPE = 2 is used. For MATEP Bulk Data entry both TYPE = 1 and 2 can be used.

## TABLEST Material Property Temperature-Dependence Table

Specifies the material property tables for nonlinear elastic temperature-dependent materials.

Format:

|         |     |      |    |      |    |        |   |   |    |
|---------|-----|------|----|------|----|--------|---|---|----|
| 1       | 2   | 3    | 4  | 5    | 6  | 7      | 8 | 9 | 10 |
| TABLEST | TID |      |    |      |    |        |   |   |    |
|         | T1  | TID1 | T2 | TID2 | T3 | -etc.- |   |   |    |

Example:

|         |       |    |       |    |      |  |  |  |  |
|---------|-------|----|-------|----|------|--|--|--|--|
| TABLEST | 101   |    |       |    |      |  |  |  |  |
|         | 150.0 | 10 | 175.0 | 20 | ENDT |  |  |  |  |

| Describer | Meaning                                                        |
|-----------|----------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                     |
| Ti        | Temperature values. (Real)                                     |
| TIDi      | Table identification numbers of TABLES1 entries. (Integer > 0) |

Remarks:

1. TIDi must be unique with respect to all TABLES1 and TABLEST table identification numbers.
2. Temperature values must be listed in ascending order.
3. The end of the table is indicated by the existence of ENDT in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag ENDT.
4. This table is referenced only by MATS1 entries that define nonlinear elastic (TYPE = "NLELAST") materials.
5. For SOL 600, this entry provides IDs of TABLES1 curves as a function of temperature for use with Marc's AF\_flowmat. The strains are plastic strain for all curves entered. The first curve must be entered at the lowest temperature encountered in the analysis run. Curves must be defined that equal or exceed the maximum temperature encountered in the run.

## TABLFTG Fatigue Loading Tabular Data

Defines tabular data for specifying fatigue cyclic loading variation.

Format:

| 1       | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9      | 10 |
|---------|-----|----|----|----|----|----|----|--------|----|
| TABLFTG | TID |    |    |    |    |    |    |        |    |
|         | y1  | y2 | y3 | y4 | y5 | y6 | y7 | "ENDT" |    |

Example:

|         |       |      |     |     |      |  |  |  |  |
|---------|-------|------|-----|-----|------|--|--|--|--|
| TABLFTG | 1     |      |     |     |      |  |  |  |  |
|         | 0.000 | -1.0 | 1.0 | 0.0 | ENDT |  |  |  |  |

| Describer | Meaning                                                  |
|-----------|----------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)               |
| yi        | Y value of each point in the time history curve. (Real). |
| "ENDT"    | Flag indicating the end of the table.                    |

Remarks:

1. The TABLFTG is referenced by a FTGLOAD entry.
2. The x-values are assumed to be in ascending order. For rainflow cycle counting purposes the actual x values are inconsequential.
3. For modal analysis using SOL 103, this would define the modal participation factors for a particular mode.

## TABLRPC

## Dynamic Load Tabular Function Referencing Channel Data File

Defines a tabular function for use in generating time-dependent loads from an externally defined channel data file.

## Format:

| 1       | 2               | 3               | 4               | 5               | 6               | 7               | 8      | 9      | 10 |
|---------|-----------------|-----------------|-----------------|-----------------|-----------------|-----------------|--------|--------|----|
| TABLRPC | TID             | XAXIS           | YAXIS           | UID             | TYPE            | CHAN            | TOTIM  | PUNCH  |    |
|         | x <sub>i1</sub> | x <sub>j1</sub> | x <sub>i2</sub> | x <sub>j2</sub> | x <sub>i3</sub> | x <sub>j3</sub> | -etc.- | "ENDT" |    |

## Examples:

|         |                  |  |  |    |     |   |  |  |  |
|---------|------------------|--|--|----|-----|---|--|--|--|
| TABLRPC | 32               |  |  | 33 | RPC | 5 |  |  |  |
| UDNAME  | 33               |  |  |    |     |   |  |  |  |
|         | C:\myrpcfile.rsp |  |  |    |     |   |  |  |  |

|         |                  |      |      |      |      |  |  |  |  |
|---------|------------------|------|------|------|------|--|--|--|--|
| TABLRPC | 32               |      |      | 33   | DAC  |  |  |  |  |
|         | 0.0              | 20.0 | 35.0 | 60.0 | ENDT |  |  |  |  |
| UDNAME  | 33               |      |      |      |      |  |  |  |  |
|         | C:\myrpcfile.dac |      |      |      |      |  |  |  |  |

|         |                  |      |      |      |      |   |  |  |  |
|---------|------------------|------|------|------|------|---|--|--|--|
| TABLRPC | 32               |      |      | -33  |      | 5 |  |  |  |
|         | 0.0              | 20.0 | 35.0 | 60.0 | ENDT |   |  |  |  |
| UDNAME  | 33               |      |      |      |      |   |  |  |  |
|         | C:\myrpcfile.rsp |      |      |      |      |   |  |  |  |

| Descriptor | Meaning                                                                                                                                                                             |
|------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| TID        | Table identification number. (Integer > 0, no default)                                                                                                                              |
| XAXIS      | Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")                                                                  |
| YAXIS      | Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR")                                                                  |
| UID        | Identification number of a UDNAME entry to specify the external file that defines the x-y pair values that define the actual tabular function. Required. (Integer != 0, no default) |
| TYPE       | Type of external file to specify via UID field. (Character: "RPC", or "DAC"; Default = "RPC")                                                                                       |
| CHAN       | Channel number to read for Y (load) values from "RPC" files. Leave blank for "DAC" files. (Integer > 0, Default = 1)                                                                |
| TOTIM      | Total time of the signal defined in channel CHAN. Optional. (Real > 0., Default=Blank). See remarks below.                                                                          |

| Describer | Meaning                                                                                                                                                                                                                                                                                         |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| PUNCH     | Specify whether equivalent TABLED1 entries containing the actual x-y pairs should be written to the PUNCH file. (Character: "YES" or "NO"; Default= "NO")                                                                                                                                       |
| $x_i/x_j$ | x-y pair Filter mechanism. The given $(x_i, x_j)$ pairs reflect the start and end x-values for inclusion from the external file; or if UID is negative, the given pairs reflect the start and end x-values to exclude from reading from the external file. See remarks below. Optional. (Real). |
| "ENDT"    | Flag indicating the end of the table.                                                                                                                                                                                                                                                           |

Remarks:

1. The TABLRPC entry can be referenced by any entry that can reference a TABLED1. The TABLRPC is internally converted to a TABLED1 entry using the channel data specified. This internal TABLED1 entry can be written to the punch (.pch) file if the PUNCH field is set to YES.
2. The CHANnel specified contains the Y (load) data. The X (time) data for each point is automatically extracted from the channel file at each point to create the x-y pairs for a TABLED1 entry. The TOTIM (total time) of the signal can be optionally supplied, in which case the time increment of each point is calculated as TOTIM / (NPNTS-1) where NPNTS are the number of signal points. This will override any time specifications for the points from the original channel file.
3.  $x_i/x_j$  field pairs are purely optional. If not needed, no continuation lines should be present, in which case the entire signal defined in the specified CHANnel is taken.
4. The  $x_i/x_j$  field pairs represent sections of the channel to read and must be in the order in which they appear in the external file. For example, to read from only  $x=0.0$  to  $x=1.0$ , specify  $x_1=0.0$  and  $x_j=1.0$  and  $x_2=ENDT$ . If the first  $x_i$  is left blank, reading begins at the first data point. If the last  $x_j$  value is left blank, then the rest of the data points are read to the end of the channel. If a specified  $x_i$  or  $x_j$  value does not exist, only the points in between are retained or excluded.
5. Any  $x_i/x_j$  field pair may be ignored by placing the character string "SKIP" in either of the two fields.
6. The end of the table is indicated by the existence of the character string "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT"
7. As this data is converted to a TABLED1 entry, all comments pertaining to the TABLED1 entry are also valid.

## TABLUDS

## User Defined Failure For Nonlinear Explicit Analysis.

Specifies that a user routine is being used to define an arbitrary function. Use in SOL700 only.

Format:

| 1       | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|---|---|---|---|---|----|
| TABLUDS | TID | GROUP | UNAME |   |   |   |   |   |    |

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myfunc 'SCA.MDSolver.Obj.Uds.Dytran.Loads '
```

*In Bulk Data:*

| 1       | 2  | 3      | 4      | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|---|---|---|---|---|----|
| TABLUDS | 12 | myfunc | EXFUNC |   |   |   |   |   |    |

| Describer | Meaning                                                                                    |
|-----------|--------------------------------------------------------------------------------------------|
| TID       | Unique output number. (Integer > 0; Required)                                              |
| GROUP     | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME     | User subroutine name associated with the entry. (Character; default=EXFUNC)                |

Remarks:

1. Since tables and user-defined functions belong to the same group, the table numbers must be unique.
2. UNAME can be:

| Subroutine Name | Function                       |
|-----------------|--------------------------------|
| EXFUNC          | Standard user defined function |

## TABRND1 Power Spectral Density Table

Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.

Format:

|         |     |       |       |    |    |    |        |   |    |
|---------|-----|-------|-------|----|----|----|--------|---|----|
| 1       | 2   | 3     | 4     | 5  | 6  | 7  | 8      | 9 | 10 |
| TABRND1 | TID | XAXIS | YAXIS |    |    |    |        |   |    |
|         | f1  | g1    | f2    | g2 | f3 | g3 | -etc.- |   |    |

Example:

|         |     |        |     |        |      |  |  |  |  |
|---------|-----|--------|-----|--------|------|--|--|--|--|
| TABRND1 | 3   |        |     |        |      |  |  |  |  |
|         | 2.5 | .01057 | 2.6 | .01362 | ENDT |  |  |  |  |

| Describer | Meaning                                                                                                            |
|-----------|--------------------------------------------------------------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                                                                         |
| XAXIS     | Specifies a linear or logarithmic interpolation for the x-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| YAXIS     | Specifies a linear or logarithmic interpolation for the y-axis. (Character: "LINEAR" or "LOG"; Default = "LINEAR") |
| fi        | Frequency value in cycles per unit time. (Real ≥ 0.0)                                                              |
| gi        | Power spectral density. (Real)                                                                                     |

Remarks:

1. The fi must be in either ascending or descending order, but not both.
2. Discontinuities may be specified between any two points except the two starting points or two end points. For example, in [Figure 9-154](#) discontinuities are allowed only between points f2 through f7. Also, if g is evaluated at a discontinuity, then the average value of g is used. In [Figure 9-154](#), the value of g at f = f3 is  $g = (g3 + g4)/2$ . If the y-axis is a LOG axis then the jump at the discontinuity is evaluated as  $y = \sqrt{y3y4}$ .
3. At least two entries must be present.
4. Any fi-gi pair may be ignored by placing "SKIP" in either of the two fields used for that entry.
5. The end of the table is indicated by the existence of "ENDT" in either of the two fields following the last entry. An error is detected if any continuations follow the entry containing the end-of-table flag "ENDT".
6. TABRND1 uses the algorithm

$$g = g_T(f)$$



where  $f$  is input to the table and  $g$  is returned. The table look-up is performed using linear interpolation within the table and linear extrapolation outside the table using the two starting or end points. See Figure 9-154. No warning messages are issued if table data is input incorrectly.

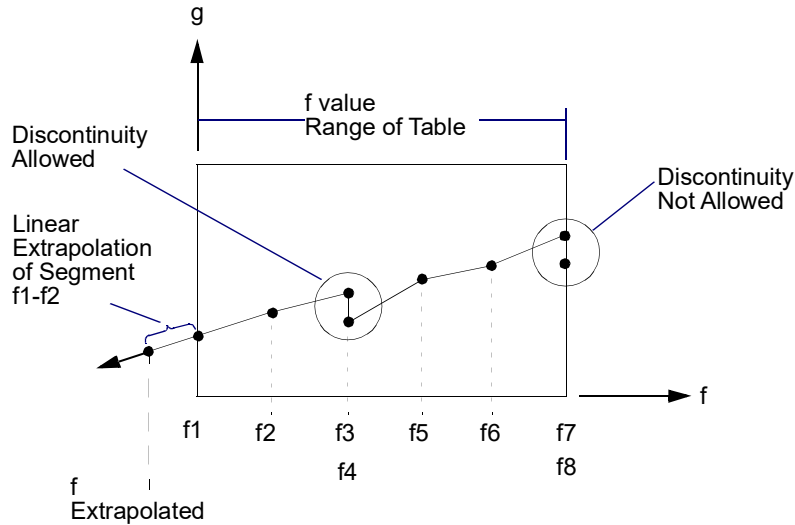


Figure 9-154 Example of Table Extrapolation and Discontinuity

7. For auto spectral density, the value of  $g$  returned must be greater than or equal to zero, as shown in Remark 6.
8. Tabular values on an axis if XAXIS or YAXIS = LOG must be positive. A fatal message will be issued if an axis has a tabular value  $\leq 0$ .
9. The algorithms used are:

| XAXIS  | YAXIS  | $f(x)$                                                                                                             |
|--------|--------|--------------------------------------------------------------------------------------------------------------------|
| LINEAR | LINEAR | $\frac{f_{i+1}-f}{f_{i+1}-f_i}g_i + \frac{f-f_i}{f_{i+1}-f_i}g_{i+1}$                                              |
| LOG    | LINEAR | $\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}g_{i+1}$                          |
| LINEAR | LOG    | $\exp\left[\frac{f_{i+1}-f}{f_{i+1}-f_i}\ln g_i + \frac{f-f_i}{f_{i+1}-f_i}\ln g_{i+1}\right]$                     |
| LOG    | LOG    | $\exp\left[\frac{\ln(f_{i+1}/f)}{\ln(f_{i+1}/f_i)}\ln g_i + \frac{\ln(f/f_i)}{\ln(f_{i+1}/f_i)}\ln g_{i+1}\right]$ |

where  $f_i < f < f_{i+1}$ .

10. If Modules are present then this entry may only be specified in the main Bulk Data section.

## TABRNDG Gust Power Spectral Density

Defines the power spectral density (PSD) of a gust for aeroelastic response analysis.

Format:

| 1       | 2   | 3    | 4   | 5  | 6 | 7 | 8 | 9 | 10 |
|---------|-----|------|-----|----|---|---|---|---|----|
| TABRNDG | TID | TYPE | L/U | WG |   |   |   |   |    |

Example:

|         |      |   |     |    |  |  |  |  |  |
|---------|------|---|-----|----|--|--|--|--|--|
| TABRNDG | 1020 | 1 | 1.3 | .1 |  |  |  |  |  |
|---------|------|---|-----|----|--|--|--|--|--|

| Describer | Meaning                                                                              |
|-----------|--------------------------------------------------------------------------------------|
| TID       | Table identification number. (Integer > 0)                                           |
| TYPE      | PSD type: von Karman (TYPE = 1) or Dryden model (TYPE = 2). (Integer = 1 or 2)       |
| L/U       | Scale of turbulence divided by velocity (units of time). See L/U in Remark 2. (Real) |
| WG        | Root-mean-square gust velocity. (Real)                                               |

Remarks:

1. This entry must be referenced by a RANDPS entry.
2. The power spectral density is given by

$$S_q(\omega) = 2(\text{WG})^2(\text{L}/\text{U}) \frac{1 + 2(p+1)k^2(\text{L}/\text{U})^2\omega^2}{[1 + k^2(\text{L}/\text{U})^2\omega^2]^{p+3/2}}$$

where:

| Type          | p   | k     |
|---------------|-----|-------|
| 1= von Karman | 1/3 | 1.339 |
| 2= Dryden     | 1/2 | 1.0   |

and  $\omega = 2\pi f$ . The units of  $S_q(\omega)$  are velocity squared per frequency ( $f$ ).

3. Other power spectral density functions may be defined using the TABRND1 entry.
4. If Modules are present then this entry may only be specified in the main Bulk Data section.

## TABSCTL Load Stepping Control Table - SOL 400

This option allows the user to provide user criteria for load stepping control. It is referred to by an NLSTEP entry. The criteria defined herein are used for controlling the load step size. The criteria come in two flavors as defined in the NLSTEP entry: limit and target. When used as limits, the time step will be reduced if the criterion would be violated. When used as target, the time step will also be increased if the calculated results are less than what is specified by the criteria.

The criteria are calculated for elements or grids as defined below. By default all elements or nodes are used for evaluating the respective criterion, but this can be limited to specific sets. (See the [NLSTEP, 2604](#) entry.)

Format:

| 1       | 2       | 3        | 4     | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|---------|----------|-------|-------|-------|-------|-------|-------|----|
| TABSCTL | TID     |          |       |       |       |       |       |       |    |
|         | ICRIT_1 | SET3_ID1 | YT1_1 | MT1_1 | YT2_1 | MT2_1 | YT3_1 | MT3_1 |    |
|         | YT4_1   | MT4_1    |       |       |       |       |       |       |    |
|         | ICRIT_2 | SET3_ID2 | YT1_2 | MT_2  | YT2_2 | MT2_1 | YT3_2 | MT3_2 |    |
|         | YT4_2   | MT4_2    |       |       |       |       |       |       |    |
|         | ...     | ...      | ...   | ...   |       |       |       |       |    |
|         | ICRIT_n | SET3_IDn | YT1_n | MT1_n | YT1_n | MT2_n | YT3_n | MT3_n |    |
|         | YT4_n   | MT4_n    |       |       |       |       |       |       |    |

Example:

|         |      |     |     |     |     |     |     |  |  |
|---------|------|-----|-----|-----|-----|-----|-----|--|--|
| TABSCTL | 17   |     |     |     |     |     |     |  |  |
|         | ESI  | 84  | .03 | 21. | 0.4 | 55. | 0.6 |  |  |
|         | ESRI | 18  | .03 | 21. |     |     |     |  |  |
|         | EPSI | ALL | 1.1 |     |     |     |     |  |  |

| Describer | Meaning                                                                                    |
|-----------|--------------------------------------------------------------------------------------------|
| TID       | ID of TABSCTL entry. (Integer > 0)                                                         |
| ICRITi    | Type of user criterion to use and the type of entity it refers to: (Character; no Default) |
| ESI       | Element Strain Increment                                                                   |
| EPSI      | Element Plastic Strain Increment                                                           |
| ECSI      | Element Creep Strain Increment                                                             |
| ENCSI     | Element Normalized Creep Strain Increment                                                  |
| ESTRI     | Element STress Increment                                                                   |
| ESRI      | Element Strain Energy Increment                                                            |
| GTI       | Grid Temperature Increment                                                                 |

| Describer | Meaning                                                                                                   |
|-----------|-----------------------------------------------------------------------------------------------------------|
|           | GDI      Grid Displacement Increment                                                                      |
|           | GRI      Grid Rotation Increment                                                                          |
|           | ENSTRI   Element Normalized STress Increment                                                              |
| SET3_IDi  | ID of a SET3 Bulk Data entry. (Integer > 0 or (ALL or BLANK); Default = ALL)                              |
| YT1_i     | First limit or target value for the current criterion i. (Real; no Default)                               |
| MT1_i     | First range or target value for the current criterion i. See Remark 2. (Real-default blank-always active) |
| YT2_i     | Second limit or target value for criterion. (Real; no Default)                                            |
| MT2_i     | Second range of applicability for criterion i. See Remark 2. (Real-default blank-always active)           |

Remarks:

1. This entry is selected by the CRITTID field of the NLSTEP Bulk Data entry.
2. For each criterion ICRIT\_n, up to four ranges of target/limit values can be given. This allows the use of different limit/target values for different ranges of the corresponding total quality. For example, for the first criterion type, YT1\_1 is the strain increment while MT1\_1 is the largest total strain for which YT1\_1 will be used. Typically, the last MTn used should be zero in which case the corresponding YTn will be used for all larger total values.
3. The TID must be unique among all TABSCTL entries.

## TEMP

## Grid Point Temperature Field

Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

| 1    | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|------|-----|----|----|----|----|----|----|---|----|
| TEMP | SID | G1 | T1 | G2 | T2 | G3 | T3 |   |    |

Example:

|      |   |    |       |    |       |  |  |  |  |
|------|---|----|-------|----|-------|--|--|--|--|
| TEMP | 3 | 94 | 316.2 | 49 | 219.8 |  |  |  |  |
|------|---|----|-------|----|-------|--|--|--|--|

| Describer | Meaning                                              |
|-----------|------------------------------------------------------|
| SID       | Temperature set identification number. (Integer > 0) |
| Gi        | Grid point identification number. (Integer > 0)      |
| Ti        | Temperature. (Real)                                  |

Remarks:

- In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- Set ID must be unique with respect to all other LOAD type entries if TEMP(Load) is specified in the Case Control Section.
- From one to three grid point temperatures may be defined on a single entry.
- If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
- If the element material is temperature dependent, its properties are evaluated at the average temperature.
- Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data are defined. Gauss point temperatures are averaged for solid elements instead of grid point temperature.
- For steady state heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMPD and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC = SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.

## TEMPAX Conical Shell Temperature

Defines temperature sets for conical shell problems.

Format:

| 1      | 2    | 3    | 4    | 5  | 6    | 7    | 8    | 9  | 10 |
|--------|------|------|------|----|------|------|------|----|----|
| TEMPAX | SID1 | RID1 | PHI1 | T1 | SID2 | RID2 | PHI2 | T2 |    |

Example:

|        |   |   |      |       |  |  |  |  |  |
|--------|---|---|------|-------|--|--|--|--|--|
| TEMPAX | 4 | 7 | 30.0 | 105.3 |  |  |  |  |  |
|--------|---|---|------|-------|--|--|--|--|--|

| Describer | Meaning                                                      |
|-----------|--------------------------------------------------------------|
| SIDi      | Temperature set identification number. (Integer > 0)         |
| RIDi      | Ring identification number (see RINGAX entry). (Integer > 0) |
| PHIi      | Azimuthal angle in degrees. (Real)                           |
| Ti        | Temperature. (Real)                                          |

Remarks:

- TEMPAX is allowed only if an AXIC entry is also present.
- SIDi must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
- Temperature sets must be selected with the Case Control command TEMP=SID.
- One or two temperatures may be defined on each entry.
- For a discussion of the conical shell problem, see [Conical Shell Element \(RINGAX\)](#) in the *MSC Nastran Reference Guide*.
- TEMP(INIT) is not used with this entry.



## TEMPB3 CBEAM3 Element Temperature Field

Defines a temperature field for the three-node beam element (CBEAM3 entry).

Format:

| 1      | 2      | 3       | 4      | 5     | 6     | 7      | 8      | 9      | 10 |
|--------|--------|---------|--------|-------|-------|--------|--------|--------|----|
| TEMPB3 | SID    | EID     | T(A)   | T(B)  | T(C)  | TPY(A) | TPZ(A) | TPY(B) |    |
|        | TPZ(B) | TPY(C)  | TPZ(C) | TC(A) | TD(A) | TE(A)  | TF(A)  | TC(B)  |    |
|        | TD(B)  | TE(B)   | TF(B)  | TC(C) | TD(C) | TE(C)  | TF(C)  |        |    |
|        |        | Element | ID     | List  |       |        |        |        |    |

Example:

|        |      |     |      |      |      |      |      |      |  |
|--------|------|-----|------|------|------|------|------|------|--|
| TEMPB3 | 101  | 23  | 45.9 | 10.0 | 0.0  | 1.3  | 23.9 | 3.8  |  |
|        |      | 2.5 | 68.0 | 91.0 | 45.0 |      | 48.0 | 80.0 |  |
|        | 20.0 |     | 33.9 |      |      | 45.6 |      |      |  |
|        | 9    | 10  | THRU | 30   | 41   | 51   | 67   | 78   |  |
|        | THRU | 110 | BY   | 2    |      |      |      |      |  |

| Describer       | Meaning                                                                                                                                                                       |
|-----------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID             | Temperature set identification number. (Integer > 0; Required)                                                                                                                |
| EID             | Element identification number. (Integer > 0; Required)                                                                                                                        |
| T(j)            | Temperature at j (j=A,B,C) on the neutral axis. (Real; Default = 0.0)                                                                                                         |
| TPi(j)          | Effective linear gradient in local direction i (i = y, z) at j (j = A, B, C). (Real; Default = 0.0)                                                                           |
| Ti(j)           | Temperature at stress recovery point i (i=C, D, E, F) defined in PBEAM3 at location j (j=A, B, C). (Real; Default = 0.0; see Remark 3.)                                       |
| Element ID List | List of CBEAM3 element identification numbers. Character strings "THRU" and "BY" may be used in the list. (Integer > 0; "THRU" or "BY". At least one element ID is required.) |

Remarks:

- In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- SID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
- If all Ti(j) fields are blank, linear temperature gradients are assumed for stress recovery.

4. Temperature field defined by TEMPB3 entry always takes precedence over the grid point temperatures given by TEMP and TEMPD entries.
5. The effective thermal gradients are defined in the local coordinate system. For their definitions, see Remark 6 of Bulk Data entry TEMPRB for the details.

## TEMPBC      Grid Point Temperatures

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions (SOLs 153 and 159 only).

Format:

| 1      | 2   | 3    | 4     | 5    | 6     | 7    | 8     | 9    | 10 |
|--------|-----|------|-------|------|-------|------|-------|------|----|
| TEMPBC | SID | TYPE | TEMP1 | GID1 | TEMP2 | GID2 | TEMP3 | GID3 |    |

Example:

|        |    |      |       |   |       |   |       |   |  |
|--------|----|------|-------|---|-------|---|-------|---|--|
| TEMPBC | 10 | STAT | 100.0 | 1 | 100.0 | 2 | 100.0 | 3 |  |
|--------|----|------|-------|---|-------|---|-------|---|--|

Alternate Format and Example:

|        |     |      |       |      |        |      |      |     |  |
|--------|-----|------|-------|------|--------|------|------|-----|--|
| TEMPBC | SID | TYPE | TEMP1 | GID1 | “THRU” | GID2 | “BY” | INC |  |
|--------|-----|------|-------|------|--------|------|------|-----|--|

|        |    |      |       |   |      |    |    |   |  |
|--------|----|------|-------|---|------|----|----|---|--|
| TEMPBC | 20 | STAT | 100.0 | 4 | THRU | 50 | BY | 2 |  |
|--------|----|------|-------|---|------|----|----|---|--|

| Describer | Meaning                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Temperature set identification number. (Integer > 0)                                                                                                                                        |
| TYPE      | Type of temperature boundary condition. See Remarks. (Character; Default = “STAT”):<br>STAT - Constant temperature boundary condition<br>TRAN - Time-varying temperature boundary condition |
| TEMPi     | Temperature (Real)                                                                                                                                                                          |
| GIDi      | Grid point identification number. (Integer>0 or “THRU” or “BY”)                                                                                                                             |
| INC       | Grid point number increment. (Integer)                                                                                                                                                      |

Remarks:

- For a constant Boundary Condition (TYPE = “STAT”), the temperature boundary load set (SID) is selected in the Case Control Section (SPC = SID). TYPE = “STAT” may be used in both steady-state (SOL 153) and transient analysis (SOL 159).
- For transient analysis (SOL 159), a constant boundary condition should be specified using the SPC Bulk Data entry.
- For a time-varying boundary condition (TYPE = “TRAN”), SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. TYPE= “TRAN” is permitted only in transient analysis (SOL 159). A function of time  $F(t - \tau)$  defined on the TLOADi entry multiplies the general load.  $\tau$  provides any required time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis.

4. In the alternate format, TEMP1 is the nodal temperature for the grid points  $GID1, GID1+INC, \dots, GID2$ . If “BY” and INC are not specified, then the grid point number increment is unity.
5. If TYPE = “STAT”, then no SPCi Bulk Data entries may be specified.
6. If TYPE = “TRAN”, then no CELAS2 or DAREA Bulk Data entries may be specified. Also, “U” must be specified in the CONV field on the entry to obtain accurate results.
7. All TEMPBC entries in the Bulk Data Section must indicate either TYPE = “STAT” or TYPE = “TRAN” but not both.
8. In transient thermal analysis, the TEMPBC option is used to set a grid, known temperature as a function of time. Internally NASTRAN uses SLOAD and CELAS2 entries to enforce the temperature as a function of time. The  $u=P/K$  or temperature is equal to SLOAD divided by CELAS2. The default stiffness for the CELAS2 entry is 1.0E10. This value is fine most of the time. However, if the user desired to run the model using thermal conductivity in the following unit (Btu/sec.inch.F), then it may run into a numerically convergence issue. This is because the thermal conductivity for this unit has conductivity value in the 1.0E-6 range. The avoidance is to set a NASTRAN system cell, TBCMAG to 1.0E2.
9. The TEMPBC with type=TRAN is no longer supported in SOL 400 transient thermal analysis. One should use the SPC1 and SPCD to enforced temperature. For example, convection coefficient as a function of time, mass flow rate as a function of time, or ambient temperature as a function of time all used the TEMPBC,TRAN option to apply a time varying nodal quantity in SOL 159. Following are the procedures for a user to convert SOL 159 into SOL 400 when there is TEMPBC,TRAN entries.

The SPC entry used to fixed a boundary temperature at a particular value at all time. However, if you have a TEMPBC,TRAN in your test file this means that all the permanent SPC must be converted into SPCD and SPC1 with a unit step function on the TABLED1 in SOL 400. The avoidance to this, using a constant temperature with SPC and a time-varying temperature in the same run, is using the large stiffness method to enforced the time varying quantity. The large stiffness used SLOAD and CELAS2 which  $u = P / K$  which  $P = SLOAD$ , and  $K = CELAS2$ , and  $u$  is the desired temperature times the time-varying quantity in the TABLEDx.

To Convert SOL 159 Models to SOL 400 Models:

- a. Executive Control Section - change SOL 159 to SOL 400.
- b. Case Control Section - replace ANALYSIS=HEAT by ANALYSIS=HTRAN, also add SPC if all temperature boundary conditions are transient (the following Case 3b).
- c. Bulk Data Section - replace the “TRAN” type TEMPBC by SPC1 and SPCD. The details are explained below.

If all temperature boundary conditions are constant, no changes are required.

If all temperature boundary conditions are transient, replace TEMPBC by SPC1 and SPCD and modify TLOAD1.

For example, replace the following entries of SOL 159 model:

```
TLOAD1,40,400,,4000
TEMPBC,400,TRAN,300.0,99
```

by

SPC = 111 (Case CC)

:

TLOAD1,40,400,,1,4000

SPCD,400,99,,300.0

SPC1,111,,99

If a model has both constant and transient temperature boundary conditions, all boundary conditions must be converted into SPC1 and SPCD.

For example, replace the following entries of SOL 159 model:

DLOAD,222,1.0,1,0,30,1.0,40

TLOAD1,40,400,,4000

TEMPBC,400,TRAN,300.0,99

SPC,111,98,,20.0

by

DLOAD,222,1.0,1,0,30,1.0,40, 1.0,50

TLOAD1,40,400,,1,4000

SPCD,400,99,,300.0

SPC1,111,,99

TLOAD1,50,500,,1,5000

SPCD,500,98,,20.0

SPC1,111,,98

TABLED1,5000,,,,,,,,

,0.0,1.0,1000.0,1.0,ENDT

10. If TYPE=TRAN and the initial temperature is non zero, then the initial temperature must be supplied. Use case control IC and bulk data TEMP entries.

**TEMPD**

## Grid Point Temperature Field Default

Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP or TEMPN1 (for heat transfer analysis) entries.

Format:

| 1     | 2    | 3  | 4    | 5  | 6    | 7  | 8    | 9  | 10 |
|-------|------|----|------|----|------|----|------|----|----|
| TEMPD | SID1 | T1 | SID2 | T2 | SID3 | T3 | SID4 | T4 |    |

Example:

|       |   |       |  |  |  |  |  |  |  |
|-------|---|-------|--|--|--|--|--|--|--|
| TEMPD | 1 | 216.3 |  |  |  |  |  |  |  |
|-------|---|-------|--|--|--|--|--|--|--|

| Describer | Meaning                                              |
|-----------|------------------------------------------------------|
| SIDi      | Temperature set identification number. (Integer > 0) |
| Ti        | Default temperature value. (Real)                    |

Remarks:

- In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
- SIDi must be unique with respect to all other LOAD type entries if TEMP(Load) is specified in the Case Control Section.
- From one to four default temperatures may be defined on a single entry.
- If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1, TEMPP3, or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
- If the element material is temperature dependent, its properties are evaluated at the average temperature.
- Average element temperatures are obtained as a simple average of the connecting grid point temperatures when no element temperature data is defined.
- For steady-state heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT) = SID requests selection of this entry. The temperature values specified here must be coincident with any temperatures boundary conditions that are specified.

8. For transient heat transfer analysis, this entry together with the TEMP and TEMPN1 entries supplies the initial condition temperatures. The Case Control command IC=SID request selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
9. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
10. For partitioned Bulk Data superelements and auxiliary models, TEMPD must be specified in all partitioned Bulk Data Sections.
11. If Modules are present then this entry may only be specified in the main Bulk Data section.

## TEMPN1

## TOP/BOT/MID Grid Point Temperature Field for Heat Shell Element in SOL 400

Defines initial temperature at grid points of heat shell elements with linear or quadratic temperature distribution across the thickness direction.

Format:

| 1      | 2   | 3  | 4  | 5  | 6  | 7  | 8  | 9 | 10 |
|--------|-----|----|----|----|----|----|----|---|----|
| TEMPN1 | SID | G1 | C1 | T1 | G2 | C2 | T2 |   |    |

Example:

|        |    |     |     |       |  |  |  |  |  |
|--------|----|-----|-----|-------|--|--|--|--|--|
| TEMPN1 | 10 | 100 | 123 | 1300. |  |  |  |  |  |
|--------|----|-----|-----|-------|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                                             |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Temperature set identification number. (Integer > 0)                                                                                                                                |
| Gi        | Grid point identification number. (Integer > 0)                                                                                                                                     |
| Ci        | Component numbers. ( $0 \leq \text{Integer} \leq 3$ ; up to 3 unique Integers may be placed in the field with no embedded blanks.) 1=TOP, 2=BOT, 3=MID. (Integer > -1; Default = 1) |
| Ti        | Temperature. (Real)                                                                                                                                                                 |

Remarks:

1. This entry is for shell elements defined on a PSHLN1 heat transfer.
2. In the steady-state solution sequences, SID is selected by the LOAD Case Control command.
3. In the transient solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the LID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an ACSRCE, RLOADi or TLOADi entry.
4. The TEMPN1 Bulk Data entry is used to complement the TEMP Bulk Data entry. The TEMP entry will just initiate the TOP grids. This entry allows specifying of the TOP, BOT, and MID or any combination. See the NLMOPTS,TEMGO,vmaptg Bulk Data entry on how to list internally generated grids.
5. For steady-state heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initialization temperatures for nonlinear analysis. The Case Control command TEMP(INIT)=SID requests selection of this entry. The temperature values specified here must be coincident with any temperature boundary conditions that are specified.
6. For transient heat transfer analysis, this entry together with the TEMPD and TEMP entries supplies the initial condition temperatures. The Case Control command IC=SID requests selections of this entry. The temperature values specified here must be coincident with any temperature boundary condition specified.



TEMPP1

Plate Element Temperature Field, Form 1

Defines a temperature field for plate, membrane, and combination elements (by an average temperature and a thermal gradient through the thickness) for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

|        |      |      |      |        |      |      |        |   |    |
|--------|------|------|------|--------|------|------|--------|---|----|
| 1      | 2    | 3    | 4    | 5      | 6    | 7    | 8      | 9 | 10 |
| TEMPP1 | SID  | EID1 | TBAR | TPRIME | T1   | T2   |        |   |    |
|        | EID2 | EID3 | EID4 | EID5   | EID6 | EID7 | -etc.- |   |    |

Example:

|        |    |    |      |      |      |      |  |  |  |
|--------|----|----|------|------|------|------|--|--|--|
| TEMPP1 | 2  | 24 | 62.0 | 10.0 | 57.0 | 67.0 |  |  |  |
|        | 26 | 21 | 19   | 30   |      |      |  |  |  |

Alternate Format and Example of Continuation Entry:

|  |      |        |      |      |        |      |  |  |  |
|--|------|--------|------|------|--------|------|--|--|--|
|  | EID2 | “THRU” | EIDi | EIDj | “THRU” | EIDk |  |  |  |
|  | 1    | THRU   | 10   | 30   | THRU   | 61   |  |  |  |

| Describer        | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                              |
|------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID              | Temperature set identification number. (Integer > 0)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |
| EIDi, EIDj, EIDk | Unique element identification number(s). (Integer > 0 or the continuation entries may have “THRU” in fields 3 and/or 6, in which case EID2 < EIDi and EIDj < EIDk.)                                                                                                                                                                                                                                                                                                                                                                                                  |
| TBAR             | Temperature at the element’s reference plane as defined by ZOFFS on the connection entry. (Real, Default 0.0)                                                                                                                                                                                                                                                                                                                                                                                                                                                        |
| TPRIME           | Effective linear thermal gradient. Not used for membranes. (Real, Default 0.0)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                       |
| T1, T2           | Temperatures for stress calculation at points defined on the element property entry. (Z1 and Z2 field on PSHELL entry.) T1 may be specified on the lower surface and T2 on the upper surface for the CQUAD4, CQUAD8, CTRIA3, CTRIA6, CQUADR, and CTRIAR elements. These data are not used for membrane elements. See Remark 9. If both T1 and T2 are blank, they are computed from the equation $T = TBAR + z \cdot TPRIME$ , where $z$ is the distance from the center fiber. The program replaces T1 with a flag, and $z$ is computed in a later operation. (Real) |

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.

In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry. TBAR and TPRIME are used for the analysis. If both are left blank the elements in essence see no thermal loading. T1 and T2 are used for post analysis for stress calculations.

2. Set ID must be unique with respect to all other LOAD type entries if TEMP(LOAD) is specified in the Case Control Section.
3. If continuation entries are present, EID1 and elements specified on the continuation entry are used. Elements must not be specified more than once.
4. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
5. For temperature field other than a constant gradient, the “effective gradient” for a homogeneous plate is

$$TPRIME = \frac{1}{I} \int_z T(z) z dz$$

where  $I$  is the bending inertia and  $z$  is the distance from the neutral surface in the positive normal direction.

6. The “average” temperature for a homogeneous plate is

$$TBAR = \frac{1}{Volume} \int_{Volume} T dVolume$$

7. If the element material is temperature dependent, its properties are evaluated at the average temperature TBAR.
8. Large “THRU” ranges will lead to System Fatal Message 3008 (“Insufficient Core”) and should be avoided, particularly for open sets.
9. If the element material is nonlinear then T1 and T2 should be left blank (see the MATS1 entry).
10. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.
11. The bending and twisting moments can be reduced to outer fiber stresses and combined with membrane stresses in the composite plate elements. If, in addition, the temperature is specified by the user at a point where outer fiber stresses are calculated, the thermal expansion due to the difference between the specified temperature and the temperature that would be produced by a uniform gradient,  $T'$ , is assumed to be completely restrained. Stated differently, the second and higher order moments of the thermal expansion are assumed to be completely restrained by elastic stiffness. The resulting stress increment is

$$\{\Delta\sigma\} = -[G_e]\{\alpha_e\}(T - T_o - T'z)$$

↑  
— temperature at reference plane

where  $[G_e]$  and  $\{\alpha_e\}$  are evaluated for the average temperature of the element  $\bar{T}$ .

## **TEMPP3**

## Plate Element Temperature Field, Form 3

TEMPP3 is no longer available. Use TEMPP1.

## TEMPRB      One-Dimensional Element Temperature Field

Defines a temperature field for the CBAR, CBEAM, CBEND, CROD, CTUBE, and CONROD elements for determination of thermal loading, temperature-dependent material properties, or stress recovery.

Format:

| 1      | 2    | 3    | 4    | 5    | 6    | 7    | 8      | 9    | 10 |
|--------|------|------|------|------|------|------|--------|------|----|
| TEMPRB | SID  | EID1 | TA   | TB   | TP1A | TP1B | TP2A   | TP2B |    |
|        | TCA  | TDA  | TEA  | TFA  | TCB  | TDB  | TEB    | TFB  |    |
|        | EID2 | EID3 | EID4 | EID5 | EID6 | EID7 | -etc.- |      |    |

Example:

|        |      |      |      |      |      |      |      |     |  |
|--------|------|------|------|------|------|------|------|-----|--|
| TEMPRB | 200  | 1    | 68.0 | 23.0 | 0.0  | 28.0 |      | 2.5 |  |
|        | 68.0 | 91.0 | 45.0 |      | 48.0 | 80.0 | 20.0 |     |  |
|        | 9    | 10   |      |      |      |      |      |     |  |

Alternate Format and Example of Continuation Entry:

|  |      |        |      |      |        |      |  |  |  |
|--|------|--------|------|------|--------|------|--|--|--|
|  | EID2 | “THRU” | EIDi | EIDj | “THRU” | EIDk |  |  |  |
|  | 2    | THRU   | 4    | 10   | THRU   | 14   |  |  |  |

| Describer        | Meaning                                                                                                                                                                                                                 |
|------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID              | Temperature set identification number. (Integer > 0)                                                                                                                                                                    |
| EIDi, EIDj, EIDk | Unique element identification number(s). (Integer > 0 or the second continuation entry may have “THRU” in fields 3 and/or 6 in which case EID2 < EIDi and EIDj < EIDk.)                                                 |
| TA, TB           | Temperature at end A and end B on the neutral axis. (Real, default 0.0)                                                                                                                                                 |
| TPij             | Effective linear gradient in direction i on end j; used with CBAR, CBEAM, and CBEND only. (Real)                                                                                                                        |
| Tij              | Temperature at point i (i=C, D, E, or F) as defined on the PBAR, PBEAM, and PBEND entries at end j (j=A or B). This data is used for stress recovery only with CBAR, CBEAM, and CBEND exclusively. See Remark 3. (Real) |

Remarks:

1. In the static solution sequences, SID must be selected by the TEMP Case Control command.  
In the dynamic solution sequences, if there is a LOADSET Case Control command, then SID must be referenced in the TID field of a selected LSEQ entry. If there is no LOADSET Case Control command, then SID must be referenced in the EXCITEID field of an RLOADi or TLOADi entry.
2. SID must be unique with respect to all other LOAD type entries if TEMP(Load) is specified in the Case Control Section.

3. If at least one nonzero or nonblank  $T_{ij}$  is present, the point temperatures given are used for stress recovery. If no  $T_{ij}$  values are given, linear temperature gradients are assumed for stress recovery. The  $T_{ij}$  values are not used in the calculation of differential stiffness. The default for  $T_{ij}$  for  $j=A$  is  $TA$  and the default for  $T_{ij}$  for  $j=B$  is  $TB$ .
4. If the second (and succeeding) continuation is present, EID1 and elements specified on the second (and succeeding) continuations are used. Elements must not be specified more than once.
5. If thermal effects are requested, all elements must have a temperature field defined either directly on a TEMPP1 or TEMPRB entry or indirectly as the average of the connected grid point temperatures defined on the TEMP or TEMPD entries. Directly defined element temperatures always take precedence over the average of grid point temperatures.
6. The effective thermal gradients in the element coordinate system for the CBAR element are defined by the following integrals over the cross section. For end "A" (end "B" is similar),

$$TA = \frac{1}{A} \int TA(y, z) dA$$

$$TP1A = \frac{I_2}{\Delta} \int_A (y - y_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (z - z_n) TA(y, z) dA$$

$$TP2A = \frac{I_1}{\Delta} \int_A (z - z_n) TA(y, z) dA - \frac{I_{12}}{\Delta} \int_A (y - y_n) TA(y, z) dA$$

$$\Delta = I_1 I_2 - I_{12}^2$$

$$\text{if } I_{12} = 0$$

$$TP1A = \frac{1}{I_1} \int_A (y - y_n) TA(y, z) dA$$

$$TP2A = \frac{1}{I_2} \int_A (z - z_n) TA(y, z) dA$$

where  $TA(y, z)$  is the temperature at point  $y, z$  (in the element coordinate system) at end "A" of the bar. See the CBAR entry description for the element coordinate system:  $I_1$ ,  $I_2$  and  $I_{12}$  are the moments of inertia about the  $z$  and  $y$  axes, respectively. The temperatures are assumed to vary linearly along the length ( $x$ -axis). Note that if the temperature varies linearly over the cross section, then  $TP1A$ ,  $TP1B$ ,  $TP2A$  and  $TP2B$  are the actual gradients.

7. If the element material is temperature-dependent, the material properties are evaluated at the average temperature  $(TA + TB)/2$ .
8. In linear and nonlinear buckling analysis, the follower force effects due to loads from this entry are not included in the differential stiffness. See [Buckling Analysis in SubDMAP MODERS](#) and [Nonlinear Static Analysis](#) in the *MSC Nastran Reference Guide*.

9. If any  $T_y$  is specified the stresses computed by the effective gradient are corrected by  $\Delta\sigma$  such that:

$$\sigma = \sigma \Big|_{T_A + y^{TP1A} + z^{TP2A}} + \Delta\sigma$$

where  $\Delta\sigma$  is in the form

$$\Delta\sigma = -\alpha E [T_{CA} - T_o - C_1 \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBAR and CBEAM

$$\Delta\sigma = -\alpha E [T_{CA} - T_o - (C_1 + \Delta N) \cdot TPIA - C_2 \cdot TP2A] \text{ etc}$$

for CBEND.

**TERMIN** Control to Terminate a SOL 600 Analysis Under Certain Conditions

Used in SOL 600 only.

Format:

| 1      | 2     | 3   | 4     | 5   | 6 | 7 | 8 | 9 | 10 |
|--------|-------|-----|-------|-----|---|---|---|---|----|
| TERMIN | ID    | NC  |       |     |   |   |   |   |    |
|        | NTYPE | NBN | ICRIT | VAL |   |   |   |   |    |

Example:

|        |   |      |    |     |  |  |  |  |  |
|--------|---|------|----|-----|--|--|--|--|--|
| TERMIN | 2 | 2    |    |     |  |  |  |  |  |
|        | 7 | 1000 | -1 | 2.0 |  |  |  |  |  |
|        | 7 | 1000 | -2 | 0.8 |  |  |  |  |  |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                   |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | ID corresponding to a Case TERMIN entry. (Integer > 0; no Default)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        |
| NC        | Number of termination conditions to be specified. (Integer > 0; Default = 1; Max number is 10)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                            |
| NTYPE     | Termination Criteria Type (Integer; no Default)<br>Enter 1 if termination occurs when a percentage of the boundary nodes are in contact.<br>Enter 2 if termination occurs when the maximum force on a rigid body is exceeded.<br>Enter 3 if termination occurs when the displacement of the rigid body exceeds the allowed displacement.<br>Enter 5 if termination occurs when the distance between the reference points of two rigid bodies is less or greater than the specified value.<br>Enter 6 if termination occurs, when any displacement in body, is greater than the specified value.<br>Enter 7 if termination occurs, when the displacement in the node, is greater than the specified value. |
| NBN       | Body number, for criterion type 7, grid ID (Integer > 0; no Default)                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                      |



| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                        |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICRIT     | <p>Criteria specification. (Integer; no Default)</p> <p>For criterion type 1, enter the percentage of nodes to be in contact for termination; default = 100.</p> <p>For criterion type 2, enter direction 1/2/3 for the x, y, z global directions</p> <p>For criterion type 5, enter the second body.</p> <p>For criterion type 6 or 7, enter the degree of freedom.</p> <p>For criterion type 6 or 7, enter -1 if the total translational displacement.</p> <p>For criterion type 6 or 7, enter -2 if the total rotation.</p> |
| VAL       | <p>Termination value. (Real; no Default)</p> <p>For criterion type 2, enter the critical force.</p> <p>For criterion type 3, enter the critical maximum displacement.</p> <p>For criterion type 5, enter the critical distance. If the value is positive, the termination occurs when the distance is less than the value. If the value is negative, the termination occurs when the distance is greater than the value in a positive sign.</p> <p>For criterion type 6 or 7, enter the critical distance (rotation).</p>      |

Remarks:

1. Different TERMIN entries may be used in different subcases.
2. Not all subcases require TERMIN entries if used in other subcases.

## TF Dynamic Transfer Function

Defines a dynamic transfer function of the form

$$(B0 + B1 \cdot p + B2 \cdot p^2)u_d + \sum_i (A0(i) + A1(i)p + A2(i)p^2)u_i = 0 \quad (9-30)$$

Where:

- $U_d$  = dependent coordinate
- $U_i$  = independent degree of freedom
- $p$  = differential operator ( $p=d/dt$ )

Can also be used as a means of direct matrix input. See Remark 4.

Format:

| 1  | 2    | 3    | 4     | 5     | 6     | 7      | 8 | 9 | 10 |
|----|------|------|-------|-------|-------|--------|---|---|----|
| TF | SID  | GD   | CD    | B0    | B1    | B2     |   |   |    |
|    | G(1) | C(1) | A0(1) | A1(1) | A2(1) | -etc.- |   |   |    |

Example:

|    |   |   |     |     |     |     |  |  |  |
|----|---|---|-----|-----|-----|-----|--|--|--|
| TF | 1 | 2 | 3   | 4.0 | 5.0 | 6.0 |  |  |  |
|    | 3 | 4 | 5.0 | 6.0 | 7.0 |     |  |  |  |

| Describer           | Meaning                                                                                                                      |
|---------------------|------------------------------------------------------------------------------------------------------------------------------|
| SID                 | Set identification number. (Integer > 0)                                                                                     |
| GD, G(i)            | Grid, scalar, or extra point identification numbers. (Integer > 0)                                                           |
| CD, C(i)            | Component numbers. (Integer zero or blank for scalar or extra points, any one of the Integers 1 through 6 for a grid point.) |
| B0, B1, B2          | Transfer function coefficients. (Real)                                                                                       |
| A0(i), A1(i), A2(i) |                                                                                                                              |

Remarks:

1. Transfer function sets must be selected with the Case Control command TFL = SID.
2. Continuation entries are optional.
3. The matrix elements defined by this entry are added to the dynamic matrices for the problem.

4. The constraint relation given in Eq. (9-30) will hold only if no structural elements or other matrix elements are connected to the dependent coordinate  $u_d$ . In fact, the terms on the left side of Eq. (9-30) are simply added to the terms from all other sources in the row for  $u_d$ .
5. See the [MSC Nastran Dynamic Analysis User's Guide](#) for a discussion of transfer functions.
6. For each SID, only one logical entry is allowed for each GD, CD combination.
7. For heat transfer analysis, the initial conditions must satisfy Eq. (9-30).
8. RC network does not support TF for thermal analysis.
9. For more information see [DMIGs, Extra Points, and Transfer Functions](#) in the *Dynamic Analysis User's Guide*.

## THPAD ROMAC's THPAD User Defined Service Element Property

Allows the user to provide the parameters for a tilting pad bearing for use with ROMAC's THPAD service.

Format:

| 1      | 2           | 3       | 4        | 5             | 6      | 7      | 8      | 9     | 10 |
|--------|-------------|---------|----------|---------------|--------|--------|--------|-------|----|
| THPAD  | RID         | TLIMIT  |          |               |        |        |        |       |    |
| Title1 |             |         |          |               |        |        |        |       |    |
| Title2 |             |         |          |               |        |        |        |       |    |
|        | RADIUS      | CLEAR   | OUTR     | LENGT<br>H    | ALPHP  | E      | ALPHJ  | ALPHS |    |
|        | NPADS       | NEL     | IECC     |               |        |        |        |       |    |
|        | ARC1        | OFFSET1 | PRELOAD1 | PVANG1        | IP1    | KP1    | MP1    | IT1   |    |
|        | ARC2        | OFFSET2 | PRELOAD  | PVANG2        | IP2    | KP2    | MP2    | IT2   |    |
|        | ...         | ...     | ...      | ...           | ...    | ...    | ...    | ...   |    |
|        | KFILM       | KPAD    | TBACK    | TJF           | PSUMP  | TIN    | KCAV   | CCAV  |    |
|        | DENSIT<br>Y | SPEC    | TA       | MUA           | TB     | MUB    | ESUMP  | TMANU |    |
|        | ERROR       | XG      | YG       |               |        | NAX    | FACTOR | XFACT |    |
|        | ITJ         | IBC     | ITUR     | IDIM          |        | ICOND  | ITB    | ITG   |    |
|        |             |         | ICAV     | ICROSS        | IDEF   | IFLEX  | IUN    | ISUMP |    |
|        | KTHETA      | DEL0    | HOTOVER  | COLDDOVE<br>R |        |        |        |       |    |
|        | NCASE       | MAXC    | IALPH    |               |        |        |        |       |    |
|        | RPM1        | FX1     | FY1      | PGNU1         | RELAX1 | OFLOW1 | DEREL1 |       |    |
|        | RPM2        | FX2     | FY2      | PGNU2         | RELAX2 | OFLOW2 | DEREL2 |       |    |
|        | ...         | ...     | ...      | ...           | ...    | ...    | ...    | ...   |    |

Example:

In the FMS Section of the Nastran input stream a connect statement is required:

```
CONNECT SERVICE GR1 'SCR.MDSolver.Obj.Uds.Elements.thpad'
```

In Bulk Data:

|                                                                 |     |         |     |       |      |    |  |  |  |
|-----------------------------------------------------------------|-----|---------|-----|-------|------|----|--|--|--|
| CBUSH2D                                                         | 100 | 200     | 1   | 2     |      | XY |  |  |  |
| PBUSH2D                                                         | 200 |         |     |       |      |    |  |  |  |
| ELEMUDS                                                         | 200 | PBUSH2D | GR1 | THPAD | FREQ |    |  |  |  |
| THPAD                                                           | 200 | 1000.0  |     |       |      |    |  |  |  |
| Tilting pad bearing used for the 8 stage certrifugal compressor |     |         |     |       |      |    |  |  |  |

|  | Load on Pad |         |       |         |       |        |        |        |  |
|--|-------------|---------|-------|---------|-------|--------|--------|--------|--|
|  | 2.5         | 0.0038  | 3.5   | 3.0     | 3.0E7 | 6.5E-6 | 6.5E-6 | 6.5E-6 |  |
|  | 5           | 30      | 0     |         |       |        |        |        |  |
|  | 60.0        | 0.5     | 0.3   | 54.0    | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 126.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 198.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60          | 0.5     | 0.3   | 270.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 60.0        | 0.5     | 0.3   | 342.0   | 0.001 | 0.0    | 0.0    | 0.0    |  |
|  | 2.0E-6      | 0.00067 | 120.0 | 175.0   | 0.0   | 175.0  | 5.0    | 0.0    |  |
|  | 8.0E-5      | 180.0   | 100.0 | 4.13E-6 | 210.0 | 6.8E-7 | 1.0    | 75.0   |  |
|  | 0.001       | 0.2     | 0.2   |         |       | 2.0    | 100.0  | 100.0  |  |
|  | 0           | 2       | 1     | 0       |       | 0      | 0      | 0      |  |
|  |             |         | 1     | 0       | 2     | 0      | 0      | 0      |  |
|  | 0.0         | 0.0     | 100.0 | 0.0     |       |        |        |        |  |
|  | 2           | 100     | 100   |         |       |        |        |        |  |
|  | 1000.0      | 0.0     | 931.5 | 1.0     | 1.0   | 0.0    | 1.0    |        |  |
|  | 2000.0      | 0.0     | 931.5 | 1.0     | 1.0   | 0.0    | 1.0    |        |  |

| Describer | Meaning                                                                                                                                           |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------|
| RID       | ROMAC bearing identification number that matches the property identification number on an ELEMUDS entry as well as a PBUSH2D entry. (Integer > 0) |
| TLIMIT    | Time limit for a call to THPAD. If the call takes longer than this, the service call will be killed. (Real)                                       |
| TITLE1    | User supplied title. (Character)                                                                                                                  |
| TITLE2    | User supplied title. (Character)                                                                                                                  |
| RADIUS    | Journal radius (Real, in)                                                                                                                         |
| CLEAR     | Ground-in pad clearance. (Real, in)                                                                                                               |
| OUTR      | Outer radius of bearing pads. (Real, in)                                                                                                          |
| LENGTH    | Axial length of bearing pads. (Real, in)                                                                                                          |
| E         | Young's modulus for the pads. (Real, psi)                                                                                                         |
| ALPHP     | Pad thermal expansion coefficient. (Real, 1/F)                                                                                                    |
| ALPHJ     | Journal thermal expansion coefficient. (Real, 1/F)                                                                                                |
| ALPHS     | Shell thermal expansion coefficient. (Real, 1/F)                                                                                                  |
| NPADS     | Number of pads. (Integer > 0)                                                                                                                     |
| NEL       | Number of elements per pad. (Integer > 0)                                                                                                         |
| IECC      | Eccentricity variation flag. (Integer)                                                                                                            |
|           | 0 Allow eccentricity variation to match load.                                                                                                     |



| Describer | Meaning                                                           |
|-----------|-------------------------------------------------------------------|
|           | 1 Eccentricity fixed at initial value.                            |
| ARCi      | Arc length for the pad. (Real, deg)                               |
| OFFSETi   | Offset factor for the pad. (Real)                                 |
| PRELOADi  | Preload factor for the pad. (Real)                                |
| PVANGi    | Location of pad pivot angle. (Real, deg)                          |
| IPi       | Pad polar moment. (Real, lb-sec <sup>2</sup> -in)                 |
| KPi       | Pad pivot stiffness. KPi=0 for rigid pivots. (Real, lb/in)        |
| MPi       | Pad mass. (Real, lb-sec <sup>2</sup> /in)                         |
| ITi       | Pad bending inertia. (Real, in <sup>4</sup> )                     |
| KFILM     | Fluid thermal conductivity. (Real, btu/in-sec-F)                  |
| KPAD      | Pad thermal conductivity. (Real, btu/in-sec-F)                    |
| TBACK     | Back of pad temperature. (Real, F)                                |
| TJF       | Temperature of the journal surface. (Real, F)                     |
| PSUMP     | Sump oil pressure. (Real, psi)                                    |
| TIN       | Pad inlet oil temperature. (Real, F)                              |
| KCAV      | Thermal conductivities cavitation ratio. (Real)                   |
| CCAV      | Cavitation latent heat ratio. (Real, btu/sec-in <sup>2</sup> -F)  |
| DENSITY   | Lubricant density. (Real, lbf-sec <sup>2</sup> /in <sup>4</sup> ) |
| SPEC      | Lubricant specific heat. (Real, btu-in/lbf-sec <sup>2</sup> -F)   |
| TA        | First temperature datum. (Real, F)                                |
| MUA       | Viscosity at TA. (Real)                                           |
| TB        | Second temperature datum. (Real, F)                               |
| MUB       | Viscosity at TB. (Real)                                           |
| ESUMP     | Sump heating factor. (Real)                                       |
| TMANU     | Clearance set temperature. (Real, F)                              |
| ERROR     | Error criterion. (Real)                                           |
| XG        | Initial guess of x/cp. (Real)                                     |
| YG        | Initial guess of y/cp. (Real)                                     |
| NAX       | Axial pressure exponent. (Real, 2.0 is recommended)               |
| FACTOR    | Perturbation effect on position iteration. (Real)                 |
| XFACT     | Percentage of cross-coupled terms in position update. (Real)      |
| ITJ       | Journal temperature flag. (Integer)                               |
|           | 0 Journal temperature found as average of film temperature.       |

| Describer | Meaning                                                 |
|-----------|---------------------------------------------------------|
| IBC       | 1 Journal temperature fixed by user.                    |
|           | 2 Journal temperature set for zero heat flux to shaft.  |
|           | Boundary condition flag. (Integer)                      |
| ITUR      | 0 Reynolds boundary condition with flow correction.     |
|           | 1 Not used.                                             |
|           | 2 Reynolds boundary condition for pressure.             |
| IDIM      | Turbulence flag. (Integer)                              |
|           | 0 Laminar solution.                                     |
|           | 1 Turbulence allowed.                                   |
| ICOND     | Dimension flag. (Integer)                               |
|           | 0 Two-dimensional conduction in the pad.                |
|           | 1 Radial conduction in the pad.                         |
| ITB       | Conduction flag. (Integer)                              |
|           | 0 Energy equation includes conduction.                  |
|           | 1 Adiabatic (isothermal) energy equation.               |
| ITG       | Back temperature flag. (Integer)                        |
|           | 0 Pad back temperature equals sump temperature.         |
|           | 1 Pad back temperature fixed by user.                   |
| ICAV      | Groove temperature flag. (Integer)                      |
|           | 0 Groove temperature found iteratively by heat balance. |
|           | 1 Groove temperature fixed at sump temperature.         |
| ICROSS    | Cavitation flag. (Integer)                              |
|           | 0 No cavitation effects in energy equation.             |
|           | 1 Include cavitation effects in energy equation.        |
| IDEF      | Cross-film viscosity flag. (Integer)                    |
|           | 0 Constant cross-film viscosity.                        |
|           | 1 Variable cross-film viscosity.                        |
|           | Deformation flag. (Integer)                             |
| IFLEX     | 0 No pad/pivot deformations.                            |
|           | 1 Pivot deformations.                                   |
|           | 2 Pad, journal and shell thermal deformations.          |
|           | 4 Pad mechanical deformations.                          |
|           | Type of beam bending for pad deformations. (Integer)    |

| Describer | Meaning                                                                        |
|-----------|--------------------------------------------------------------------------------|
|           | 0 Curved beam analysis.                                                        |
|           | 1 Straight beam analysis.                                                      |
|           | 2 Simple curvature calculation.                                                |
| IUN       | Unloaded pads flag. (Integer)                                                  |
|           | 0 Effect of unloaded pads is considered.                                       |
|           | 1 Effect of unloaded pads is neglected.                                        |
| ISUMP     | Sump temperature flag. (Integer)                                               |
|           | 0 Sump temperature set to pad inlet oil temperature.                           |
|           | 1 Sump temperature found iteratively by heat balance.                          |
| KTHETA    | Pivot rotational stiffness. (Real, lb-in, Default = 0.0)                       |
| DELO      | Initial pad angle for moment balance. (Real, rad, Default = 0.0)               |
| HOTOVER   | Percentage of hot oil available carried to the groove. (Real, Default = 100.0) |
| COLDOVER  | Percentage of groove oil made up of inlet oil. (Real, Default = 0.0)           |
| NCASE     | Number of speed/load cases. (Integer > 0)                                      |
| MAXC      | Maximum number of position iterations. (Real, Default = 100)                   |
| IALPH     | New stiffness smoothing factor. (Real)                                         |
| RPMi      | Speed of journal. (Real, rpm)                                                  |
| FXi       | Applied force in the negative x-direction. (Real, lb)                          |
| FYi       | Applied force in the negative y-direction. (Real, lb)                          |
| PGNUi     | Whirl ratio of shaft. (Real)                                                   |
| RELAXi    | Position iteration relaxation factor. (Real)                                   |
| OFLOWi    | Oil flow to bearing. (Real, cips)                                              |
| DERELi    | Relaxation factor for pivot deformation iteration. (Real)                      |

Remarks:

1. This entry triggers the call to the THPAD service. The GROUP must match the GROUP field of the CONNECT SERVICE FMS entry.
2. On the FMS CONNECT entry, only the CONNECT SERVICE can be used with this entry.
3. PID must match an existing ELEMUDS and PBUSH2D PIDs.
4. All units must be in the English system.
5. Refer to <http://www.virginia.edu/romac/> for more information.



6. Use of this entry requires, that the user, has obtained the source for the THPAD routine from the *University of Virginia Rotating Machinery and Controls Laboratory (ROMAC)* and use the MSC Nastran Software Development Kit (SDK) to build it as a User Defined Service (UDS). SDK and UDS build instructions are available in the [MSC Nastran Rotordynamics User's Guide](#).

## TIC Transient Analysis Initial Condition

Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom (See Remark 6.). This entry may not be used for heat transfer analysis.

Format:

| 1   | 2   | 3 | 4 | 5  | 6  | 7 | 8 | 9 | 10 |
|-----|-----|---|---|----|----|---|---|---|----|
| TIC | SID | G | C | U0 | V0 |   |   |   |    |

Example:

|     |     |    |   |     |     |  |  |  |  |
|-----|-----|----|---|-----|-----|--|--|--|--|
| TIC | 100 | 10 | 3 | 0.1 | 0.5 |  |  |  |  |
|-----|-----|----|---|-----|-----|--|--|--|--|

| Describer | Meaning                                                                                                                                                  |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Set identification number. (Integer > 0)                                                                                                                 |
| G         | Grid, scalar, or extra point or modal coordinate identification number. (Integer > 0). See Remark 4.                                                     |
| C         | Component numbers. (Any one of the integers 1 through 6 for grid points, integer zero or blank for scalar or extra points and -1 for modal coordinates.) |
| U0        | Initial displacement. (Real)                                                                                                                             |
| V0        | Initial velocity. (Real)                                                                                                                                 |

Remarks:

1. Transient analysis initial condition sets must be selected with the IC Case Control command. Note the use of IC in the Case Control command versus TIC on the Bulk Data entry. For heat transfer, the IC Case Control command selects TEMP or TEMPD entries for initial conditions and not the TIC entry.
2. If no TIC set is selected in the Case Control Section, all initial conditions are assumed to be zero.
3. Initial conditions for coordinates not specified on TIC entries will be assumed to be zero.
4. In direct transient analysis (SOL 109 and 129) as well as in modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC or IC(PHYSICAL) Case Control command, G may reference only grid, scalar or extra points. In modal transient analysis (SOL 112) wherein the TIC Bulk Data entry is selected by an IC(MODAL) Case Control command, G may reference only modal coordinates or extra points.
5. The initial conditions for the independent degrees-of-freedom specified by this Bulk Data entry are distinct from, and may be used in conjunction with, the initial conditions for the enforced degrees-of-freedom specified by TLOAD1 and/or TLOAD2 Bulk Data entries.
6. SOL700 does not support U0 (initial displacement).

## TICEL

## Transient Initial Conditions of Elements

Defines the initial values of element variables at the beginning of the analysis. Used in SOL 700 only.

Format:

|       |     |       |       |        |       |        |        |   |    |
|-------|-----|-------|-------|--------|-------|--------|--------|---|----|
| 1     | 2   | 3     | 4     | 5      | 6     | 7      | 8      | 9 | 10 |
| TICEL | SID | SETID | NAME1 | VALUE1 | NAME2 | VALUE2 | -etc.- |   |    |

Example:

|       |   |    |         |      |     |      |  |  |  |
|-------|---|----|---------|------|-----|------|--|--|--|
| TICEL | 3 | 40 | DENSITY | 100. | SIE | 1.E5 |  |  |  |
|-------|---|----|---------|------|-----|------|--|--|--|

| Describer | Meaning                                                                                 |
|-----------|-----------------------------------------------------------------------------------------|
| SID       | Set identification number. (Integer > 0; Required)                                      |
| SETID     | Number of a SET1 entry defining the elements to be initialized. (Integer > 0; Required) |
| NAMEi     | Element variable to be initialized. See Remark 5. (Character; Required)                 |
| VALUEi    | Value of the variable. (Real; Required)                                                 |

Remarks:

1. Initial conditions for elements that are not specified on TICEL entries are assumed to be zero except density, which is set to the reference density.
2. Only initial conditions that are selected in the Case Control Section (IC = SID) will be activated by the solver.
3. As many continuation lines as required can be used to specify all the variables being initialized. A blank field terminates the list.
4. Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder,...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence, and overrules the TICEUL1 initialization for the element.
5. The following variables for NAMEi can be used to initialize the Eulerian regions:

|         |                          |
|---------|--------------------------|
| XVEL    | x-velocity               |
| YVEL    | y-velocity               |
| ZVEL    | z-velocity               |
| DENSITY | Density                  |
| SIE     | Specific internal energy |
| Q       | Artificial viscosity     |

|      |                   |
|------|-------------------|
| DIV  | Divergence        |
| VOID | Void fraction     |
| FMAT | Material fraction |
| XMOM | x-momentum        |
| YMOM | y-momentum        |
| ZMOM | z-momentum        |

6. To initialize the pressure use density. And depending on the equation of states also define the specific internal energy (SIE).
7. For the Euler solvers, you can, in addition to the “normal” element variables that the solver has defined, also define an initial radial velocity field. You have to enter the location of the center from where the radial emerges, the velocity to be applied to the element center and the decay coefficient for the velocity field. The center is defined by the keywords “X-CENTER, Y-CENTER, Z-CENTER”, the radial velocity by “R-VEL” and the decay coefficient by "DECAY". You have to input these keywords in the above order, and have every keyword followed by its value.

## TICEUL1

## Transient Initial Conditions of Eulerian Regions

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

Format:

| 1       | 2   | 3    | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|-----|------|---|---|---|---|---|---|----|
| TICEUL1 | SID | TSID |   |   |   |   |   |   |    |

Example:

|         |     |     |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|
| TICEUL1 | 300 | 200 |  |  |  |  |  |  |  |
|---------|-----|-----|--|--|--|--|--|--|--|

| Describer | Meaning                                                                        |
|-----------|--------------------------------------------------------------------------------|
| SID       | Unique TICEUL1 number referenced from a PEULER1 entry. (Integer > 0; Required) |
| TSID      | Group of geometric region TICREG ID. (Integer > 0; Required)                   |

Remarks:

1. Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When a Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.

## TICEUDS

User-defined Transient Initial Conditions of Euler Elements or Lagrangian grid points.

Defines the initial values of element or grid point variables at the beginning of the analysis by a user written subroutine. Use in SOL700 only.

Format:

| 1       | 2   | 3     | 4     | 5     | 6    | 7 | 8 | 9 | 10 |
|---------|-----|-------|-------|-------|------|---|---|---|----|
| TICEUDS | SID | GROUP | UNAME | SETID | COPT |   |   |   |    |

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE initex 'SCAI.MDSolver.Obj.Uds.Dytran.InitOut '
```

*In Bulk Data:*

| 1       | 2  | 3      | 4      | 5   | 6    | 7 | 8 | 9 | 10 |
|---------|----|--------|--------|-----|------|---|---|---|----|
| TICEUDS | 12 | initex | EXINIT | 100 | GRID |   |   |   |    |

| Describer | Meaning                                                                                          |
|-----------|--------------------------------------------------------------------------------------------------|
| SID       | Unique output number. (Integer > 0; Required)                                                    |
| GROUP     | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default)       |
| UNAME     | User subroutine name associated with the entry. (Character; default=EXINIT)                      |
| SETID     | Number of a SET1 entry defining the elements or grids to be initialized. (Integer > 0, Required) |
| COPT      | Flag for assigning initial velocities. (Character; default=Element)                              |
|           | Element            Apply the initial variables on elements                                       |
|           | GRID              Apply the initial variables on grids                                           |

Remarks:

1. Only can be used for SOL 700.
2. Initial conditions to be used must be selected in the Case Control Section (TIC = SID).
3. UNAME=EXINIT can only be used.

## TICREG

## Transient Initial Conditions of Eulerian Regions

Defines the initial values sets for Eulerian regions. The Eulerian regions are defined by geometric shapes. Used in SOL 700 only.

Format:

|        |      |      |      |     |     |      |       |   |    |
|--------|------|------|------|-----|-----|------|-------|---|----|
| 1      | 2    | 3    | 4    | 5   | 6   | 7    | 8     | 9 | 10 |
| TICREG | TRID | TSID | TYPE | VID | MID | TSID | LEVEL |   |    |

Example:

|        |     |     |        |     |     |   |     |  |  |
|--------|-----|-----|--------|-----|-----|---|-----|--|--|
| TICREG | 300 | 200 | SPHERE | 400 | 100 | 3 | 4.0 |  |  |
|--------|-----|-----|--------|-----|-----|---|-----|--|--|

| Describer | Meaning                                                                                                |
|-----------|--------------------------------------------------------------------------------------------------------|
| TRID      | Unique TRID number. (Integer > 0; Required)                                                            |
| TSID      | ID of group of Euler regions referenced from the TICEUL1 entry. (Integer > 0; Required)                |
| TYPE      | The type of Eulerian region. (Character; Required)                                                     |
|           | SURF                      Region inside or outside a surface.                                          |
|           | SPHERE                    Region inside a sphere.                                                      |
|           | CYLINDER                Region inside a cylinder.                                                      |
|           | BOX                        Region inside a box.                                                        |
|           | ELEM                      Region defined by list of Euler elements.                                    |
| VID       | ID of a geometric entity. (Integer > 0; Required)                                                      |
|           | <u>Type:</u> <u>Region:</u>                                                                            |
|           | SURF                      SURFINI                                                                      |
|           | SPHERE                  SPHERE                                                                         |
|           | CYLINDER                CYLINDR                                                                        |
|           | BOX                        BCBOX                                                                       |
|           | ELEM                      SET1                                                                         |
| MID       | ID of a MATDEUL entry defining this material. (Integer ≥ 0; Default = 0)                               |
| TSID      | ID of a TICVAL entry containing a list of initial values for this material. (Integer ≥ 0; Default = 0) |
| LEVEL     | Level indicator for this material and initial values. (Real; Default = 0.0)                            |

Remarks:

1. A number of TICREG may exist in the input file with the same TSID. The TICEUI entry will combine all TICREGs with the same TSID into one initial definition for the Euler elements that are referenced from the same PEULER1 definition.
2. When the material number is left blank or zero, the Eulerian elements inside the region will be void. Note that this is not allowed in the Riemann solution-based Euler solvers, as they will not handle void elements. If you define void elements and select either the 1stOrder or 2ndOrder scheme, an error message will be issued and the analysis will stop.
3. All level indicators LEVEL of the same TSID group must have different values. The level indicator can be negative.
4. See also the parameter MICRO for the accuracy of the initial value generation.



## TICVAL

## Transient Initial Condition Set

Defines the initial values of an Eulerian geometric region. Used in SOL 700 only.

Format:

| 1      | 2     | 3      | 4      | 5      | 6     | 7      | 8     | 9      | 10 |
|--------|-------|--------|--------|--------|-------|--------|-------|--------|----|
| TICVAL | TSID  | METHOD | NAME1  | VALUE1 | NAME2 | VALUE2 | NAME3 | VALUE3 |    |
|        | NAMEi | VALUEi | -etc.- |        |       |        |       |        |    |

Example:

|        |      |     |         |      |      |     |     |     |  |
|--------|------|-----|---------|------|------|-----|-----|-----|--|
| TICVAL | 3    |     | DENSITY | 100. | YVEL | 25. | SIE | 3.7 |  |
|        | XVEL | 3.5 |         |      |      |     |     |     |  |

| Describer | Meaning                                                                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| TSID      | Unique TICVAL number referenced from a TICEUL entry. (Integer > 0; Required)                                                                                                                                                                |
| METHOD    | Type of input definition. (Character; Default = NORMAL)<br><br>RADIAL     Initializes material with radial profiles. The entries VALUEi are interpreted as TABLED1 IDs. See Remarks 2., 3., and 4.<br><br>NORMAL     Normal initialization. |
| NAMEi     | Variable to be initialized. See TICEL. (Character; Required)                                                                                                                                                                                |
| VALUEi    | Value of the variable. (Real; Required)                                                                                                                                                                                                     |

Remarks:

1. Element variables for Eulerian elements can be initialized with a TICEL or a TICEUL1 entry. The TICEL entry initializes a set of elements, while the TICEUL1 entry initializes either a set of elements or geometrical regions (sphere, cylinder, ...). When an Euler element is part of both a TICEL and a TICEUL1 entry, the TICEL entry takes precedence and overrules the TICEUL1 initialization for the element.
2. METHOD = RADIAL allows to map results of a spherical symmetric 1-D solution onto a full 3-D model. For initialized variables SIE and DENSITY, a 1-D table has to be defined that specifies the variable value for a number of distances from the center. The center is by default (0,0,0) but can be changed by setting X-CENTER, Y-CENTER, Z-CENTER. The velocity is a radial velocity and has to be specified as R-VEL. Its values is also a TABLED1 ID.
3. DYPARAM,SPHERSYM can be used to define a proper 1-D spherical mesh and speeds up the run by taking only the mesh-size in radial direction into account.

4. Radial initialization of JWL is supported. The entries DETSPH and the JWL entry from the 1-D spherical solution stage have to be included in the remap run. Alternatively, the 1-D solution may be run with JWL and the follow-up run with ideal gas, provided that all JWL material has fully ignited. Radial initialization of EOSIG is not supported. In the follow-up run, ideal gas material has to be used instead of IG material.

## TIC3Transient Analysis Initial Velocity with Increment Options

Allows for the definition of a velocity field of grid points consisting of a rotation and a translation specification. Used in SOL 700 only.

Format:

| 1    | 2    | 3    | 4    | 5     | 6    | 7    | 8 | 9 | 10 |
|------|------|------|------|-------|------|------|---|---|----|
| TIC3 | SID  | G    |      | SCALE |      |      |   |   | +  |
| +    | XVEL | YVEL | ZVEL | XROT  | YROT | ZROT |   |   | +  |
| +    | G1   | G2   | THRU | G3    | BY   | G4   |   |   |    |

Example:

|      |       |   |      |      |    |      |  |  |   |
|------|-------|---|------|------|----|------|--|--|---|
| TIC3 | 7     | 5 |      | 10.  |    |      |  |  | + |
| +    | 100.0 |   |      | 5.0  |    | -7.5 |  |  | + |
| +    | 1     | 2 | THRY | 1000 | BY | 23   |  |  |   |

| Describer        | Meaning                                                                                                                                         |
|------------------|-------------------------------------------------------------------------------------------------------------------------------------------------|
| SID              | Number of a set of loads. (Integer > 0; Required)                                                                                               |
| G                | Number of a grid point at the center of rotation. (Integer > 0; Required)                                                                       |
| SCALE            | Scale factor of initial velocity. (Real, Default=1.0)                                                                                           |
| XVEL, YVEL, ZVEL | Initial translational velocity components. (Real; Default = 0.0)                                                                                |
| XROT, YROT, ZROT | Initial rotational velocity components. (Real; Default = 0.0)                                                                                   |
| G1, G2, ...      | Grid points to be initialized. THRU indicates a range of grid points. BY is the increment to be used within this range. (Integer > 0; Required) |

Remarks:

1. Any number of TIC3 entries can be used.
2. The rotational velocity components are defined in radians per unit time.
3. For six degree of freedom grid points, the angular velocity components are also initialized.
4. Initial conditions for grid points that are not specified on TICn entries are assumed to be zero.
5. If the THRU specification is used, the grid points in the range definition are required to exist. The BY option enables grid points to be ignored in this range.
6. None of the fields in the list of grid points can be blank or zero, since this indicates the end of the list.
7. The initial conditions to be used in SOL 700 must be selected in the Case Control Section (IC = SID).

8. If grid points are part of a rigid body, it is recommended you enable double precision in SOL700. It is possible that a single precision SOL700 analysis will not assign the correct initial velocities.

## TIM2PSD

## FFT conversion tool for use in random vibration fatigue analysis

Fast Fourier Transformation (FFT) conversion tool for use in SOL 108 or SOL 111 random vibration fatigue analysis. This entry is used to convert time history data into power spectral density (PSD) functions directly input to the fatigue analysis.

## Format:

| 1       | 2        | 3       | 4      | 5      | 6      | 7     | 8     | 9          | 10 |
|---------|----------|---------|--------|--------|--------|-------|-------|------------|----|
| TIM2PSD | ID       | SRATE   | WINDOW | FORMAT | MEANS  | NSKIP |       |            |    |
|         | “EVENT”  | EVNTID1 |        |        |        |       | T1    | $\delta_1$ |    |
|         |          | EVNTID2 |        |        |        |       | T2    | $\delta_2$ |    |
|         |          | ...     |        |        |        |       |       |            |    |
|         |          | EVNTIDn |        |        |        |       | Tn    | $\delta_n$ |    |
|         | “DELETE” | EVNTID1 | Ti1_1  | Tf1_1  | Ti2_1  | Tf2_1 | Ti3_1 | Tf3_1      |    |
|         |          | EVNTID2 | Ti1_2  | Tf1_2  | Ti2_2  | Tf2_2 | Ti3_2 | Tf3_2      |    |
|         |          | ...     |        |        |        |       |       |            |    |
|         |          | EVNTIDn | Ti1_n  | Tf1_n  | Ti2_n  | Tf2_n | Ti3_n | Tf3_n      |    |
|         | “MAP”    | LCID1   | CHAN1  | LCID2  | CHAN2  | LCID3 | CHAN3 |            |    |
|         |          | LCID4   | CHAN4  | ...    | -etc.- |       |       |            |    |

## Example:

|         |        |       |         |     |     |   |     |     |  |
|---------|--------|-------|---------|-----|-----|---|-----|-----|--|
| TIM2PSD | 42     | 512.0 | HANNING | CSV | YES | 1 |     |     |  |
|         | EVENT  | 80001 |         |     |     |   | 2.0 | 1.0 |  |
|         |        | 80002 |         |     |     |   | 1.0 | 0.5 |  |
|         |        | 80003 |         |     |     |   | 2.0 | 1.0 |  |
|         | DELETE | 80001 | 1.1     | 1.9 |     |   |     |     |  |
|         | MAP    | 1     | 3       | 2   | 5   | 3 | 7   |     |  |

| Describer | Meaning                                                                                                                                                           |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Unique identification number referenced by FATIGUE case control, otherwise TIM2PSD entry is ignored (Integer > 0).                                                |
| SRATE     | Number of samples per second (required) - thus $dt = 1/SRATE$ (Real; No Default).                                                                                 |
| WINDOW    | Window function to use. Choices are HANNING or NONE. This is applied to the "block" of data extracted from the total time signal. (Character; Default = HANNING). |

| Descriptor | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                 |
|------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| FORMAT     | Format of time signal files (RPC or CSV) (Character; Default = CSV). RPC files are binary channel data usually with the .rsp file extension. CSV files are typical MS Excel comma separated files in which the channels are in the columns (i.e. three columns of data is three channels of data - y-values only. x-values assumed to be in ascending order with no regards to length of time between points).                                                                                                                                                                                                                                                                                                                                                                                                                          |
| MEANS      | Used to decide if overall signal mean values are to be calculated (YES or NO) (Character; Default = NO). Currently this is only calculated and not applied in the analysis. Ignored if no mean stress correction specified on FTGPARM entry.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                            |
| NSKIP      | Number of header lines to skip if an FORMAT=CSV file (Integer>=0; Default=0).<br>The next continuation lines are required and repeat for as many Events as necessary.                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                                   |
| EVENT      | Flag indicating that event parameters are to follow (one optional set for each event).<br>EVNTID <sub>i</sub> Number of this Event - must correspond to the TID of a FTGLOAD entry of TYPE=PSD that is referenced by an active FTGEVNT entry. (Note: it is NOT the actual FTGEVNT ID).<br>T <sub>i</sub> Length of window function in time for this Event (Real>0.0; no Default).<br>δ <sub>i</sub> Overlap or gap in time between windows for this Event (real) (+ means overlap) (Real>=0.0; Default = 0.0).<br>The next continuation lines are optional and can repeat for as many Events as necessary.                                                                                                                                                                                                                              |
| DELETE     | Flag indicating that event parameters are to follow for signal deletion. For any Event, time value pairs can be specified for deleting up to three (3) portions of the time signal for each Event.<br>EVNTID <sub>i</sub> Number of this Event - correspond to a previously defined EVNTID <sub>i</sub> in the "EVENT" section above.<br>T <sub>i_1</sub> Used to specify sections in an Event (defined by pairs of time values t1-t2, t3-t4, t5-t6) to delete from Event files before FFT process is applied (Real>0; t1 <= t2 <= t3 < t4 <= ...; no Defaults; if none are specified, entire signal is used). Only three (3) delete pairs per event are currently supported.<br>T <sub>f_i_1</sub><br>The next continuation lines are optional and only necessary if the SUBCASEs corresponding to the channel data is not one-to-one. |
| MAP        | Flag used to map the channel data and load case (SUBCASE) IDs.<br>LCID <sub>i</sub> SUBCASE of transfer function (TF) corresponding to ith load event. SUBCASEs must be in ascending order in the Case Control! There cannot be more SUBCASEs listed than there are channels of data in the CSV/RPC files.<br>CHAN <sub>i</sub> Channel # corresponding to ith load event. Channels must be referenced in ascending order and cannot be repeated.                                                                                                                                                                                                                                                                                                                                                                                       |

Remarks:

1. TIM2PSD bulk data entries are ignored if not selected by a FATIGUE case control.

2. The actual RPC/CSV files are specified with a UDNAME entry referenced by TID field on the FTGLOAD entry for each Event (FTGEVNT). This is a requirement in the presence of a TIM2PSD entry.
3. All events must be in the same format, use the same window function, have the same sample rate, and have the same number and order of channels.
4. Each event can be a different length and can have a different block length and gap.
5. If LOGLVL=1 or 2 on the FTGPARM entry, the input time histories and direct PSD computations are written to CSV files for possible plotting with MS Excel.
6. The TID on the FTGLOAD is also used as the SID of the auto-generated RANDPS entries in this process.
7. A file containing the auto-generated RANDPS and TABRND1 entries is created in the same directory as the referenced UDNAME filename. This file can then be used as an include file for subsequent analyses that use the same generated PSD data rather than using the TIM2PSD entry again in subsequent runs.
8. This process also determines the overall mean values of the time histories for each event. The means are normally thrown away in conversion to PSDs and cross-PSDs. Currently the mean effect is not taken into account in the subsequent fatigue analysis. These means are appended to the end of the file containing the auto-generated RANDPS and TABRND1 entries.
9. MAP flag is not necessary if all channels in RPC/CSV file are used in the exact same order as the SUBCASEs corresponding to the transfer functions. SUBCASE IDs must be in ascending order. Channels must be referenced in ascending order.

## TIRE Selection of Tires used in the Analysis

The TIRE bulk data entry facilitates the selection of all the CDTire/NVH tires to be used in the simulation. This is done by specifying the name of each CDTire/NVH tire that is to be included in the simulation on the tire bulk data entry.

Format:

| 1    | 2        | 3      | 4      | 5                  | 6      | 7      | 8      | 9   | 10  |
|------|----------|--------|--------|--------------------|--------|--------|--------|-----|-----|
| TIRE | ID       |        |        |                    |        |        |        |     |     |
|      | “CDTIRE” | TNAME1 | TNAME2 | TNAME3             | TNAME4 | TNAME5 | TNAME6 | ... | ... |
|      | ...      | ...    | ...    | TNAME <sub>n</sub> |        |        |        |     |     |

where, n is the number of CDTire/NVH tires to be used in the simulation.

Example:

|      |        |       |       |       |       |  |  |  |  |
|------|--------|-------|-------|-------|-------|--|--|--|--|
| TIRE | 100    |       |       |       |       |  |  |  |  |
|      | CDTIRE | TIRFL | TIRFR | TIRRL | TIRRR |  |  |  |  |

| Describer          | Meaning                                                                                                                                                 |
|--------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID                 | Set identification number referenced by TIRE case control command. (Integer > 0; Required).                                                             |
| “CDTIRE”           | Keyword for indicating that CDTire/NVH tires are used in the model.                                                                                     |
| TNAME <sub>i</sub> | The name of the <sup>i</sup> <sup>th</sup> CDTire/NVH tire. This name is specified in the DTI entry exported by CDTire while generating the tire model. |



## TLOAD1 Transient Response Dynamic Excitation, Form 1

Defines a time-dependent dynamic load or enforced motion of the form

$$\{P(t)\} = \{A\} \cdot F(t - \tau)$$

for use in transient response analysis.

Format:

| 1      | 2   | 3        | 4                 | 5    | 6     | 7   | 8   | 9 | 10 |
|--------|-----|----------|-------------------|------|-------|-----|-----|---|----|
| TLOAD1 | SID | EXCITEID | DELAYI/<br>DELAYR | TYPE | TID/F | US0 | VS0 |   |    |

Example:

|        |   |   |    |      |    |  |  |  |  |
|--------|---|---|----|------|----|--|--|--|--|
| TLOAD1 | 5 | 7 | 15 | LOAD | 13 |  |  |  |  |
|--------|---|---|----|------|----|--|--|--|--|

| Describer | Meaning                                                                                                                                                                              |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Set identification number. (Integer > 0) See Remarks 1. and 5.                                                                                                                       |
| EXCITEID  | Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$ . See Remarks 2. and 3. (Integer > 0) |
| DELAYI    | Identification number of DELAY Bulk Data entry that defines time delay $\tau$ . See Remark 8. and 13. (Integer > 0 or blank)                                                         |
| DELAYR    | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 8. and 13. (Real or blank)                           |
| TYPE      | Defines the type of the dynamic excitation. See Remarks 2., 3. and 12. (Integer, character or blank; Default = 0)                                                                    |
| TID/F     | Identification number of TABLEDi entry that gives $F(t)$ . (Integer > 0). Value of F to be used for all times. (Real, non-zero)                                                      |
| US0       | Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9., 11. and 14. (Real; Default = 0.0)                                                               |
| VS0       | Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10., 11. and 14. (Real; Default = 0.0)                                                                 |

Remarks:

1. Dynamic excitation sets must be selected with the Case Control command DLOAD = SID.

2. The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

| TYPE                                                                                                     | TYPE of Dynamic Excitation                                                       |
|----------------------------------------------------------------------------------------------------------|----------------------------------------------------------------------------------|
| 0, L, LO, LOA or LOAD                                                                                    | Applied load (force or moment) (Default)                                         |
| 1, D, DI, DIS, or DISP                                                                                   | Enforced displacement using large mass or SPC/SPCD data                          |
| 2, V, VE, VEL or VELO                                                                                    | Enforced velocity using large mass or SPC/SPCD data.                             |
| 3, A, AC, ACC or ACCE                                                                                    | Enforced acceleration using large mass or SPC/SPCD data                          |
| 4                                                                                                        | FLOW boundary condition on the face of an Eulerian solid element (SOL 700 only). |
| 12                                                                                                       | Velocity of the center of gravity of a rigid body (SOL 700 only)                 |
| 13                                                                                                       | Force or moment on the center of gravity of a rigid body (SOL 700 only).         |
| For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400. |                                                                                  |

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

3. TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control  
 EXCITEID may reference DAREA, static and thermal load set entries
- There is a LOADSET request in Case Control  
 The program may reference DAREA entries as well as static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control  
 EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.
- There is a LOADSET request in Case Control  
 The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

4. EXCITEID may reference sets containing QHBDY, QBDYi, QVECT, QVOL and TEMPBC entries when using the heat transfer option.
5. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
6. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
7. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. Please refer to [Fourier Transform](#) in the *MSC Nastran Dynamic Analysis User's Guide*.
8. If the DELAYI/DELAYR field is blank or zero,  $\tau$  will be zero.
9. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(US0)$  where  $\{A\}$  is defined by the EXCITEID field.
10. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(VS0)$  where  $\{A\}$  is defined by the EXCITEID field.
11. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
12. For TYPE=4, TID must be blank if it references a FLOW entry. Use the FLOWT entry to define a time dependent flow boundary on the face of an Eulerian element.
13. For RC network solver in thermal analysis, the DELAY1/DELAYR is ignored.
14. Fields US0 and VS0 are not supported in SOL 700.
15. DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.
16. If Modules are present then this entry may only be specified in the main Bulk Data section.
17. If enforced motion is applied in a modal transient solution the default solution algorithm is the COUPLED method, rather than the UNCOUPLED method. If you wish to use the UNCOUPLED solver in a modal transient solution with enforced motion, specify PARAM,NONCUP,-2 in your input file. When using the COUPLED solver in transient response, it is recommended to have at least two timesteps with 0.0 loading before starting your load.

## TLOAD2

## Transient Response Dynamic Excitation, Form 2

Defines a time-dependent dynamic excitation or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\} \tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

for use in a transient response problem, where  $\tilde{t} = t - T1 - \tau$

Format:

| 1      | 2   | 3        | 4                 | 5    | 6  | 7  | 8 | 9 | 10 |
|--------|-----|----------|-------------------|------|----|----|---|---|----|
| TLOAD2 | SID | EXCITEID | DELAYI/<br>DELAYR | TYPE | T1 | T2 | F | P |    |
|        | C   | B        | US0               | VS0  |    |    |   |   |    |

Example:

|        |     |    |     |  |     |     |      |  |  |
|--------|-----|----|-----|--|-----|-----|------|--|--|
| TLOAD2 | 4   | 10 | 5.0 |  | 2.1 | 4.7 | 12.0 |  |  |
|        | 2.0 |    |     |  |     |     |      |  |  |

| Descriptor | Meaning                                                                                                                                                                              |
|------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID        | Set identification number. See Remarks 1. and 6. (Integer > 0)                                                                                                                       |
| EXCITEID   | Identification number of a static load set or a DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines $\{A\}$ . See Remarks 2. and 3. (Integer > 0) |
| DELAYI     | Identification number of DELAY Bulk Data entry that defines time delay $\tau$ . See Remark 5. (Integer > 0 or blank)                                                                 |
| DELAYR     | Value of time delay $\tau$ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)                                   |
| TYPE       | Defines the type of the dynamic excitation. See Remarks 2. and 3. (Integer; character or blank; Default = 0)                                                                         |
| T1         | Time constant. (Real $\geq 0.0$ )                                                                                                                                                    |
| T2         | Time constant. (Real; T2 > T1)                                                                                                                                                       |
| F          | Frequency in cycles per unit time. (Real $\geq 0.0$ ; Default = 0.0)                                                                                                                 |
| P          | Phase angle in degrees. (Real; Default = 0.0)                                                                                                                                        |
| C          | Exponential coefficient. (Real; Default = 0.0)                                                                                                                                       |
| B          | Growth coefficient. (Real; Default = 0.0)                                                                                                                                            |

| Describer | Meaning                                                                                                               |
|-----------|-----------------------------------------------------------------------------------------------------------------------|
| US0       | Factor for initial displacements of the enforced degrees-of-freedom. See Remarks 9, 11, and 16. (Real; Default = 0.0) |
| VSO       | Factor for initial velocities of the enforced degrees-of-freedom. See Remarks 10, 11, and 16. (Real; Default = 0.0)   |

**Remarks:**

- Dynamic excitation sets must be selected with the Case Control command with DLOAD=SID.
- The type of the dynamic excitation is specified by TYPE (field 5) according to the following table:

| TYPE                                                                                                     | TYPE of Dynamic Excitation                                               |
|----------------------------------------------------------------------------------------------------------|--------------------------------------------------------------------------|
| 0, L, LO, LOA or LOAD                                                                                    | Applied load (force or moment) (Default)                                 |
| 1, D, DI, DIS, or DISP                                                                                   | Enforced displacement using large mass or SPC/SPCD data                  |
| 2, V, VE, VEL or VELO                                                                                    | Enforced velocity using large mass or SPC/SPCD data                      |
| 3, A, AC, ACC or ACCE                                                                                    | Enforced acceleration using large mass or SPC/SPCD data                  |
| 12                                                                                                       | Velocity of the center of gravity of a rigid body. (SOL 700 only)        |
| 13                                                                                                       | Force or moment on the center of gravity of a rigid body. (SOL 700 only) |
| For enforced displacement, velocity and acceleration, the large mass method is not supported in SOL 400. |                                                                          |

The enforced motion options (SPC/SPCD) defined by TYPE=1, 2, 3 are currently used for SOLs 109, 112, 146, 200, and where applicable SOL 400.

- TYPE (field 5) also determines the manner in which EXCITEID (field 3) is used by the program as described below

Excitation specified by TYPE is applied load

- There is no LOADSET request in Case Control  
EXCITEID may reference DAREA, static and thermal load set entries
- There is a LOADSET request in Case Control  
The program may reference DAREA entries as well as static and thermal load set entries specified by the LID or TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID.

Excitation specified by TYPE is enforced motion

- There is no LOADSET request in Case Control  
EXCITEID will reference SPCD entries. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference DAREA and static and thermal load set entries just as in the case of applied load excitation.

- There is a LOADSET request in Case Control

The program will reference SPCD entries specified by the LID field in the selected LSEQ entry corresponding to EXCITEID. If such entries indicate null enforced motion, the program will then assume that the excitation is enforced motion using large mass and will reference static and thermal load set entries specified by the LID and TID fields, respectively, in the selected LSEQ entry corresponding to EXCITEID, just as in the case of applied load excitation.

4. EXCITEID (field 3) may reference sets containing QHBDY, QB DYi, QVECT, and QVOL and TEMPBC entries when using the heat transfer option.
5. If the DELAYI/DELAYR field is blank or zero,  $\tau$  will be zero.
6. SID need not be unique for all ACSRCE, RLOAD1, RLOAD2, TLOAD1 and TLOAD2 dynamic load entries. The DLOAD = SID Case Control command will select all dynamic load entries with the set identification of SID.
7. If the heat transfer option is used, the referenced QVECT entry may also contain references to functions of time, and therefore A may be a function of time.
8. If TLOADi entries are selected in SOL 111 or 146 then a Fourier analysis is used to transform the time-dependent loads on the TLOADi entries to the frequency domain and then combine them with loads from RLOADi entries. Then the analysis is performed as a frequency response analysis but the solution and the output are converted to and printed in the time domain. In this case, B will be rounded to the nearest integer. Please refer to [Fourier Transform](#) in the *MSC Nastran Dynamic Analysis User's Guide*.
9. The USO field is used only when the dynamic excitation defined by the TYPE field is enforced velocity or enforced acceleration using SPC/SPCD specification. The initial displacements for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(US0)$  where  $\{A\}$  is defined by the EXCITEID field.
10. The VS0 field is used only when the dynamic excitation defined by the TYPE field is enforced acceleration using SPC/SPCD specification. The initial velocities for the enforced degrees-of-freedom in this case are given by the product  $\{A\}(VS0)$  where  $\{A\}$  is defined by the EXCITEID field.
11. The initial conditions for the enforced degrees-of-freedom implied by the US0 and VS0 fields are distinct from, and may be used in conjunction with, the initial conditions for the independent degrees-of-freedom specified by a TIC Bulk Data entry (which, in turn, is selected by an IC Case Control command).
12. The continuation entry is optional.
13. TYPE=4 (SOL 700) is not supported using TLOAD2.
14. For SOL 700, TLOAD2 is converted to TLOAD1 and a TABLED1 is generated internally.
15. RC network solver does not support TLOAD2 for thermal analysis.
16. Fields US0 and VS0 are not supported in SOL 700.
17. DELAYI/DELAYR (Integer>0 or Real) is not supported for the follower force in SOL129 and SOL400. The corresponding loads are neglected in the analysis.

18. If comparing results between pre-2005 versions and later versions, please note that results are shifted by one time step in later versions. The tload2 should be updated accordingly if there is a need to compare to these pre-2005 versions.
19. If Modules are present then this entry may only be specified in the main Bulk Data section.
20. If enforced motion is applied in a modal transient solution the default solution algorithm is the COUPLED method, rather than the UNCOUPLED method. If you wish to use the UNCOUPLED solver in a modal transient solution with enforced motion, specify PARAM,NONCUP,-2 in your input file. When using the COUPLED solver in transient response, it is recommended to have at least two timesteps with 0.0 loading before starting your load.

## TMPSET

## Temperature Group Set Definition

Define a time-dependent dynamic thermal load group for use in TTEMP Bulk Data entry in SOL 400.

Format:

|        |    |    |    |    |    |    |    |    |    |
|--------|----|----|----|----|----|----|----|----|----|
| 1      | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
| TMPSET | ID | G1 | G2 | G3 | G4 | G5 | G6 | G7 |    |

Alternate Format:

|        |    |    |        |    |      |     |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|
| TMPSET | ID | G1 | “THRU” | G2 | “BY” | INC |  |  |  |
|--------|----|----|--------|----|------|-----|--|--|--|

The Continuation Entry formats may be used more than once and in any order. They may also be used with either format above.

Continuation Entry Format 1:

|  |    |    |     |     |        |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|
|  | G8 | G9 | G10 | G11 | -etc.- |  |  |  |  |
|--|----|----|-----|-----|--------|--|--|--|--|

Continuation Entry Format 2:

|  |    |        |    |      |     |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|
|  | G8 | “THRU” | G9 | “BY” | INC |  |  |  |  |
|--|----|--------|----|------|-----|--|--|--|--|

Example:

|        |    |      |      |    |    |    |  |  |  |
|--------|----|------|------|----|----|----|--|--|--|
| TMPSET | 15 | 5    | THRU | 21 | BY | 4  |  |  |  |
|        | 27 | 30   | 32   | 33 |    |    |  |  |  |
|        | 35 | THRU | 44   |    |    |    |  |  |  |
|        | 67 | 68   | 72   | 75 | 84 | 93 |  |  |  |

| Describer | Meaning                                                       |
|-----------|---------------------------------------------------------------|
| ID        | Temperature group identification number. (Integer > 0)        |
| Gi        | Grid point Identification numbers in the group. (Integer > 0) |

Remarks:

1. This entry is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0).
2. GROUP\_ID determines the group of a specified the time-dependent distribution of temperatures. It is used by the TTEMP Bulk Data entry to define the corresponding TABLEDi entry. GROUP\_ID must be unique for all of the other TMPSET entries.
3. The temperature of grid point Gi must be defined using TEMP, TEMPD, TEMPP1, or TEMPRB Bulk Data entry. These bulk data entries must have the same SID as that referenced on the associated TTEMP Bulk Data entry.



**TODYNA****Defines the Start of Direct Text to Dytran**

All bulk data entries between TODYNA and ENDDYNA will be passed directly from SOL 700 to Dytran.  
Used in SOL 700 only.

Format:

| 1      | 2 | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|---|---|---|---|---|---|---|---|----|
| TODYNA |   |   |   |   |   |   |   |   |    |

Example:

|        |     |        |  |       |        |  |  |  |  |
|--------|-----|--------|--|-------|--------|--|--|--|--|
| TODYNA |     |        |  |       |        |  |  |  |  |
| MAT1   | 345 | 29.0E6 |  | 0.285 | 0.0004 |  |  |  |  |
| ENDDYN |     |        |  |       |        |  |  |  |  |

## TOMVAR Topometry Design Variable

Defines a design region for topometry optimization (element-by-element optimization).

Format:

| 1      | 2        | 3     | 4   | 5         | 6     | 7   | 8   | 9     | 10 |
|--------|----------|-------|-----|-----------|-------|-----|-----|-------|----|
| TOMVAR | ID       | TYPE  | PID | PNAME/FID | XINIT | XLB | XUB | DELXV |    |
|        | "DLINK"  | TID   | C0  | C1        |       |     |     |       |    |
|        | "DDVAL"  | DSVID |     |           |       |     |     |       |    |
|        | "STRESS" | STLIM |     |           |       |     |     |       |    |

Example:

Design all element's thickness referencing PSHELL ID = 5 with initial design = 10.0 ( $t_0 = 10.0$  input element thickness), lower bound  $0.5 \cdot t_0$  and upper bound  $1.5 \cdot t_0$ .

|        |        |     |        |   |   |      |  |  |  |
|--------|--------|-----|--------|---|---|------|--|--|--|
| TOMVAR | 1      | PS1 | PSHELL | 5 | T | 10.0 |  |  |  |
|        | STRESS | 5.0 |        |   |   |      |  |  |  |

Example:

Design all element's Young Modulus referred by PSHELL ID = 100 with initial design XINIT = 3.E+5, XLB=1.0, and XUB= 1.0E+6.

|        |    |        |     |   |       |     |       |  |  |
|--------|----|--------|-----|---|-------|-----|-------|--|--|
| TOMVAR | 10 | PSHELL | 100 | E | 3.E+5 | 1.0 | 1.E+6 |  |  |
|--------|----|--------|-----|---|-------|-----|-------|--|--|

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                              |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Unique topometry design region identification number. (Integer > 0)                                                                                                                                                                                                                                                                                                                                                  |
| TYPE      | Property entry type. Used with PID to identify the elements to be designed. See Remark 2. (Character: "PBAR", "PSHELL", "PSOLID", and "PCOMP", etc.)                                                                                                                                                                                                                                                                 |
| PID       | Property entry identifier (Integer > 0). This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1, DVPREL2, DVMREL1, and DVMREL2 entries. Topometry, topology, and sizing variables cannot share the same properties. (Integer > 0). Combined topometry, topology, topography, sizing, and shape variables are allowed.                                                                                  |
| PNAME/FID | Property name or property material name, such as "T", "A", "E", and "GE", or field position of the property entry or word position in the element property table of the analysis model. Property names that begin with an integer such as 12I/T**3 may only be referenced by field position. Only one property value for each property can be designed by TOMVAR in a job. See Remark 2. (Character or Integer > 0.) |

| Describer | Meaning                                                                                                                                                                                                            |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| XINIT     | Initial value. (Real or blank, no Default). Typically, XINIT is defined to match the mass target constraint (so the initial design does not have violated constraints) or the analysis model input property value. |
| XLB       | Lower bound. (Real or blank; Default = blank) . The default is $XLB=0.5*XINIT$ .                                                                                                                                   |
| XUB       | Upper bound . (Real or blank; Default = blank). The default is $XLB=1.5*XINIT$ .                                                                                                                                   |
| DELXV     | Fractional change allowed for the design variable during approximate optimization. See Remark 6. (Real > 0.0; Default = 0.5)                                                                                       |
| “DLINK”   | Indicates that this line relates a ply thickness to another ply thickness. See Remark 10.                                                                                                                          |
| TID       | TOMVAR entry identifier. (Integer > 0)                                                                                                                                                                             |
| C0        | Constant term. (Real; Default = 0.0)                                                                                                                                                                               |
| C1        | Coefficient term. (Real; no Default)                                                                                                                                                                               |
| “DDVAL”   | Indicates that this line defines discrete TOMVAR variables.                                                                                                                                                        |
| DSVID     | DDVAL entry identifier. (Integer > 0)                                                                                                                                                                              |
| “STRESS”  | Indicates that this line defines a stress limit.                                                                                                                                                                   |
| STLIM     | Von Mises stress upper bound. See Remark 11. (Real >0.0)                                                                                                                                                           |

### Remarks:

1. Multiple TOMVAR's are allowed in a single file.
2. Property name and FID > 0 can be used for element property values just like the DVPREL1 Bulk Data entry. Only property name can be used for material property values like DVMREL1.
3. Property name “A” is shared by material and property entries. If “A” is used for PNAME, it is a Material. For PROD, PBEAM and PBAR, the integer field ID must be used to point to the area of these entries. TOMVAR supports MAT1 only.
4. PBARL, PBEAML, PBRSECT, PCOMPG and PBMSECT are not supported.
5. For TYPE = PSHELL and PNAME selecting a material property, all the MATi fields on the PSHELL must be the same.
6. Combined topometry, topography, topology, sizing, and shape optimization is supported in a single file. However, topometry and topology cannot reference the same property ID. It is possible to topometry certain elements while sizing others. It is allowed to simultaneously design the same elements with topometry and desvar (sizing and/or shape) variables but topometry and sizing cannot reference the same property name.
7. Topometry optimization with element response constraints is slow due to many design variables. In this case, fully stressed design (FSD) is an alternative for certain problems
8. Parameters DESPCH and DESPCH1 specify when the topometry optimized design values are written to the element result history file jobname.des that can be imported to Patran and other post-processor to view topometry optimized results.
9. The TOMVAR entry cannot be used with thermal loads.

10. The DLINK line can only be used when TYPE=PCOMP. The PID must be the same as the PID given on the TOMVAR referenced by TID and the PNAMEs in the two TOMVAR entries must differ
11. "STRESS" limits can only be used for PTYPE=PSHELL and PNAME= T. The Von Mises stress (at element center) constraints apply to all shell elements in both designed and non-designed regions. All TOMVAR entries must have the same STLIM.

## TOPVAR Topological Design Variable

Define a topology design region for topology optimization.

Format:

| 1      | 2        | 3     | 4     | 5     | 6     | 7     | 8     | 9   | 10 |
|--------|----------|-------|-------|-------|-------|-------|-------|-----|----|
| TOPVAR | ID       | LABEL | PTYPE | XINIT | XLB   | DELXV | POWER | PID |    |
|        | "SYM"    | CID   | MS1   | MS2   | MS3   | CS    | NCS   |     |    |
|        | "CAST"   | CID   | DD    | DIE   | ALIGN |       |       |     |    |
|        | "EXT"    | CID   | ED    | ALIGN |       |       |       |     |    |
|        | "PRINT"  | CID   | PD    |       |       |       |       |     |    |
|        | "TDMIN"  | TVMIN | TVMAX |       |       |       |       |     |    |
|        | "STRESS" | STLIM |       |       |       |       |       |     |    |

Example:

|        |        |     |        |   |   |      |   |   |  |
|--------|--------|-----|--------|---|---|------|---|---|--|
| TOPVAR | 1      | PS1 | PSHELL | 5 | T | 10.0 | 4 | 1 |  |
|        | STRESS | 5.0 |        |   |   |      |   |   |  |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                               |
|-----------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Unique topology design region identification number. (Integer > 0)                                                                                                                                                                                                                                                                                                                                    |
| LABEL     | User-supplied name for printing purpose. (Character)                                                                                                                                                                                                                                                                                                                                                  |
| PTYPE     | Property entry name. Used with PID to identify the elements to be designed. (Character: "PBAR", "PSHELL", "PSOLID", etc.)                                                                                                                                                                                                                                                                             |
| XINIT     | Initial value. (Blank or Real, $XLB < XINIT \leq 1.0$ ; Default = Blank). Typically, XINIT is defined to match the mass constraint on DRESP1=FRMASS, so the initial design does not have violated constraints. In this case, the default is set to the constraint value. If the mass (DRESP1=FRMASS or WEIGHT) is the objective, the default is 0.9. The default of XINIT is 0.6 for the other cases. |
| XLB       | Lower bound to prevent the singularity of the stiffness matrix. (Real; Default = 0.001)                                                                                                                                                                                                                                                                                                               |
| DELXV     | Fractional change allowed for the design variable during approximate optimization. See Remark 3. (Real > 0.0; Default = 0.2)                                                                                                                                                                                                                                                                          |
| POWER     | A penalty factor used in the relation between topology design variables and element Young's modulus. (Real > 1.0; Default = 3.0, or 4.0 if symmetry constraints used). $2.0 \leq POWER \leq 5.0$ is recommended.                                                                                                                                                                                      |
| PID       | Property entry identifier. This PID must be unique for PIDs referenced by other TOPVAR, DVPREL1 and DVPREL2 entries. Topology and sizing variables cannot share the same properties. (Integer > 0)                                                                                                                                                                                                    |
| "SYM"     | Indicates that this line defines symmetry constraints.                                                                                                                                                                                                                                                                                                                                                |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                  |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| CID       | Rectangular coordinate system ID used for specifying manufacturing constraints. See Remark 4. (Blank or Integer > 0; Default = blank)                                                                                                                                                                                                                                                                                                                                    |
| MSi       | Mirror symmetry plane. See Remark 5. & 7. (Character, 'XY', 'YZ', or 'ZX')                                                                                                                                                                                                                                                                                                                                                                                               |
| CS        | Cyclic symmetry axis. (character X, Y, Z). See Remark 12.                                                                                                                                                                                                                                                                                                                                                                                                                |
| NCS       | Number of cyclic symmetric segments in 360 degrees (Integer > 0). See Remark 9.                                                                                                                                                                                                                                                                                                                                                                                          |
| “CAST”    | Indicates that this line defines casting constraints (i.e., die draw direction constraints). See Remarks 6., 7., 8., and 10.                                                                                                                                                                                                                                                                                                                                             |
| DD        | Draw Direction. DDi=X, Y, Z or X-, Y-, Z- for a single die option (DIE=1) where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively. DDi=X, Y, and Z for two die option (DIE =2) (Character)                                                                                                                                                                                                                                                         |
| DIE       | Die Options. (Blank or integer 1 or 2; Default = 1)<br>= 1 (or blank). A single die will be used and the die slides in the given draw direction (i.e., material grows from the bottom in the draw direction)<br>= 2. Two dies will be used and the dies split apart along the draw direction (i.e., material grows from the splitting plane in opposite direction along the axis specified by the draw direction DDi. The splitting plane is determined by optimization) |
| ALIGN     | Indicates whether the designed property finite element mesh is precisely aligned with the draw direction or extrusion direction. (Character: “YES” or “NO” or Blank; Default = blank = “NO”) See Remark 10.                                                                                                                                                                                                                                                              |
| “EXT”     | Indicates that this line defines extrusion constraints (i.e., enforce constant cross-section) See Remark 6. and 7.                                                                                                                                                                                                                                                                                                                                                       |
| ED        | Extrusion direction. (Character, X, Y, or Z)                                                                                                                                                                                                                                                                                                                                                                                                                             |
| “PRINT”   | Indicates that this line defines overhang constraints (maximum overhang angle is 45 degree). See Remark 12.                                                                                                                                                                                                                                                                                                                                                              |
| CID       | Rectangular coordinate system ID used for specifying overhang constraints.                                                                                                                                                                                                                                                                                                                                                                                               |
| PD        | PRINT Direction. PD =X, Y, Z or X-, Y-, and Z- where X-, Y-, Z- indicates the opposite direction of X, Y, and Z respectively (Character).                                                                                                                                                                                                                                                                                                                                |
| “TDMIN”   | Indicates that this line defines a minimum and/or maximum member size., See remarks 11. and 12.                                                                                                                                                                                                                                                                                                                                                                          |
| TVMIN     | Minimum member size. See Remarks 11. and 12. (Real>=0.0 or blank)                                                                                                                                                                                                                                                                                                                                                                                                        |
| TVMAX     | Maximum member size. See Remarks 11. and 12. (Real > TVMIN or blank)                                                                                                                                                                                                                                                                                                                                                                                                     |
| “STRESS”  | Indicates that this line defines a stress limit.                                                                                                                                                                                                                                                                                                                                                                                                                         |
| STLIM     | Von Mises stress upper bound. See Remark 13.. (Real >0.0)                                                                                                                                                                                                                                                                                                                                                                                                                |

## Remarks:

1. The topologically designable element properties include PROD, PBAR, PBARL, PBEND, PBEAM, PBEAML, PSHELL, PSHEAR, PSOLID, and PWELD. Multiple TOPVAR's are allowed in a single file. Combined topology, topography (BEADVAR), topometry (TOMVAR) sizing, and shape optimization is supported in a single file. However, TOPVAR cannot be used with DVMREL1 and DVMREL2 entries.
2. All designed element properties must refer to a MAT1 entry or MAT9 entry (PSOLID only); therefore, a PCOMP/PCOMPG cannot be used as designed property in topology optimization. PCOMP/PCOMPG's can be used as non-designed properties in a topology optimization job.
3. If DELXV is blank, the default is taken from the specification of DELX parameter on the DOPTPRM entry.
4. Only CORD1R and CORD2R can be used as a referenced coordinate system to specify topology manufacturing constraints. Only one reference coordinate system CID is allowed for each TOPVAR entry.
5. One, two or three different mirror symmetry planes can present (such as MS1=XY, MS2=YZ, and MS3=ZX).
6. Casting ("CAST") and Extrusion ("EXT") manufacturability constraints can be applied to PTYPE="PSOLID" only. Casting constraints cannot be combined with extrusion constraints for the same TOPVAR entry.
7. Some symmetry constraint types can be combined with casting or extrusion constraints. The referenced coordinate system CID must be the same for the combined constraints. Some possible combinations are:
  - For "EXT" constraints, possible combinations are (ED=X, MSi=XY, and/or ZX or CS=X), (ED=Y, MSi=YZ, and/or XY or CS=Y), (ED=Z, MSi=ZX, and/or YZ or CS=Z).
  - For "CAST" constraints, possible combinations are (DD=X or X-, MSi=XY and/or ZX or CS=X), (DD=Y or Y-, MSi=YZ and/or XY or CS=Y), (DD=Z or Z-, MSi=ZX and/or YZ or CS=Z).
8. For two dies option (DIE=2), the splitting plane is optimized. For a single die DIE=1, the parting plane is the bottom surface of the designed part in the draw direction.
9. The first symmetry segment starts at the X-axis when CS = Z (at Z-axis when CS = Y, and at the Y-axis when CS = X). One cycle symmetry can be combined with one mirror symmetry constraint as long as the axis of cyclic symmetry is normal to the plane of mirror symmetry. For example, MSi = YZ and CS = X, MSi = XZ and CS = Y, and MSi = XY and CS = Z. This feature can also be used for < 360 degrees but NCS must be given in 360 degrees.
10. It is recommended to use aligned mesh for casting property due to smaller tolerance used.
11. Without a TDMIN continuation line, the minimum member size constraint is taken from the specification of TDMIN parameter on the DOPTPRM entry. This option is applied on 2 and 3 D elements only. Minimum member size constraints can be used with "SYM", "CAST", and "EXT" constraints.

12. TVMIN and TVMAX are dimensional quantities. A guideline is that TVMIN be at least three times a representative element dimension. TVMAX must be greater than TVMIN and it is recommended that it be twice as big. If TVMAX is blank, no maximum member size is imposed. It is recommended that TVMIN always be used when TVMAX is specified.
13. “STRESS” limits can only be used for PTYPE=PSOLID and PSHELL referencing MAT1 only. The Von Mises stress (at element center) apply to all solid and/or shell elements in both designed and non-designed regions. All TOPVAR entries must have the same STLIM.
14. The TOPVAR entry cannot be used with thermal loads.
15. For normal mode topology optimization, lower and higher mode may switch during optimization. This often occurs while maximizing or constraining the first eigenfrequency. This leads to a diverging solution. A workaround is using the mean value of a few of the lowest eigenfrequency (3-6) by DRESP2.
16. The CASI solver is strongly recommended for solid elements topology problems for efficiency.
17. “PRINT” (overhang constraints) can be used for PTYPE=PSOLID only and limited to one TOPVAR entry. The result is mesh dependent. If used with minimum member size, the location of reference coordinate system may have some influence on the optimal design.



## TRIM Trim Variable Constraint

Specifies constraints for aeroelastic trim variables.

Format:

| 1    | 2      | 3    | 4      | 5      | 6   | 7      | 8   | 9    | 10 |
|------|--------|------|--------|--------|-----|--------|-----|------|----|
| TRIM | SID    | MACH | Q      | LABEL1 | UX1 | LABEL2 | UX2 | AEQR |    |
|      | LABEL3 | UX3  | -etc.- |        |     |        |     |      |    |

Example:

|      |      |     |      |       |     |        |     |     |  |
|------|------|-----|------|-------|-----|--------|-----|-----|--|
| TRIM | 1    | 0.9 | 100. | URDD3 | 1.0 | ANGLEA | 7.0 | 0.0 |  |
|      | ELEV | 0.2 |      |       |     |        |     |     |  |

| Describer | Meaning                                                                                                                                             |
|-----------|-----------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Trim set identification number. (Integer > 0). See remarks 1 and 2.                                                                                 |
| MACH      | Mach number. (Real $\geq 0.0$ and $\neq 1.0$ )                                                                                                      |
| Q         | Dynamic pressure. (Real > 0.0)                                                                                                                      |
| LABELi    | The label identifying aerodynamic trim variables defined on an AESTAT, AESURF or AEPARM entry. (Character)                                          |
| UXi       | The magnitude of the aerodynamic extra point degree-of-freedom. (Real)                                                                              |
| AEQR      | Flag to request a rigid trim analysis (Real $\geq 0.0$ and $\leq 1.0$ ; Default =1.0). A value of 0.0 provides a rigid trim analysis, see Remark 5. |

Remarks:

1. The TRIM entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all the TRIM and TRIM2 entries.
3. The selected TRIM entry specifies the constrained values of the listed extra point degrees-of-freedom ("trim variables") for a particular loading condition. These variables are listed on AESTAT, AESURF and/or AEPARM entries.
4. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.
5. AEQR=0.0 can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading). AEQR=1.0 provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

**TRIM2** Trim Variable Definition

Defines the state of the aerodynamic extra points for a trim analysis. All undefined extra points will be set to zero.

Format:

| 1     | 2      | 3      | 4      | 5      | 6      | 7 | 8 | 9    | 10 |
|-------|--------|--------|--------|--------|--------|---|---|------|----|
| TRIM2 | SID    | MACH   | Q      |        |        |   |   | AEQR |    |
|       | LABEL1 | VALUE1 | LABEL2 | VALUE2 | -etc.- |   |   |      |    |

Example:

|       |       |     |        |      |      |     |  |  |  |
|-------|-------|-----|--------|------|------|-----|--|--|--|
| TRIM2 | 1     | 0.9 | 100.   |      |      |     |  |  |  |
|       | URDD3 | 1.0 | ANGLEA | FREE | ELEV | 0.2 |  |  |  |

| Describer | Meaning                                                                                                                                                                    |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID       | Trim set identification number (Integer>0). See Remarks 1. and 2.                                                                                                          |
| MACH      | Mach number. (Real $\geq 0.0$ and $\neq 1.0$ ). See Remark 5.                                                                                                              |
| Q         | Dynamic pressure. (Real > 0.0)                                                                                                                                             |
| AEQR      | Flag to request a rigid trim analysis (Real $\geq 0.0$ and $\leq 1.0$ ; Default = 1.0). A value of 0.0 provides a rigid trim analysis. See Remark 6.                       |
| LABELi    | The label identifying aerodynamic trim variables defined on an AESTAT, AESURF, or AEPARM entry (Character)                                                                 |
| VALUEi    | The value assigned to LABELi. Either a real number that indicates the variable's fixed value, or one of the following words: FREE, LINKED, or SCHED. See Remarks 3. and 4. |

Remarks:

1. The TRIM2 entry must be selected with the Case Control command TRIM=SID.
2. The SID must be unique among all TRIM and TRIM2 entries.
3. A value of FREE indicates that the controller value will be solved for by the trim process. A value of LINKED indicates that the controller value will be set by an AELINK entry. A value of SCHED indicates that the controller value will be set by a CSSCHD entry. The LINKED and SCHED inputs are optional and provided as a convenience to the user. Nastran will determine the linked and scheduled controller states from the AELINK and CSSCHD entries, respectively.
4. All aerodynamic extra points that have not been defined on a TRIM2, AELINK, or CSSHED entry will be fixed to a value of zero.
5. If MACH is less than 1.0, then the Doublet-Lattice theory is used. If MACH is greater than 1.0, then the ZONA51 theory is used.

6.  $AEQR=0.0$  can be used to perform a rigid trim analysis (ignoring the effects of structural deformation on the loading).  $AEQR=1.0$  provides standard aeroelastic trim analysis. Intermediate values are permissible, but have no physical interpretation (they may be useful for model checkout).

## TRMCPL Trim Component Interface Coupling Parameters

Defines the interface coupling parameters for computing interface coupling matrices.

Format:

| 1      | 2   | 3     | 4     | 5       | 6       | 7       | 8       | 9   | 10 |
|--------|-----|-------|-------|---------|---------|---------|---------|-----|----|
| TRMCPL | TID | CTYPE | PLTOL | GAPTOL1 | GAPTOL2 | GAPTOL3 | GAPTOL4 | RID |    |

Example:

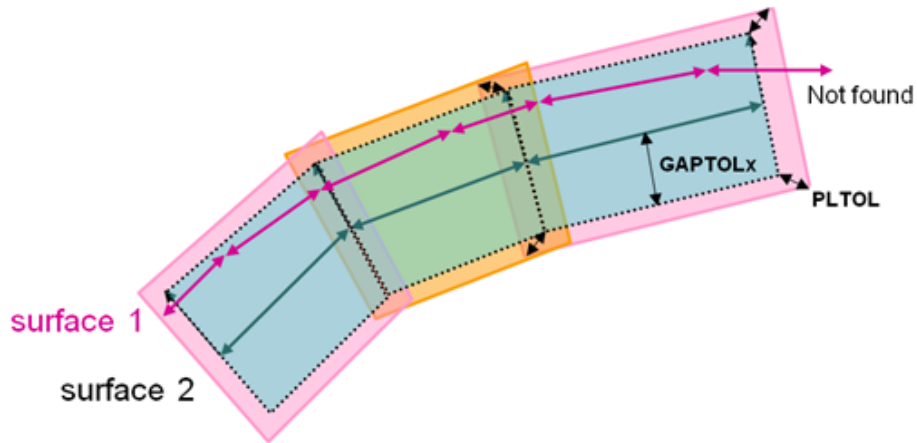
|        |   |        |      |   |  |  |  |  |  |
|--------|---|--------|------|---|--|--|--|--|--|
| TRMCPL | 1 | SGLUED | 0.12 | 5 |  |  |  |  |  |
|--------|---|--------|------|---|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                                                                                                                                                  |
|-----------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| TID       | Identification number of trim component. (Integer>0; Required)                                                                                                                                                                                                                                                                                                                                                                                                                           |
| CTYPE     | Interface coupling type; see Remark 2. (Character, “SGLUED”, “SSLIDE”, “SOPEN”, “SIMPER” or “SAIRGAP”; Required)                                                                                                                                                                                                                                                                                                                                                                         |
| PLTOL     | Relative tolerance for in-plane/in-element/normal acceptance with respect to CTYPE. (Real>0.0; Default=0.1)                                                                                                                                                                                                                                                                                                                                                                              |
| GAPTOLx   | <ul style="list-style-type: none"> <li>■ GAPTOL1 Absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=0.01)</li> <li>■ GAPTOL2 Second absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL1)</li> <li>■ GAPTOL3 Third absolute tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL2)</li> <li>■ GAPTOL4 Fourth tolerance for extrusion with respect to CTYPE. (Real&gt;0.0; Default=GAPTOL3)</li> </ul> |
| RID       | ID of a region of a TRMC. (Blank or Integer≥0, default=0)                                                                                                                                                                                                                                                                                                                                                                                                                                |

Remarks:

- For each interface coupling type, CTYPE, this entry is used to compute the interface matrix with incongruent meshes at the interface.
- For the meanings of values on CTYPE, please see Bulk Data entry, ACPMCP for the detailed descriptions.
- TRMCPL is not a required entry for any trim component referenced by TRIMGRP and should be placed in the main bulk data section or under 'BEGIN BULK'.

4. GAPTOL<sub>x</sub> is the absolute tolerance for mapping surfaces of (structure, trim components) and (trim components, cavity). Grids on surfaces that are separated further than GAPTOL<sub>x</sub> will not be considered for coupling. GAPTOL<sub>x</sub> are utilized for successive projections. GAPTOL1 will be used for first projection. Then, GAPTOL2, GAPTOL3 and GAPTOL4, if provided, will be used for subsequent projections for the remaining unprojected grids. GAPTOL<sub>x</sub> is in the same unit as the one used for length.
5. PLTOL is relative tolerance for computing volume contributions. PLTOL is in percentage.



6. TRMCPL entries of a (TID,RID) pair correspond to ACPEMCP with the same (TID,RID) pair.
7. TRMCPL entry with CTYPE specified must have CTYPE field with non-zero input on the corresponding ACPEMCP entry.

## TSTEP Transient Time Step

Defines time step intervals at which a solution will be generated and output in transient analysis.

Format:

| 1     | 2   | 3      | 4   | 5   | 6 | 7 | 8 | 9 | 10 |
|-------|-----|--------|-----|-----|---|---|---|---|----|
| TSTEP | SID | N1     | DT1 | NO1 |   |   |   |   |    |
|       |     | N2     | DT2 | NO2 |   |   |   |   |    |
|       |     | -etc.- |     |     |   |   |   |   |    |

Example:

|       |   |    |      |   |  |  |  |  |  |
|-------|---|----|------|---|--|--|--|--|--|
| TSTEP | 2 | 10 | .001 | 5 |  |  |  |  |  |
|       |   | 9  | 0.01 | 1 |  |  |  |  |  |

| Describer | Meaning                                                                                        |
|-----------|------------------------------------------------------------------------------------------------|
| SID       | Set identification number. (Integer > 0)                                                       |
| Ni        | Number of time steps of value DTi. (Integer $\geq 1$ )                                         |
| DTi       | Time increment. (Real > 0.0)                                                                   |
| NOi       | Skip factor for output. Every NOi-th step will be saved for output. (Integer > 0; Default = 1) |

Remarks:

1. TSTEP entries must be selected with the Case Control command TSTEP = SID.
2. Note that the entry permits changes in the size of the time step during the course of the solution. Thus, in the example shown, there are 10 time steps of value .001 followed by 9 time steps of value .01. Also, the user has requested that output be recorded for  $t = 0.0, .005, .01, .02, .03$ , etc.
3. See [Guidelines and Tools for Effective Dynamic Analysis](#) in *MSC Nastran Dynamic Analysis User's Guide* or a discussion of considerations leading to the selection of time steps.
4. In modal frequency response analysis (SOLs 111 and 146), this entry is required only when TLOADi is requested; i.e., when Fourier methods are selected.
5. The maximum and minimum displacement at each time step and the SIL numbers of these variables can be printed by altering DIAGON(30) before the transient module TRD1 and by altering DIAGOFF(30) after the module. This is useful for runs that terminate due to overflow or excessive run times.
6. For heat transfer analysis in SOL 159, use the entry.
7. If Modules are present then this entry may only be specified in the main Bulk Data section.

## TSTEPNL Parameters for Nonlinear Transient Analysis

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. is intended for SOLs 129, 159, 600 and SOLs 400 and 700.

Format:

| 1       | 2      | 3      | 4     | 5      | 6      | 7     | 8       | 9       | 10 |
|---------|--------|--------|-------|--------|--------|-------|---------|---------|----|
| TSTEPNL | ID     | NDT    | DT    | NO     | METHOD | KSTEP | MAXITER | CONV    |    |
|         | EPSU   | EPSP   | EPSW  | MAXDIV | MAXQN  | MAXLS | FSTRESS |         |    |
|         | MAXBIS | ADJUST | MSTEP | RB     | MAXR   | UTOL  | RTOLB   | MINITER |    |

Example:

|         |       |       |       |      |       |     |     |    |  |
|---------|-------|-------|-------|------|-------|-----|-----|----|--|
| TSTEPNL | 250   | 100   | .01   | 1    | ADAPT | 2   | 10  | PW |  |
|         | 1.E-2 | 1.E-3 | 1.E-6 | 2    | 10    | 2   | .02 |    |  |
|         | 5     | 5     | 0     | 0.75 | 16.0  | 0.1 | 20. |    |  |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                       |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Identification number. (Integer > 0)                                                                                                                                                                                                                                                                                                                          |
| NDT       | Number of time steps of value DT. See Remark 2. (Integer ≥ 3)                                                                                                                                                                                                                                                                                                 |
| DT        | Time increment. See Remark 2. (Real > 0.0)                                                                                                                                                                                                                                                                                                                    |
| NO        | Time step interval for output. Every NO-th step will be saved for output. See Remark 3. (Integer ≠ 0; Default = 1)                                                                                                                                                                                                                                            |
| METHOD    | Method for stiffness matrix update and the transient time integration schemes. For SOLs 129 and 159, only METHOD=ADAPT is allowed and it is also the default. For SOL 400, all options are allowed. The default is AUTO with non-contact analysis and FNT with contact analysis. See Remark 4. (Character: "AUTO", "ITER", "ADAPT", "SEMI", "FNT", or "PFNT") |
| KSTEP     | KSTEP is the number of converged bisection solutions between stiffness updates for ADAPT method. (Default = 2). It is the number of iterations before stiffness updates for ITER method (Default = 10). See Remarks 18. and 19. (Integer > 0).                                                                                                                |
| MAXITER   | Limit on number of iterations for each time step. See Remarks 5., 6., and 18. (Integer ≠ 0; Default = 10 for ADAPT method and 25 for non-ADAPT methods)                                                                                                                                                                                                       |
| CONV      | Flags to select convergence criteria. See Remarks 7. and 21. (Character; Default = "PW" for SOLs 129 and 159, "UPW" for SOL 400 with non-contact analysis, and "PV" for SOL 400 with contact analysis.)                                                                                                                                                       |
| EPSU      | Error tolerance for displacement (U) criterion. See Remark 20. (Real ≠ 0.0; Default = 1.0E-2 for all methods except PFTN. For PFTN, Default = -1.0E-2)                                                                                                                                                                                                        |
| EPSP      | Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for SOLs 129 and 159, 1.0E-2 for SOL 400)                                                                                                                                                                                                                                               |



| Descriptor | Meaning                                                                                                                                                                                                                |
|------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| EPSW       | Error tolerance for work (W) or energy criterion. See Remark 20. (Real > 0.0; Default = 1.0E-6 for SOLs 129 and 159, 1.0E-2 for SOL 400 and non-PFNT methods, -1.0E-2 for SOL 400 and PFNT method)                     |
| MAXDIV     | Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark 8. (Integer ≠ 0; Default = 2)                                                                        |
| MAXQN      | Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 9. (Integer ≥ 0; Default = 10 for all methods except PFNT. For PFNT, Default = 0)                                            |
| MAXLS      | Maximum number of line searches allowed per iteration. See Remark 9. (Integer ≥ 0; Default = 2 for all methods except PFNT. For PFNT, Default = 0)                                                                     |
| FSTRESS    | Fraction of effective stress ( $\bar{\sigma}$ ) used to limit the subincrement size in the material routines. See Remark 10. (0.0 < Real < 1.0; Default = 0.2)                                                         |
| MAXBIS     | Maximum number of bisections allowed for each time step. See Remark 11. and 12. (-9 ≤ Integer ≤ 9; Default = 5 except for MAXITER < 0 and SOL 400. For MAXITER < 0 and SOL 400, Default = 0)                           |
| ADJUST     | Time step skip factor for automatic time step adjustment. See Remark 13. (Integer ≥ 0; Default = 5)                                                                                                                    |
| MSTEP      | Number of steps to obtain the dominant period response. See Remark 14. (10 ≤ Integer ≤ 200; Default = variable between 20 and 40 for SOL 129 and between 10 and 20 for SOL 400.)                                       |
| RB         | Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 14. (0.1 ≤ Real ≤ 1.0; Default = 0.6)                                                               |
| MAXR       | Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 15. (Real ≥ 1.0; Default = 32.0)                                                                           |
| UTOL       | Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 16. (0.001 < Real ≤ 1.0; Default = 0.1)                                                 |
| RTOLB      | Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 17. (Real > 2.0; Default = 20.0)                                                                            |
| MINITER    | Minimum number of iterations for a load increment, SOL 400 only. (Default = 1 except for the contact analysis. For contact analysis, Default = 2) It is recommended to set MINITER = 2 when high accuracy is expected) |

Remarks:

- For SOL 129 and SOL 159, the Bulk Data entry is selected by the Case Control command = ID. Each residual structure subcase requires a entry and either applied loads via TLOADi data or initial values from a previous subcase. Multiple subcases are assumed to occur sequentially in time with the initial values of time and displacement conditions of each subcase defined by the end conditions of the previous subcase.



For SOL 400, the is selected in the STEP ID's listed within a SUBCASE ID. For SOL 400 each SUBCASE is independent of the previous statement.

2. NDT is used to define the total duration for analysis, which is  $NDT * DT$ . Since DT is adjusted during the analysis, the actual number of time steps, in general, will not be equal to NDT). Also, DT is used only as an initial value for the time increment.
3. For printing and plotting the solution with SOLs 129 and 159, data recovery is performed at time 0,  $NO*DT$ ,  $2*NO*DT$ , ..., and the last converged step, where DT is internally computed time increment and may change at every time step. For SOL 400 and  $NO > 0$ , data recovery is performed at time 0,  $NO*DTI$ ,  $2*NO*DTI$ , ..., and the last converged step, where DTI is the user input initial time increment and it is a constant. For SOL 400 thermal analysis and  $NO < 0$ , the SOL 159 scheme is used. For SOL 400 structural analysis and  $NO < 0$ , the SOL 129 scheme is used for SOL 400, i.e., data recovery is performed at time 0,  $|NO|*DT$ ,  $2*|NO|*DT$ , ..., and the converged step. See Remark 13. on how DT is changed.
4. The stiffness update strategy as well as the transient integration method is selected in the METHOD field.
  - a. METHOD="ADAPT": The program automatically adjusts the incremental time and uses bisection. During the bisection process, the stiffness matrix is updated every KSTEPth converged bisection solution. This is the **only** method available for SOLs 129 and 159 and is also their **default**.
  - b. METHOD="AUTO": The stiffness matrix is automatically updated to improve the convergence. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0. KSTEP value is ignored during iteration. This is the **default** method for SOL 400.
  - c. METHOD = "ITER": The stiffness is updated at every KSTEPth iterations. Also, the program automatically adjusts the incremental time and uses bisection. The automatic time adjustment can be deselected by using ADJUST=0.
  - d. METHOD="SEMI": Same as the AUTO method except that the stiffness updated at the first iteration, and then starts the AUTO iteration scheme.
  - e. METHOD = "FNT": This is the Full Newton iteration method; the stiffness is updated at every iteration. In comparison with the PFNT method, the defaults for FNT are EPSU = 0.01, EPSW = 0.01 and MAXLS = 2. See Remark 19.
  - f. METHOD = "PFNT": This is the Pure Full Newton iteration method. The PFNT method is the same as the FNT method except that the defaults for PFNT method are EPSU = -0.01, EPSW = -0.01, and MAXLS = 0. See Remark 19.
5. This remark applies to SOLs 129 or 159 only. The number of iterations for a time step is limited to MAXITER. If MAXITER is negative, the analysis is terminated when the divergence condition is encountered twice during the same time step or the solution diverges for five consecutive time steps. If MAXITER is positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is treated as a divergent process. See Remark 8.

6. This remark applies to SOL 400 only. The number of iterations for a load increment is limited to  $|\text{MAXITER}|$ . If the solution does not converge in  $|\text{MAXITER}|$  iterations, the load increment is bisected and the analysis is repeated. If the load increment cannot be bisected (i.e.,  $\text{MAXBIS}$  is attained or  $\text{MAXBIS} = 0$ ) and  $\text{MAXDIV}$  is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution still diverges. If  $\text{MAXDIV}$  is negative, the analysis is terminated immediately.

If  $\text{MAXITER}$  is negative, the solution is continued to the end of the current step, even if the solution is divergent. In this case, the best attainable solution is computed for each time step. The default for  $\text{MAXBIS} = 1$ , if  $\text{MAXITER} < 0$ . Also for SOL 400, the value of  $\text{MAXITER}$  for the AUTO method is an approximation. The program will try to obtain a converged solution if it senses the solution can converge.

7. The convergence test flags (U = displacement error test, P = load equilibrium error test, W = work error test, V = vector component method, N = Length method) and the error tolerances (EPSU, EPSP, and EPSW) define the convergence criteria. All requested criteria (combination of U, P, W, V, and/or N) are satisfied upon convergence. Note that at least two iterations are necessary to check the displacement convergence criterion. For SOL 400, if the U criterion is selected together with P or W, then for the first iteration of a load increment, the U criterion will not be checked. For V and N, see Remark 21.
8.  $\text{MAXDIV}$  provides control over diverging solutions. Depending on the rate of divergence, the number of diverging solutions ( $\text{NDIV}$ ) is incremented by 1 or 2. The solution is assumed to diverge when  $\text{NDIV}$  reaches  $\text{MAXDIV}$  during the iteration. If the bisection option is used (allowed  $\text{MAXBIS}$  times) the time step is bisected upon divergence. Otherwise, the solution for the time step is repeated with a new stiffness based on the converged state at the beginning of the time step. If  $\text{NDIV}$  reaches  $\text{MAXDIV}$  again within the same time step, the analysis is terminated for SOLs 129 and 159. For SOL 400, the termination of analysis is dependent on the sign of  $\text{MAXDIV}$ . If  $\text{MAXDIV}$  is positive, the best attainable solution is computed and the analysis is continued to the next time step. Best solutions for 4 time steps are computed. The analysis is terminated if the solution is still diverges. If  $\text{MAXDIV}$  is negative, the analysis is terminated immediately. See Remark 6.
9. Nonzero values of  $\text{MAXQN}$  and  $\text{MAXLS}$  will activate the quasi-Newton update and the line search process, respectively.
10. The number of subincrements in the material routines is determined such that the subincrement size is approximately  $\text{FSTRESS} \cdot \bar{\sigma}$ .  $\text{FSTRESS}$  is also used to establish a tolerance for error correction in elastoplastic material, i.e., error in yield function  $< \text{FSTRESS} \cdot \text{yield stress}$ .  
If the limit is exceeded at the converging state, the program will terminate with a fatal error message. Otherwise, the stress state is adjusted to the current yield surface, resulting in  $\delta = 0$ .
11. The remark applies to SOLs 129 or 159 only. The bisection process is activated when divergence occurs and  $\text{MAXBIS} \neq 0$ . The number of bisections for a time increment is limited to  $|\text{MAXBIS}|$ . If  $\text{MAXBIS}$  is positive and the solution does not converge after  $\text{MAXBIS}$  bisections, the best solution is computed and the analysis is continued to the next time step. If  $\text{MAXBIS}$  is negative and the solution does not converge in  $|\text{MAXBIS}|$  bisection, the analysis is terminated.

12. This remark applies to SOL 400 only. The bisection process is activated when divergence occurs and  $MAXBIS \neq 0$ . The number of bisections for a time step is limited to the absolute value of  $MAXBIS$ . Different actions are taken when the solution diverges depending on the sign of  $MAXBIS$ . If  $MAXBIS$  is positive, the stiffness is updated on the first divergence, and the load is bisected on the second divergence. If  $MAXBIS$  is negative, the load is bisected every time the solution diverges until the limit on bisection is reached. If the solution does not converge after  $|MAXBIS|$  bisections, the analysis is continued or terminated depending on the sign of  $MAXDIV$ . See Remark 8.
13. **ADJUST** controls the automatic time stepping. Since the automatic time step adjustment is based on the mode of response and not on the loading pattern, it may be necessary to limit the adjustable step size when the period of the forcing function is much shorter than the period of dominant response frequency of the structure. It is the user's responsibility to ensure that the loading history is properly traced with the **ADJUST** option. The **ADJUST** option should be suppressed for the duration of short pulse loading. In particular, for impact problems in SOL 400, it is recommended that the **ADJUST** option be suppressed since the short duration pulses during the impact may not be tracked well by the frequency-based algorithm. It should also be noted that the **TZEROMAX** process (where the step is repeated with the same/smaller time step after the first 2 increments) is not currently available for contact problems. If unsure, start with a value for **DT** that is much smaller than the pulse duration in order to properly represent the loading pattern.
  - If **ADJUST** = 0, then the automatic adjustment is deactivated. This is recommended when the loading consists of short duration pulses.
  - If **ADJUST** > 0, the time increment is continually adjusted for the first few steps until a good value of  $\Delta t$  is obtained. After this initial adjustment, the time increment is adjusted every **ADJUST**-th time step only.
  - For SOL 400, if **ADJUST** > 0 and **NO** > 0 (see Remark 3.), the analysis time step can reduce without limit, but it cannot increase more than  $DT*NO$ . This means that, if **NO** = 1, the analysis time step can only reduce, but cannot increase. The user should use **NO** > 1 to take the advantage of the range of automatic time stepping. If **NO** < 0, the time step can both increase and decrease without limit. But in this case, the user will not know the exact output locations.
  - If **ADJUST** is one order greater than **NDT**, then automatic adjustment is deactivated after the initial adjustment.
14. **MSTEP** and **RB** are used to adjust the time increment during analysis. The recommended value of **MSTEP** is 10 to 40. A larger value (e.g., 40) is required for highly nonlinear problems. By default, the program automatically computes the value of **MSTEP** based on the changes in the stiffness.

The time increment adjustment is based on the number of time steps desired to capture the dominant frequency response accurately. The time increment is adjusted as follows:

$$\Delta t_{n+1} = f(r)\Delta t_n$$

where

$$r = \frac{1}{MSTEP} \left( \frac{2\pi}{\omega_n} \right) \left( \frac{1}{\Delta t_n} \right)$$

where:

|     |      |     |                            |
|-----|------|-----|----------------------------|
| f = | 0.25 | for | $r < 0.5 \cdot RB$         |
| f = | 0.5  | for | $0.5 \cdot RB \leq r < RB$ |
| f = | 1.0  | for | $RB \leq r < 2.0$          |
| f = | 2.0  | for | $2.0 \leq r < 3.0/RB$      |
| f = | 4.0  | for | $r \geq 3.0/RB$            |

15. MAXR is used to define the upper and lower bounds for adjusted time step size, i.e.,

$$\text{MIN}\left(\frac{DT}{2^{\text{MAXBIS}}}, \frac{DT}{\text{MAXR}}\right) \leq \Delta t \leq \text{MIN}(\text{MAXR} \cdot DT, \text{NO} \cdot DT)$$

16. UTOL is a tolerance used to filter undesirable time step adjustments; i.e.,

$$\frac{\|\dot{U}_n\|}{\|\dot{U}\|_{\text{max}}} < \text{UTOL}$$

Under this condition no time step adjustment is performed in a structural analysis (SOL 129). In a heat transfer analysis (SOL 159) the time step is doubled.

17. The bisection is activated if the incremental rotation for any degree-of-freedom ( $\Delta\theta_x, \Delta\theta_y, \Delta\theta_z$ ) exceeds the value specified by RTOLB. This bisection strategy is based on the incremental rotation and controlled by MAXBIS.
18. For non-ADAPT (except FNT and PFNT) methods, the stiffness will be updated at convergence if the number of iterations since last iteration is equal or greater than KSTEP. In addition, for SOL 400 and ADAPT method, the stiffness will be updated if  $3 \cdot \text{MAXITER}$  iterations are performed. For SOL 400 and non-ADAPT methods, the stiffness will be updated if MAXITER iterations are performed.
19. For FNT and PFNT methods, whether the stiffness matrix will be updated between the convergence of a load increment and the start of the next load increment depends on the value of KSTEP. In this case, KSTEP = -1, 'BLANK', or 1. A user fatal error will be issued if other value is input. If KSTEP = 1, the stiffness matrix will not be updated. If KSTEP = 'BLANK', the program will decide whether to update depending element type. If KSTEP = -1, the stiffness matrix will be forced to be updated.
20. If EPSU > 0.0, the displacement error is computed file with respect to the total displacements. If EPSU < 0.0, the displacement error is computed with respect to the delta displacements of a load increment. If EPSW > 0.0, the energy error is computed with respect to the total energy. If EPSW < 0.0, the energy error is computed respect to the delta energy of a load increment. The options EPSU < 0.0 and EPSW < 0.0 are available for SOL 400 only.
21. V and N are additional methods for convergence checking using the displacement (U) and/or load (P) criteria. V stands for vector component checking. In this method, convergence checking is performed on the maximum vector component of all components in the model. N stands for length checking. In this method, the length of a vector at a grid point is first computed by the SRSS (square root of the sum of the squares) method. Then convergence checking is performed on the maximum length of all grid points in the model. For example, if CONV=UV, then V checking method will be

performed with the U criteria, i.e., the maximum displacement component of all displacement components in the model is used for convergence checking. For V and N, the EPSU is always negative, i.e., the error is computed with respect to the delta displacements of a load increment, even if positive value is requested by users. CONV=V is the same as CONV=UPV and CONV=D is the same as CONV=UPD. If both V and N are specified; V will take precedence over N. For example, CONV=VN is the same as CONV=V.

By default, for UPV or UPN, separate checks are made over force and moment vectors, and translation and rotation vectors. While the force/translation check is valid always, the moment or rotation check is only valid for 6 DOF elements (beams, shells, etc.). In certain cases (i.e., simply supported or hinged structures where moments are numerically small, small rotation problems), it may be beneficial to turn off the additional convergence testing done for moments and/or rotations.

22. For SOL 700, only fields ID, NDT, and DT are used.
23. In NLTRAN analysis, the output is also influenced by NLPACK. Please refer to NLPACK for an example.

## TTEMP

## Temperature Distribution of Transient Response

Defines a time-dependent temperature distribution for the nonlinear transient analysis in SOL 400.

$$\{T(t)\} = \{A(T(x)) \cdot F(t)\}$$

where  $A(T(x))$  defines a spatial temperature distribution and  $F(t)$  a time function.  $T(t)$  is the time dependent temperature distribution for use in the nonlinear elements in nonlinear transient analysis.

Format:

| 1     | 2   | 3        | 4   | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|----------|-----|---|---|---|---|---|----|
| TTEMP | SID | GROUP_ID | TID |   |   |   |   |   |    |

Example:

|       |    |     |    |  |  |  |  |  |  |
|-------|----|-----|----|--|--|--|--|--|--|
| TTEMP | 11 | 101 | 31 |  |  |  |  |  |  |
|-------|----|-----|----|--|--|--|--|--|--|

| Descriptor | Meaning                                                                  |
|------------|--------------------------------------------------------------------------|
| SID        | Temperature set identification number. (Integer > 0)                     |
| GROUP_ID   | Temperature group identification number (Integer > 0 or =-1)             |
| TID        | Identification number of TABLEDi entry that gives $F(t)$ . (Integer > 0) |

Remarks:

1. The temperature distribution TTEMP must be selected by a Case Control command TEMP(Load)=SID in order to be used in the nonlinear transient analysis.
2. This command is used in SOL 400 only when ANALYSIS=NLTRAN (nonlinear transient analysis) and the temperature load is applied. It only applies to the nonlinear elements in the Residual (SEID=0). There should be only one temperature set for each STEP.
3. GROUP\_ID determines the distribution of temperatures. It references the TMPSET Bulk Data entry to define all grid points, which reference the same TABLEDi entry. Each grid point can have its own GROUP\_ID if necessary. GROUP\_ID=-1 means all grid points are in one group and reference the same TTEMP Bulk Data entry.
4. If the TEMP(INIT) Case Control command references a TTEMP entry, then only the spatial distribution of the TTEMP will be used as the initial temperature distribution for the TEMP(INIT) command.
5. TTEMP may be used in nonlinear transient analysis in SOL 400. For pure linear analysis in SOL 400, DLOAD/TLOADi has to be used to define a time-dependent temperature distribution.

## UDNAME

### User Defined File Name

Provides the name of a file that can be referenced from other bulk data entries such as FTGLOAD, TABLRPC, and MATDIGI.

Format:

| 1      | 2    | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|------|---|---|---|---|---|---|---|----|
| UDNAME | UDID |   |   |   |   |   |   |   |    |
|        | NAME |   |   |   |   |   |   |   |    |

Example:

|        |            |  |  |  |  |  |  |  |  |
|--------|------------|--|--|--|--|--|--|--|--|
| UDNAME | 10         |  |  |  |  |  |  |  |  |
|        | myfile.dat |  |  |  |  |  |  |  |  |

| Describer | Meaning                                                                                                                                    |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------|
| UDID      | Unique UDID (Integer>0). See Remark 1.                                                                                                     |
| NAME      | Name of a file (with or without path) such as the external loading time history in DAC or RPC format or DIGIMAT material file (Character). |

Remarks:

1. The UDID is referenced by FTGLOAD, TABLRPC, or MATDIGI entries
2. The NAME is limited to a maximum of 256 characters ( corresponding to 4 lines of data in fields 2-9 ). For Fatigue analysis using .rsp (RPC) files, there is a 128 character limit (corresponding to 2 lines of data in fields 2-9). Embedded blanks are not supported.
3. If only a NAME with no path (e.g., *sine01.dac*) is supplied, the file is assumed to be located in the same directory as the Nastran input file. If an absolute or relative path is supplied (e.g, *//local/user/fatigue/sine01.dac*), it will be used.

**UDSESV** Define the number and names of user state variables for material user subroutines (SOL 400 only)

Define the number and names of the user state variables. Each state variable has default nominal name if a user name is not given. This is a global entry.

Format:

|        |     |          |      |          |     |          |     |          |    |
|--------|-----|----------|------|----------|-----|----------|-----|----------|----|
| 1      | 2   | 3        | 4    | 5        | 6   | 7        | 8   | 9        | 10 |
| UDSESV |     | NSTATS   |      |          |     |          |     |          |    |
|        | SV2 | SV2_NAME | SV3  | SV3_NAME | SV4 | SV4_NAME | SV5 | SV5_NAME |    |
|        | SV6 | SV6_NAME | .ETC |          |     |          |     |          |    |

Example:

|        |     |      |     |      |  |  |  |  |  |
|--------|-----|------|-----|------|--|--|--|--|--|
| UDSESV |     | 3    |     |      |  |  |  |  |  |
|        | SV2 | VAR2 | SV3 | VAR3 |  |  |  |  |  |

| Describer | Meaning                                                                                                               |
|-----------|-----------------------------------------------------------------------------------------------------------------------|
| NSTATS    | The number of user defined state variables. (Integer >= 1)                                                            |
| SVi       | The default nominal name of state variable (CHARACTER, i >= 1, where i is the index number of the state variable)     |
| SVi_NAME  | The state variable name defined by user (CHARACTER, Default = SVi, where i is the index number of the state variable) |

Remarks:

1. This is a global entry that defines user state variables for material user subroutines. The temperature will always be passed to material use subroutine as the first state variable; its name should not be redefined in this entry.
2. If a state variable is not given a name, SVi will be used as its name. The number i is the index number of the state variable.
3. For output, either state variables names given in UDSESV or default SVi names can be used in NLOUT entry in case control. The state variables names will be used as keywords for output selection.
4. The 1st state variable is always temperature. The remaining user defined state variables are defined and used only by user, Nastran will not use them.
5. This entry only supports UMAT and UCOHES user subroutines, see UNAME in MATUDS
6. This entry is required when MATDIGI is defined to define the number of state variables.
7. The number of state variables is limited to 100 state variables (including temperature) for SOL400.



**UNBALNC****Specifies an Unbalanced Load for Rotordynamic Transient or Frequency Response Analysis**

Used in rotordynamic analyses to specify a rotating unbalance load in terms of a cylindrical system with the rotor rotation as the z-axis.

**Format:**

| 1       | 2       | 3     | 4       | 5                 | 6                  | 7     | 8 | 9 | 10 |
|---------|---------|-------|---------|-------------------|--------------------|-------|---|---|----|
| UNBALNC | SID     | MASS  | GRID    | X1                | X2                 | X3    |   |   |    |
|         | ROFFSET | THETA | ZOFFSET | T/F <sub>on</sub> | T/F <sub>OFF</sub> | CFLAG |   |   |    |
|         | UFT1    | UFT2  | UFT3    | UFR1              | UFR2               | UFR3  |   |   |    |
|         | MCT1    | MCT2  | MCT3    | MCR1              | MCR2               | MCR3  |   |   |    |
|         | SCR1    | SCR2  | SCR3    |                   |                    |       |   |   |    |

**Example:**

|         |      |      |      |      |      |      |  |  |  |
|---------|------|------|------|------|------|------|--|--|--|
| UNBALNC | 100  | 0.1  | 1001 | 0.0  | 1.0  | 0.0  |  |  |  |
|         | 0.02 | 30.0 | 0.5  |      |      | MASS |  |  |  |
|         | 1001 | 1002 | 1003 | 1004 | 1005 | 1006 |  |  |  |
|         | 2001 | 2002 | 2003 | 2004 | 2005 | 2006 |  |  |  |
|         | 3001 | 3003 | 3004 |      |      |      |  |  |  |

| Describer         | Meaning                                                                                                                                                                                                  |
|-------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SID               | Set identification number. (Integer; Required; no Default)                                                                                                                                               |
| MASS              | Mass of imbalance. See Remark 4. (Real or Integer; if integer must be > 0; Required; no Default)                                                                                                         |
| GRID              | Grid identification number of applying imbalance. See Remark 10. (Integer; Required; no Default)                                                                                                         |
| X1, X2, X3        | Components of the vector, from GRID, in the displacement coordinate system of GRID, which is used to define a cylindrical coordinate system centered at GRID; see Remark 5. (Real; Required; no Default) |
| ROFFSET           | Offset of mass in the radial direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 1.0)                                                       |
| THETA             | Angular position, in degrees, of the mass in the unbalance coordinate system. (Real; Default = 0.0)                                                                                                      |
| ZOFFSET           | Offset of mass in the z-direction of the unbalance coordinate system. See Remark 4. (Real or Integer; if integer, must be > 0; Default = 0.0)                                                            |
| T/F <sub>ON</sub> | Start time or frequency for applying imbalance load. See Remark 6. (Real ≥ 0.0; Default = 0.0)                                                                                                           |

| Describer          | Meaning                                                                                                                                                                                                                                                                                                          |
|--------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| T/F <sub>OFF</sub> | Stop time or frequency for terminating imbalance load. See Remark 6. (Real > 0.0; Default = 999999.0)                                                                                                                                                                                                            |
| UFT1-3             | EPOINTS to output the unbalanced forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)                                                                                                                                                                                                            |
| UFR1-3             | EPOINTS to output the unbalanced forces in the R1, R2, and R3 directions. See Remark 6. (Integer > 0)                                                                                                                                                                                                            |
| MCT1-3             | EPOINTS to output the mass-correction forces in the T1, T2, and T3 directions. See Remark 6. (Integer > 0)                                                                                                                                                                                                       |
| MCR1-3             | EPOINTS to output the mass-correction forces in the R1, T2, and R3 directions. See Remark 6. (Integer > 0)                                                                                                                                                                                                       |
| SCR1-3             | EPOINTS to output the speed-correction forces in the R1, R2, and R3 directions. (Integer > 0)                                                                                                                                                                                                                    |
| CFLAG              | Correct flag to specify whether 1) the mass will be used to modify the total mass in the transient response calculations, 2) the effect of the rotor spin rate change will be included in the transient response calculation, or 3) both; see Remark 7. (Character: NONE, MASS, SPEED, or BOTH, Default = NONE). |

Remarks:

- Multiple UNBALNC entries with the same SID value are allowed.
- UNBALNC can be specified on reference and/or non-reference rotors. For transient response, UNBALNC is specified by the RGYRO Case Control command. For frequency response, it is specified by the DLOAD Case Control command; the RGYRO Case Control command, however, is still required to activate the unbalance information and the rotodynamics capability. For frequency response SYNC analysis, the rotor speed ratio between all rotors should be 1.0. Speed ratio is calculated based on the ratio of SPTID entry of its RSPINR entry to SPDIT field of its reference rotor RSPINR entry. A User Fatal Message will be issued if the ratio is not equal to 1.0.
- The imbalance load will be generated based on the mass value, offset values, and the rotor spin speed.
- If the entry is a real number, the value is considered constant. If the entry is an integer number, the value references a TABLEDi entry that specifies the value as a function of time for transient response or frequency for frequency response.
- A cylindrical coordinate system is used to determine the initial position and rotation direction of the mass unbalance. Theta is measured from the plane defined by the z-axis and the user specified vector (X1, X2, X3). Theta = 0.0 is in the direction of the user-specified vector (X1, X2, X3). Unbalance rotation is in the positive z-direction.
- For frequency response analysis, the EPOINTS and the continuation cards are ignored.
- If the mass loss (increase) is relatively small, the correction may safely be ignored. (CFLAG = NONE). These error corrections terms are applied for analysis in fixed reference frame only.

8. The mass specified on UNBALNC is scaled with PARAM, WTMASS factor for both transient and frequency response analysis. For the rotating reference frame, the unbalance load should be defined through UNBALNC only.

Equations in fixed reference frame:

- a. For Frequency response analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} mr\omega^2(\cos\theta_k + i\sin\theta_k) \\ mr\omega^2(\sin\theta_k + i\cos\theta_k) \end{Bmatrix}$$

- b. For transient analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \cdot \begin{Bmatrix} m(t)r(t)(\Omega_j^2(t)\cos\theta_{jk}(t) + \dot{\Omega}_j(t)\sin\theta_{jk}(t)) \\ m(t)r(t)(\Omega_j^2(t)\sin\theta_{jk}(t) - \dot{\Omega}_j(t)\cos\theta_{jk}(t)) \end{Bmatrix}$$

where,

$m(t)$  Specified by the user (MASS field in UNBALNC)

$r(t)$  Specified by the user (ROFFSET field in UNBALNC)

$\Omega_j(t)$  Specified by the user through RSINT/RSPINR entry

$\dot{\Omega}_j(t)$  Derived from  $\Omega_j(t)$

$\theta_{jk}(t)$  Instantaneous angular location, measured from axis 1 to axis 2 ( $= \int (\Omega_j(t)dt + \theta_k)$ )

$\theta_k$  Specified by the user (THETA field in UNBALNC) 0

j Rotor on which the grid is located

Equations in rotating reference frame:

- c. For frequency response analysis

$$\begin{Bmatrix} F_x(\omega) \\ F_y(\omega) \end{Bmatrix} = WTMASS \bullet \begin{Bmatrix} mr\omega^2(\cos\theta_k) \\ mr\omega^2(\sin\theta_k) \end{Bmatrix}$$

- d. For transient analysis

$$\begin{Bmatrix} F_x(t) \\ F_y(t) \end{Bmatrix} = WTMASS \bullet \begin{Bmatrix} m(t)r(t)\Omega_{jk}^2(t) \cos \theta_k \\ m(t)r(t)\Omega_{jk}^2(t) \sin \theta_k \end{Bmatrix}$$

9. The moment due to ZOFFSET is given by,

$$\begin{Bmatrix} M_x(\omega) \\ M_y(\omega) \end{Bmatrix} = \begin{Bmatrix} -F_y(\omega) \bullet ZOFFSET \\ F_x(\omega) \bullet ZOFFSET \end{Bmatrix}$$

10. The GRID specified on UNBALNC and the GRIDA specified on RSPINR/RSPINT for the same rotor must use the same CD.

**UNGLUE****Defines Grids that Should Be Eliminated From Glued Contact**

This entry may be used in SOLs 101 or 400 but is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact.

Format:

| 1      | 2   | 3    | 4   | 5    | 6   | 7      | 8 | 9 | 10 |
|--------|-----|------|-----|------|-----|--------|---|---|----|
| UNGLUE | ID  | BID  | ID1 | THRU | ID2 | BY     | N |   |    |
|        | ID3 | THRU | ID4 | ID5  | ID6 | -etc.- |   |   |    |

Example:

|        |    |      |      |      |      |      |      |  |  |
|--------|----|------|------|------|------|------|------|--|--|
| UNGLUE | 10 | 101  | 20   | THRU | 300  | BY   | 2    |  |  |
|        |    | 3457 | 8456 | 4712 | 1000 | THRU | 2000 |  |  |

| Describer | Meaning                                                                                                                            |
|-----------|------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Identification number referenced by a SUBCASE or STEP Case Control BCONTACT or UNGLUE command. See Remark 1. (Integer; no Default) |
| BID       | Identification of the specified BCBODY. (Integer > 0; no Default).                                                                 |
| Idi       | ID list of grids. (Integer > 0; no Default).                                                                                       |
| THRU      | Enter THRU if a range of grids is required. (Character)                                                                            |
| BY        | Enter BY if the range of grids is not consecutive. (Character)                                                                     |
| N         | BY increment. (Integer > 0).                                                                                                       |

Remarks:

1. To place an entry in loadcase 0, set ID=0, which does not need any corresponding Case Control command BCONTACT=0 or UNGLUE=0 and is always automatically executed. To place an entry in any physical loadcase (SUBCASE or STEP), ID must be selected by the Case Control command BCONTACT=ID or UNGLUE=ID. Note that UNGLUE Case Control will take precedence over the BCONTACT Case Control for this entry with same ID. UNGLUE is ignored by Permanent contact.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.

**UNGLUE** Defines Grids that Should Be Eliminated From Glued Contact in SOL 600

This entry is only necessary if glued contact has been specified and some of the grids should use standard contact instead of glued contact. This option is normally used for crack analysis where the grids along the crack are not glued but all other grids on a contact body have glued contact. SOL 600 only.

Format:

|          |          |          |          |          |          |          |          |          |           |
|----------|----------|----------|----------|----------|----------|----------|----------|----------|-----------|
| <b>1</b> | <b>2</b> | <b>3</b> | <b>4</b> | <b>5</b> | <b>6</b> | <b>7</b> | <b>8</b> | <b>9</b> | <b>10</b> |
| UNGLUE   | IDBC     | IBOD     | D1       | THRU     | D2       | BY       | N3       |          |           |
|          |          |          | D4       | THRU     | D5       | BY       | N6       |          |           |

Example:

|        |   |   |   |      |     |  |  |  |  |
|--------|---|---|---|------|-----|--|--|--|--|
| UNGLUE | 1 | 1 | 1 | THRU | 100 |  |  |  |  |
|--------|---|---|---|------|-----|--|--|--|--|

| Descriptor | Meaning                                                                                                                                                                                                                                                                                    |
|------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| IDBC       | Identification number of a Case Control BCONTACT command defining the subcase to which these items should be available.<br>Enter 0 if this entry should apply to Marc's increment zero. Enter -9999 if it should apply to all subcases. (Integer; no Default)                              |
| IBOD       | Identification number of a matching BCBODY Bulk Data entry of a flexible surface defining the body with GRIDS to be removed from glued contact. The BCBODY may include additional grids (not included in this entry) that remain glued. (Integer; Default = 1) See Remarks 4. and 5. (3,1) |
| IDi        | ID list of grids. (Integer > 0; no Default).                                                                                                                                                                                                                                               |
| THRU       | Enter THRU if a range of grids is required. (Character)                                                                                                                                                                                                                                    |
| BY         | Enter BY if the range of grids is not consecutive. (Character)                                                                                                                                                                                                                             |
| N          | BY increment. (Integer > 0).                                                                                                                                                                                                                                                               |

Remarks:

1. Repeat the continuation entry until all grids for the particular body are described.
2. This entry may be repeated as many times as necessary to define all subcases and bodies with grids that should not have glued contact.
3. This entry corresponds to Marc's DEACT GLUE option. Items (i,j) indicate the corresponding data block and field.
4. In certain models, there are no BCBODY entries (for example; self contact) because the entire model comprises one body. For such models IBOD may be left blank.

5. If IBOD is a positive number, it will be converted to the body number using the BCBODY entries. To override this conversion and use the body number directly, enter IBOD as a negative number whose magnitude is equal to the body number desired.
6. Multiple UNGLUE entries with the same IDBC are not allowed.

## USET Degree-of-Freedom Set Definition

Defines a degree-of-freedom set.

Format:

| 1    | 2     | 3   | 4  | 5   | 6  | 7   | 8  | 9 | 10 |
|------|-------|-----|----|-----|----|-----|----|---|----|
| USET | SNAME | ID1 | C1 | ID2 | C2 | ID3 | C3 |   |    |

Example:

|      |    |     |    |    |   |  |  |  |  |
|------|----|-----|----|----|---|--|--|--|--|
| USET | U4 | 333 | 26 | 17 | 0 |  |  |  |  |
|------|----|-----|----|----|---|--|--|--|--|

| Descriptor | Meaning                                                                                                                                                      |
|------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SNAME      | Set name. (One to four characters, or the string "ZERO" followed by the set name.)                                                                           |
| IDI        | Grid or scalar point identification number. (Integer > 0)                                                                                                    |
| CI         | Component number. (Integer zero or blank for scalar points, or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME = "ZEROi", where i is a set name, then the degrees-of-freedom are omitted from set i.
3. A maximum of 18 degrees-of-freedom may be designated on a single entry.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.
6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message "SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE". This entry should only reference new sets defined on DEFUSET Bulk Data entries.



## USET1 Degree-of-Freedom Set Definition, Alternate Form

Defines a degrees-of-freedom set.

Format:

| 1     | 2     | 3   | 4      | 5   | 6   | 7   | 8   | 9   | 10 |
|-------|-------|-----|--------|-----|-----|-----|-----|-----|----|
| USET1 | SNAME | C   | ID1    | ID2 | ID3 | ID4 | ID5 | ID6 |    |
|       | ID7   | ID8 | -etc.- |     |     |     |     |     |    |

Example:

|       |    |     |   |   |    |   |   |   |  |
|-------|----|-----|---|---|----|---|---|---|--|
| USET1 | SB | 345 | 2 | 1 | 36 | 5 | 9 | 7 |  |
|       | 40 |     |   |   |    |   |   |   |  |

Alternate Format and Example:

|       |       |     |     |        |     |  |  |  |  |
|-------|-------|-----|-----|--------|-----|--|--|--|--|
| USET1 | SNAME | C   | ID1 | “THRU” | ID2 |  |  |  |  |
| USET1 | SB    | 123 | 170 | THRU   | 180 |  |  |  |  |

| Describer | Meaning                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------|
| SNAME     | Set name. (One to four characters or the word “ZERO” followed by the set name.)                                                                              |
| C         | Component numbers. (Integer zero or blank for scalar points or any unique combinations of the Integers 1 through 6 for grid points with no embedded blanks.) |
| IDI       | Grid or scalar point identification number. (Integer > 0; for “THRU” option, ID1 < ID2.)                                                                     |

Remarks:

1. SNAME may refer to any of the set names given in [Degree-of-Freedom Sets](#) or their new names on the DEFUSET entry. However, it is recommended that SNAME refer only to the set names U1 through U6 or their new names on the DEFUSET entry. If set names a through v are used then the degrees-of-freedom may also have to be defined in the applicable super sets on other USETi entries.
2. If SNAME=“ZEROi”, where i is a set name, then the degrees-of-freedom are omitted from set i.
3. If the alternate format is used, all of the points ID1 through ID2 are assigned to the set.
4. If degrees-of-freedom defined by USET entries are found to be singular and AUTOSPC is requested for a degree-of-freedom that is also in a set that AUTOSPC may change, then the set defined by the USET entry will be removed by the AUTOSPC operation. An avoidance is to use PARAM,AUTOSPC,NO.
5. The USET1 entry is processed by the GP4 module with its effect appearing in the USET table. User-written DMAPs must therefore include the GP1 and GP4 modules if USET entries are used.

6. If a USETi Bulk Data entry lists a standard degree-of-freedom set, such as S or M, the program may fail in the PARTN module with the message “SYSTEM FATAL MESSAGE 3007, ILLEGAL INPUT TO SUBROUTINE”. This entry should only reference new sets defined on DEFUSET Bulk Data entries.

## USRSUB6

## Defines User Subroutines for SOL 600

Defines user subroutines used in SOL 600 only.

Format:

| 1       | 2  | 3   | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|----|-----|----|----|----|----|----|----|----|
| USRSUB6 | U1 | U2  | U3 | U4 | U5 | U6 | U7 | U8 |    |
|         | U9 | U10 |    |    |    |    |    |    |    |

Examples:

|          |               |       |        |  |  |  |  |  |  |
|----------|---------------|-------|--------|--|--|--|--|--|--|
| USRSUB6  | UDAMAG        | uvoid | TENSOF |  |  |  |  |  |  |
| USRSUB6* | SEPFORBB<br>C |       |        |  |  |  |  |  |  |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                                              |
|-----------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Ui        | Name of user subroutine(s) to be included. See Marc Volume D for list of available user subroutines. Do not include the .f extension on this entry, however, the actual file on the disk must have the .f extension. If any user subroutine exceeds 8 characters, use the wide field format for the primary line and all continuation lines. (Character; no Default) |

Notes:

1. All user subroutines must reside in the directory where the Nastran input file resides.
2. All user subroutines on disk must be in lower case and have an extension of .f. The names entered in the bulk data entry may be in upper or lower case. They will be converted to lower case.
3. SOL 600 combines all user subroutines into one large subroutine named u600.f and u600.f is passed to the Marc command line when spawned from Nastran.
4. If only one user subroutine is required, an alternate is to use PARAM,MARCUSUB,name.

## UXVEC

### Control Parameter State

Specification of a vector of aerodynamic control point (extra point) values. These data define the control positions corresponding to user defined nonlinear control forces that have been defined by AEDW, AEPRESS and AEFORCE entries. Only nonzero values need to be defined.

Format:

| 1     | 2      | 3   | 4      | 5   | 6      | 7 | 8 | 9 | 10 |
|-------|--------|-----|--------|-----|--------|---|---|---|----|
| UXVEC | ID     |     |        |     |        |   |   |   |    |
|       | LABEL1 | UX1 | LABEL2 | UX2 | -etc.- |   |   |   |    |

Example:

|       |        |      |        |      |  |  |  |  |  |
|-------|--------|------|--------|------|--|--|--|--|--|
| UXVEC | 1001   |      |        |      |  |  |  |  |  |
|       | THRUST | 1.E4 | ANGLEA | .015 |  |  |  |  |  |

| Describer | Meaning                                                                                                                    |
|-----------|----------------------------------------------------------------------------------------------------------------------------|
| ID        | Control vector identification number, see Remark 1. (Integer > 0)                                                          |
| LABELi    | Controller name. This must correspond to an existing AESURF, AESTAT or AEPARM label or INTERCPT (Character). See Remark 4. |
| UXi       | The magnitude of the aerodynamic extra point degree-of-freedom (Real)                                                      |

Remarks:

1. The ID is referenced by the AEUXREF Case Control command and/or by AEDW, AEPRESS, and/or AEFORCE entries.
2. The units of the user defined AEPARM controllers are implied by their use on this entry and the corresponding values on the force vector definition. The user must be self-consistent in all uses of the same controller. AESURF controllers are expressed in radians as are the rigid body angles ANGLEA and BETA. The rigid body rates, ROLL, PITCH and YAW are nondimensional rates pb/2V, qc/2V, rb/2V; respectively. V is the velocity and b and c are the reference span and chord lengths, respectively.
3. LABELs that are part of the UX vector that are omitted in the UXVEC specification are assigned a value of 0.0.
4. INTERCPT = 1.0 is implied on all UXVEC input.  
INTERCPT = 0.0 indicates that associated load is a perturbation.  
If there are no AESTAT entires, INTERCPT=1.0 must be input on one or more UXVEC entries to establish the intercept aerodynamics.
5. When multiple control point values are listed for a single controller, it is necessary to add an additional AEPARM controller with value fixed to 1.0 to maintain accuracy.

6. If LABEL<sub>i</sub> is AESTAT and UX<sub>i</sub> = 1.0 and INTERCPT is 1.0, either explicit or by default and there are no other LABEL<sub>j</sub>, UX<sub>j</sub> pairs, the term is added to the internal forces for the AESTAT. If LABEL<sub>i</sub> is an AESTAT and UX<sub>i</sub> is not 1.0, it is an error.

VCCT

Virtual Crack Closure Technique - SOL 400

Format:

| 1    | 2      | 3     | 4     | 5     | 6    | 7      | 8        | 9         | 10 |
|------|--------|-------|-------|-------|------|--------|----------|-----------|----|
| VCCT | ID     | IDCR  | ITYPE | IGROW | INCM | METHOD |          |           |    |
|      | CGI    | GC    |       |       |      |        | GC-II    | GC-III    |    |
|      | TABCGI | TABGC |       |       |      |        | TABGC-II | TABGC-III |    |
|      | G1     | G2    | G3    | G4    | G5   | etc.   |          |           |    |

Example:

|      |     |       |   |   |   |   |  |   |  |
|------|-----|-------|---|---|---|---|--|---|--|
| VCCT | 101 | 1     | 2 | 2 | 2 | 1 |  | 1 |  |
|      |     | 2000. |   |   |   |   |  |   |  |
|      |     | 0     |   |   |   |   |  |   |  |
|      | 51  | 52    |   |   |   |   |  |   |  |

| Describer | Meaning                                                                                                                                                                                                                                                                                                                                     |
|-----------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID        | Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)                                                                                                                                                                                                                                                   |
| IDCR      | Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR. (Integer > 0; Default = 1)                                                                                                                                                                  |
| ITYPE     | Type of crack propagation. (Integer; Default = 0)<br>0 No crack propagation<br>2 Direct crack propagation                                                                                                                                                                                                                                   |
| IGROW     | Specifies how the crack grows. (Integer; Default = 2)<br>1 Uses remeshing (not presently available)<br>2 Release user tyings or glued contact                                                                                                                                                                                               |
| INCM      | Specifies the crack growth increment (Integer; Default = 1)<br>1 Uses a fixed increment or a user subroutine                                                                                                                                                                                                                                |
| METHOD    | Specifies the method used for the estimated crack growth direction (Integer; Default=1)<br>1 Maximum hoop stress criterion (Default)                                                                                                                                                                                                        |
| CGI       | Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law. (Not presently used)                                                                                                    |
| GC        | Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9. For SOL 600, if it is desired that the mode I crack resistance be zero, enter GC as a negative value. |

| Describer | Meaning                                                                                                                                                            |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| GC-II     | Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.                                                                                 |
| GC-III    | Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.                                                                                |
| TABCGI    | TABLEMi or TABL3Di for CGI (crack growth increment). (Integer; Default = 0)                                                                                        |
| TABGC     | TABLEMi or TABL3Di for scaling GC (crack growth resistance). (Integer; Default = 0)                                                                                |
| TABGC-II  | TABLEMi or TABL3Di for scaling GC-II (Integer; Default = 0)                                                                                                        |
| TABGC-III | TABLEMi or TABL3Di for scaling GC-III (Integer; Default = 0)                                                                                                       |
| Gi        | Grids along the crack front – for a shell there is only one node. See Remark 6. (Integer or “THRU” or “BY”; no Default; at least one value, G1, must be provided.) |

**Remarks:**

1. The grids entered on this entry **MUST** be associated with SOL400 elements that have had their capabilities extended by use of either a PSNLN1, PSNLN2, PSLDN1, PLCOMP or PCOMPLS or a combination of these entries.
2. ID corresponds to a Case Control VCCT entry.
3. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
4. This entry may be repeated as many times as necessary to describe all the cracks in the model.
5. The 4<sup>th</sup> line may be repeated as many times as necessary to describe all grids on the crack front
6. If G1 is negative, the absolute value of G1 is used as the ID of a SET3 entry providing the list of grids. For this case, G2, G3, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
7. The values of the entries on the second line that are not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
8. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

## VCCT

## Virtual Crack Closure Technique in SOL 600

Provides data to simulate crack growth using virtual crack closure technology methods in SOL 600.

Format:

| 1    | 2      | 3     | 4      | 5     | 6    | 7       | 8        | 9         | 10 |
|------|--------|-------|--------|-------|------|---------|----------|-----------|----|
| VCCT | ID     | IDCR  | ITYPE  | IGROW | INCM | METHOD  | TIME     | IACT      |    |
|      | CGI    | GC    | GTH    | C     | M    | GMIN    | GC-II    | GC-III    |    |
|      | TABCGI | TABGC | TABGTH | TABC  | TABM | TABGMIN | TABGC-II | TABGC-III |    |
|      | G1     | G2    | G3     | G4    | G5   | etc.    |          |           |    |

Alternate Format:

|      |        |       |        |       |      |         |          |           |  |
|------|--------|-------|--------|-------|------|---------|----------|-----------|--|
| VCCT | ID     | IDCR  | ITYPE  | IGROW | INCM | METHOD  | TIME     | IACT      |  |
|      | CGI    | GC    | GTH    | C     | M    | GMIN    | GC-II    | GC-III    |  |
|      | TABCGI | TABGC | TABGTH | TABC  | TABM | TABGMIN | TABGC-II | TABGC-III |  |
|      | G1     | THRU  | G2     | BY    | G3   |         |          |           |  |

Example:

|      |     |       |     |    |     |   |  |   |  |
|------|-----|-------|-----|----|-----|---|--|---|--|
| VCCT | 101 | 1     | 2   | 2  | 2   | 1 |  | 1 |  |
|      |     | 2000. | 12. | 4. | 2.0 |   |  |   |  |
|      |     | 0     | 0   | 0  | 0   |   |  |   |  |
|      | 51  | 52    |     |    |     |   |  |   |  |

| Describer      | Meaning                                                                                                                                                                                                            |
|----------------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID             | Identification of a matching Case Control VCCT entry. See Remark 2. (Integer; no Default)                                                                                                                          |
| IDCR           | Identification of this particular crack. IDCR must be unique among all VCCT cracks with the same ID but can replace a crack with the same IDCR and a different ID using the IACT field. (Integer > 0; Default = 1) |
| ITYPE<br>(6-1) | Type of crack propagation. (Integer; Default = 2)<br><br>0 No crack propagation<br>1 Fatigue type crack propagation<br>2 Direct crack propagation                                                                  |
| IGROW<br>(6-2) | How crack grows (Integer; Default = 2)<br><br>1 Uses remeshing (not presently available)                                                                                                                           |



| Describer       | Meaning                                                                                                                                                                                                                               |
|-----------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| INCM<br>(6-3)   | 2 Release user tyings or glued contact                                                                                                                                                                                                |
|                 | Specifies the crack growth increment (Integer; Default = 2)                                                                                                                                                                           |
| METHOD<br>(6-4) | 1 Uses a fixed increment or a user subroutine                                                                                                                                                                                         |
|                 | 2 Uses Paris law                                                                                                                                                                                                                      |
| TIME<br>(6-5)   | Specifies the method used for the estimated crack growth direction (Integer; Default=1)                                                                                                                                               |
|                 | 1 Maximum hoop stress criterion (Default)                                                                                                                                                                                             |
| IACT<br>(3-1)   | Time period for fatigue load sequence. (Real; no Default) Only enter if ITYPE=1                                                                                                                                                       |
|                 | Flag for activating or deactivating an existing crack (Integer; Default = 0)                                                                                                                                                          |
| CGI<br>(7-1)    | 0 Leave as is                                                                                                                                                                                                                         |
|                 | 1 Activate                                                                                                                                                                                                                            |
|                 | 2 Deactivate                                                                                                                                                                                                                          |
| GC<br>(7-2)     | Crack growth increment (Real; Default = 0.0). If the option of releasing tyings or glued contact is used, the length of the released element edge is used. Leave blank for fatigue growth defined by the Paris law.                   |
| GTH<br>(7-3)    | Crack growth resistance. (Real; Default = 0.0) Ignored for fatigue growth. If different crack growth resistance values are needed from modes I, II, III, this is the mode I value and modes II and III are entered in fields 8 and 9. |
| C<br>(7-4)      | Paris law energy release rate threshold. (Real; Default = 0.0)                                                                                                                                                                        |
| M<br>(7-5)      | Paris law parameter C. (Real; Default = 0.0) Only enter if INCM=2                                                                                                                                                                     |
| GMIN<br>(7-6)   | Paris law parameter m. (Real; Default = 0.0) Only enter if INCM=2                                                                                                                                                                     |
| GC-II (7-4)     | Minimum growth increment. (Real; Default = 0.0) Only enter if INCM=2                                                                                                                                                                  |
| GC-III          | Crack growth resistance, Mode II (Real; Default = 0.0) Ignored for fatigue growth.                                                                                                                                                    |
| TABCGI<br>(8-1) | Crack growth resistance, Mode III (Real; Default = 0.0) Ignored for fatigue growth.                                                                                                                                                   |
| TABGC<br>(8-2)  | TABLEMi or TABL3Di for CGI (crack growth increment). (Integer; Default = 0)                                                                                                                                                           |
|                 | TABLEMi or TABL3Di for scaling CG (fracture toughness). (Integer; Default = 0)                                                                                                                                                        |

| Describer        | Meaning                                                                                                                                          |
|------------------|--------------------------------------------------------------------------------------------------------------------------------------------------|
| TABGTH<br>(8-3)  | TABLEMi or TABL3Di for scaling GTH (Paris law energy release rate) (Integer; Default = 0)                                                        |
| TABC<br>(8-4)    | TABLEMi or TABL3Di for scaling C (Paris law parameter C). (Integer; Default = 0)                                                                 |
| TABM<br>(8-5)    | TABLEMi or TABL3Di for scaling M (Paris law parameter m). (Integer; Default = 0)                                                                 |
| TABGMIN<br>(8-6) | TABLEMi or TABL3Di for scaling GMIN (Minimum growth increment). (Integer; Default = 0)                                                           |
| TABGC-II         | TABLEMi or TABL3Di for scaling GC-II (Integer; Default = 0)                                                                                      |
| TABGC-III        | TABLEMi or TABL3Di for scaling GC-III (Integer; Default = 0)                                                                                     |
| Gi<br>(5-i)      | Grids along the crack front – for a shell there is only one node. (Integer; no Default; at least one value, G1, must be provided. See Remark 6.) |

#### Remarks:

1. This entry corresponds to Marc's VCCT model definition and history definition options.
2. (i,j) corresponds to Marc Vol C VCCT entry ith datablock jth field
3. ID corresponds to a Case Control VCCT command. Set ID=0 to enter VCCT entries into Marc's model definition.
4. If tables are not required, enter at least one field with a zero value. Do not enter a blank line.
5. This entry may be repeated as many times as necessary to describe all the cracks in the model.
6. The 4<sup>th</sup> line may be repeated as many times as necessary to describe all grids on the crack front
7. If the Alternate Format is used, entries may only be made in the fields indicated, however this line may be repeated as many times as necessary to describe all grids on the crack front.
8. If G1 is negative, the absolute value of G1 is used as the ID of a SET3 entry providing the list of grids. For this case, G2, G3, etc. must be blank (only one SET3 ID per VCCT entry is allowed).
9. Most analyses do not use all of the entries on the second line, however a meaningful analysis will specify one or more of these values. Those not needed should be set to 0.0 or blank. If no tables are required to specify the variation with time, temperature or some other variable, one or all of the table entries on the third line can be set to zero or blank.
10. If TABLEM1 is used, accumulated crack growth will be used for the X coordinates instead of the usual value of temperature.

## VIEW

## View Factor Definition

Defines radiation cavity and shadowing for radiation view factor calculations.

Format:

| 1    | 2     | 3       | 4     | 5  | 6  | 7      | 8 | 9 | 10 |
|------|-------|---------|-------|----|----|--------|---|---|----|
| VIEW | IVIEW | ICAVITY | SHADE | NB | NG | DISLIN |   |   |    |

Example:

|      |   |   |      |   |   |      |  |  |  |
|------|---|---|------|---|---|------|--|--|--|
| VIEW | 1 | 1 | BOTH | 2 | 3 | 0.25 |  |  |  |
|------|---|---|------|---|---|------|--|--|--|

| Describer | Meaning                                                                                                                                                                                                                                                                                                                              |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| IVIEW     | Identification number. (Integer > 0)                                                                                                                                                                                                                                                                                                 |
| ICAVITY   | Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)                                                                                                                                                                                                                               |
| SHADE     | Shadowing flag for the face of CHBDYi element. (Character; Default = "BOTH")<br>NONE means the face can neither shade nor be shaded by other faces.<br>KSHD means the face can shade other faces.<br>KBSD means the face can be shaded by other faces.<br>BOTH means the face can both shade and be shaded by other faces. (Default) |
| NB        | Subelement mesh size in the beta direction. (Integer > 0; Default = 1)                                                                                                                                                                                                                                                               |
| NG        | Subelement mesh size in the gamma direction. (Integer > 0; Default = 1)                                                                                                                                                                                                                                                              |
| DISLIN    | The displacement of a surface perpendicular to the surface. See <a href="#">Figure 9-155</a> . (Real; Default = 0.0)                                                                                                                                                                                                                 |

Remarks:

1. VIEW must be referenced by CHBDYE, CHBDYG, or CHBDYP elements to be used.
2. ICAVITY references the cavity to which the face of the CHBDYi element belongs; a zero or blank value indicates this face does not participate in a cavity.
3. NB, NG, and DISLIN are used in the calculation of view factors by finite difference or contour integration techniques. They are not used with the VIEW3D entry.
4. A summary of the shadowing conditions can be requested by the PARAM,MESH,YES Bulk Data entry.
5. SHADE references shadowing for CHBDYi elements participating in a radiation cavity, the VIEW calculation can involve shadowing.

6. DISLIN should only be used with LINE type CHBDYE and CHBDYP surface elements. DISLIN > 0.0 means into the cavity. See Figure 9-155.

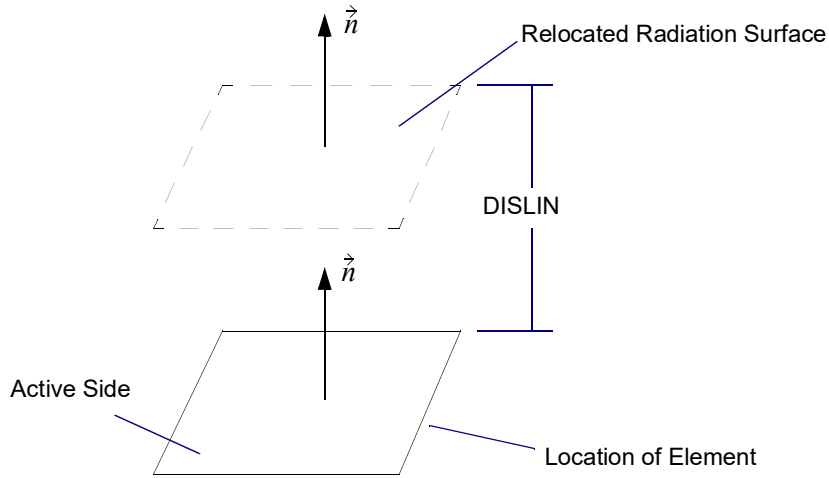


Figure 9-155 DISLIN Convention

7. NB and NG define the subelement mesh refinement when using the VIEW module (as opposed to the VIEW3D module) for the calculation of view factors.

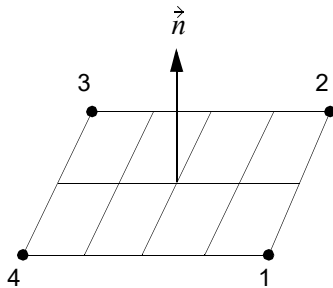


Figure 9-156 Typical AREA4 surface element where NB=2 and NG=4

8. For RC network solver in thermal analysis, the SHADE, NB, NG and DISLIN are ignored.

## VIEW3D

## View Factor Definition - Gaussian Integration Method

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

Format:

| 1      | 2       | 3    | 4    | 5    | 6    | 7    | 8    | 9      | 10 |
|--------|---------|------|------|------|------|------|------|--------|----|
| VIEW3D | ICAVITY | GITB | GIPS | CIER | ETOL | ZTOL | WTOL | RADCHK |    |

Example:

|        |   |   |   |   |  |        |  |  |  |
|--------|---|---|---|---|--|--------|--|--|--|
| VIEW3D | 1 | 2 | 2 | 4 |  | 1.0E-6 |  |  |  |
|--------|---|---|---|---|--|--------|--|--|--|

| Describer | Meaning                                                                                                                                                                                                      |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICAVITY   | Radiant cavity identification number on RADCAV entry. (Integer > 0)                                                                                                                                          |
| GITB      | Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)                                   |
| GIPS      | Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)                                         |
| CIER      | Discretization level used in the semi-analytic contour integration method. ( $1 \leq \text{Integer} \leq 20$ ; Default = 4)                                                                                  |
| ETOL      | Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real $\geq 0.0$ ; Default = 0.1)                                                       |
| ZTOL      | Assumed level of calculation below which the numbers are considered to be zero. (Real $\geq 0.0$ ; Default = 1.E-10)                                                                                         |
| WTOL      | Assumed degree of warpage above which the actual value of $F_{ii}$ will be calculated. ( $0.0 \leq \text{Real} \leq 1.0$ ; Default = 0.01)                                                                   |
| RADCHK    | Type of diagnostic output desired for the radiation exchange surfaces. (Integer; Default = 3)                                                                                                                |
|           | -1 No diagnostic output requested                                                                                                                                                                            |
|           | 1 Grid table and element connectivity                                                                                                                                                                        |
|           | 2 Surface Diagnostics - Surface type, area, skewness, taper, warpage, grid point sequencing, aspect ratio, and shading flags.                                                                                |
|           | 3 Area, view factor, area-view factor product with error estimate, existence flags for partial self-shadowing, third-body shadowing with error estimate, and enclosure summations for view factor. (Default) |
|           | 0 Same as RADCHK = 1, 2, and 3                                                                                                                                                                               |
|           | 12 Same as RADCHK = 1 and 2                                                                                                                                                                                  |

| Describer | Meaning                  |
|-----------|--------------------------|
| 13        | Same as RADCHK = 1 and 3 |
| 23        | Same as RADCHK = 2 and 3 |

**Remarks:**

1. For ETOL, when the error estimate exceeds the value input for the ETOL entry, the contour method is employed to develop an improved view factor.
2. For ZTOL, the use of a geometry scale that results in small numerical values of  $A_i F_{ij}$  should be avoided.
3. When WTOL is exceeded, the actual value of  $F_{ij}$  will be calculated when using the adaptive view module. Warpage will not be considered in the calculation of  $F_{ij}$ .
4. For axisymmetric analysis, RADCHK = -1 or 3 only.

## VIEWEX

## Advanced RC Radiation Solver Parameters - SOL 400

Defines the radiation solver and correlating solver parameters for radiation calculations in RC heat transfer.  
Note: You must have a copy of the external radiation code to use it.

## Format: (NEVADA)

| 1      | 2        | 3                 | 4                  | 5              | 6                       | 7             | 8          | 9            | 10 |
|--------|----------|-------------------|--------------------|----------------|-------------------------|---------------|------------|--------------|----|
| VIEWEX | ICAVITY  | Run Interactively | RADK Distro Method | Orbital        | Re-use existing results |               |            |              |    |
|        | "NEVADA" | RENO Reflection   | Restart            | Reno Ray count | Vegas Ray count         | Energy Cutoff | Confidence | GRID closure |    |
|        |          | GRID iterations   | Time Scale         | RADK cutoff    |                         |               |            |              |    |

## Example:

|        |        |     |      |       |      |    |     |       |  |
|--------|--------|-----|------|-------|------|----|-----|-------|--|
| VIEWEX | 2      | T   | FULL | T     | F    |    |     |       |  |
|        | NEVADA | T   | T    | 5000  | 5000 | -3 | 99. | 0.001 |  |
|        |        | 300 | 1.0  | 1.0-8 |      |    |     |       |  |

## Format: (TSS)

| 1      | 2       | 3                 | 4                  | 5       | 6                       | 7 | 8 | 9 | 10 |
|--------|---------|-------------------|--------------------|---------|-------------------------|---|---|---|----|
| VIEWEX | ICAVITY | Run Interactively | RADK Distro Method | Orbital | Re-use existing results |   |   |   |    |
|        | "TSS"   |                   |                    |         |                         |   |   |   |    |

## Example:

|        |     |   |      |   |   |  |  |  |  |
|--------|-----|---|------|---|---|--|--|--|--|
| VIEWEX | 3   | T | FULL | T | F |  |  |  |  |
|        | TSS |   |      |   |   |  |  |  |  |

## Format: (THERMICA)

| 1      | 2           | 3                 | 4                  | 5                | 6                       | 7                        | 8                   | 9                      | 10 |
|--------|-------------|-------------------|--------------------|------------------|-------------------------|--------------------------|---------------------|------------------------|----|
| VIEWEX | ICAVITY     | Run Interactively | RADK Distro Method | Orbital          | Re-use existing results |                          |                     |                        |    |
|        | "THERMIC A" | Solar Flux        | Planet Albedo      | Planet BlackBody | Restart                 | Suppress VF Articulation | Radiation ray count | Orbital flux ray count |    |
|        |             | Confidence        | Time scale         | RADK cutoff      |                         |                          |                     |                        |    |

## Example:

|        |   |   |      |   |   |  |  |  |  |
|--------|---|---|------|---|---|--|--|--|--|
| VIEWEX | 4 | T | FULL | T | F |  |  |  |  |
|--------|---|---|------|---|---|--|--|--|--|

|  |          |        |     |       |   |   |      |      |  |
|--|----------|--------|-----|-------|---|---|------|------|--|
|  | THERMICA | 1380.0 | 0.3 | -19.0 | T | F | 5000 | 5000 |  |
|  |          | 99.0   | 1.0 | 1.0-8 |   |   |      |      |  |

Format: (TRASYS)

| 1      | 2        | 3                 | 4                  | 5                | 6                       | 7           | 8 | 9 | 10 |
|--------|----------|-------------------|--------------------|------------------|-------------------------|-------------|---|---|----|
| VIEWEX | ICAVITY  | Run Interactively | RADK Distro Method | Orbital          | Re-use existing results |             |   |   |    |
|        | "TRASYS" | Axi Radial mesh   | Axi Axial mesh     | Axi Angular mesh | Time scale              | RADK cutoff |   |   |    |

Example:

|        |        |   |      |   |     |       |  |  |  |
|--------|--------|---|------|---|-----|-------|--|--|--|
| VIEWEX | 5      | T | FULL | T | F   |       |  |  |  |
|        | TRASYS | 1 | 1    | 4 | 1.0 | 1.0-8 |  |  |  |

Format: (SRR)

| 1      | 2       | 3                    | 4                  | 5                 | 6                       | 7                 | 8                | 9               | 10 |
|--------|---------|----------------------|--------------------|-------------------|-------------------------|-------------------|------------------|-----------------|----|
| VIEWEX | ICAVITY | Run Interactively    | RADK Distro Method | Orbital           | Re-use existing results |                   |                  |                 |    |
|        | "SRR"   | Gebhart Solver       | Convergence Tol    | Max Iter          | Fij smoothing method    | Fij Filter cutoff | Fij Smothing Tol | Fij Smooth Iter |    |
|        |         | Bij smoothing method | Bij Filter cutoff  | Bij Smoothing Tol | Bij Max Iter            |                   |                  |                 |    |

Example:

|        |     |      |       |       |      |       |       |    |  |
|--------|-----|------|-------|-------|------|-------|-------|----|--|
| VIEWEX | 6   | T    | FULL  | T     | F    |       |       |    |  |
|        | SRR | GS   | 1.0-5 | 50    | CROP | 1.0-8 | 1.0-4 | 50 |  |
|        |     | CROP | 1.0-8 | 1.0-4 | 50   |       |       |    |  |

Format: (SRQ)

| 1      | 2       | 3                 | 4                  | 5        | 6                       | 7             | 8                 | 9               | 10 |
|--------|---------|-------------------|--------------------|----------|-------------------------|---------------|-------------------|-----------------|----|
| VIEWEX | ICAVITY | Run Interactively | RADK Distro Method | Orbital  | Re-use existing results |               |                   |                 |    |
|        | "SRQ"   | Flux solver       | Convergence Tol    | Max Iter | Fij smoothing method    | Filter cutoff | Fij Smoothing Tol | Fij Smooth Iter |    |

Example:

|        |     |    |       |    |      |       |       |    |  |
|--------|-----|----|-------|----|------|-------|-------|----|--|
| VIEWEX | 7   | T  | FULL  | T  | F    |       |       |    |  |
|        | SRQ | GS | 1.0-5 | 50 | CROP | 1.0-8 | 1.0-4 | 50 |  |



| Describer                | Meaning                                                                                                                                                                      |
|--------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ICAVITY                  | Cavity identification number (Integer > 0; Required)                                                                                                                         |
| Run Interactively        | Run the radiation code interactively (Character; "T" or "F", Default "F")<br>Do not currently have batch mode for Thermica or TSS<br>Do not have interactive mode for TRASYS |
| RADK Distro Method       | How to redistribute RADK onto elements. (Character; "FULL", "AREA", or "DIRECT"; Default "FULL")                                                                             |
| Orbital                  | Use orbital analysis for radiation (Character; "T" or "F"; Default "F")<br>Not supported in SindaRad                                                                         |
| Re-use existing results  | Re-use previous radiation results (Character; "T" or "F"; Default "F")                                                                                                       |
| "NEVADA"                 | Identification that NEVADA will be used (Character)                                                                                                                          |
| RENO Reflection          | Use reflection method of ray tracing for RADK (Character; "T" or "F"; Default "T")                                                                                           |
| Restart                  | Use Restart (Character; "T" or "F"; Default "F")                                                                                                                             |
| Reno Ray count           | Number or rays cast for Reno module (Integer > 0; Default 5000)                                                                                                              |
| Vegas Ray count          | Number or rays cast for Vegas module (Integer > 0; Default 5000)                                                                                                             |
| Energy Cutoff            | Energy cutoff level (Integer; Default -3)                                                                                                                                    |
| Confidence               | Confidence Level % (Real > 0.0; Default 99.0)                                                                                                                                |
| GRID closure             | GRID closure tolerance (Real > 0.0; Default 0.001)                                                                                                                           |
| GRID iterations          | Maximum GRID iterations (Integer > 0; Default 300)                                                                                                                           |
| Time Scale               | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)                                           |
| RADK cutoff              | RADK filter smallest element (Real ≥ 0.0; Default 1.0e-8)                                                                                                                    |
| "TSS"                    | Identification that TSS will be used (Character)                                                                                                                             |
| "THERMICA"               | Identification that THERMICA will be used (Character)                                                                                                                        |
| Solar Flux               | Quantity of solar flux (Real > 0.0; Default 1380.0 W/m <sup>2</sup> )                                                                                                        |
| Planet Albedo            | Planetary Albedo (Real; Default 0.3; assumes Earth orbit)                                                                                                                    |
| Planet BlackBody         | Planet Blackbody (Real; Default -19.0; assumes Earth orbit)                                                                                                                  |
| Restart                  | Use Restart option (Character, "T" or "F"; Default "F")                                                                                                                      |
| Suppress VF Articulation | Suppress view factor articulation (Character; "T" or "F"; Default "T")                                                                                                       |
| Radiation ray count      | Number or rays cast for radiation calculation (Integer > 0; Default 5000)                                                                                                    |
| Orbital flux ray count   | Number or rays cast for orbital flux (Integer > 0; Default 5000)                                                                                                             |
| Confidence               | Confidence Level % (Real > 0.0; Default 99.0)                                                                                                                                |
| Time Scale               | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0)                                           |

| Describer            | Meaning                                                                                                                            |
|----------------------|------------------------------------------------------------------------------------------------------------------------------------|
| RADK cutoff          | RADK filter smallest element (Real $\geq$ 0.0; Default 1.0e-8)                                                                     |
| “TRASYS”             | Identification that TRASYS will be used (Character)                                                                                |
| Axi Radial mesh      | Axisymmetric element mesh in radial direction (Integer > 0; Default 1)                                                             |
| Axi Axial mesh       | Axisymmetric element mesh in axial direction (Integer > 0; Default 1)                                                              |
| Axi Angular mesh     | Axisymmetric element mesh in angular direction (Integer > 0; Default 4)                                                            |
| Time Scale           | Orbital time scale factor, number of time units in an hour. Ex. If using seconds, value would be 3600.0. (Real > 0.0; Default 1.0) |
| RADK cutoff          | RADK filter smallest element (Real $\geq$ 0.0; Default 1.0e-8)                                                                     |
| “SRR”                | Identification that the SindaRad RADK method will be used (Character)                                                              |
| Gebhart Solver       | Which RADK solver to use (Character; “GS” or “FGS”; Default “GS”)                                                                  |
| Convergence Tol      | Tolerance for convergence of RADK calculation (Real $\geq$ 0.0; Default 1.0e-5)                                                    |
| Max Iter             | Maximum allowable iterations to converge (Integer > 0; Default 50)                                                                 |
| Fij smoothing method | How to filter view factors (Character; “CROP” or “HIGH”; Default “CROP”)                                                           |
| Fij Filter cutoff    | Parameter for filter (Real > 0.0; Default 1.0e-8)                                                                                  |
| Fij Smoothing Tol    | Tolerance for smoothing (Real $\geq$ 0; Default 1.0e-4)                                                                            |
| Fij Smooth Iter      | Maximum allowable iterations to smoothing (Integer > 0; Default 50)                                                                |
| Bij smoothing method | How to filter conductors (Character; “CROP” or “HIGH”; Default “CROP”)                                                             |
| Bij Filter cutoff    | Parameter for filter (Real > 0.0; Default 1.0e-8)                                                                                  |
| Bij Smoothing Tol    | Tolerance for smoothing (Real $\geq$ 0; Default 1.0e-4)                                                                            |
| Bij Max Iter         | Maximum allowable iterations to smoothing (Integer > 0; Default 50)                                                                |
| “SRQ”                | Identification that the SindaRad QRad method will be used (Character)                                                              |
| Flux Solver          | Which QRad solver to use (Character, “GS” or “CG”; Default “GS”)                                                                   |
| Convergence Tol      | Tolerance for convergence of QRad calculation (Real $\geq$ 0.0; Default 1.0e-5)                                                    |
| Max Iter             | Maximum allowable iterations to converge (Integer > 0; Default 50)                                                                 |
| Fij smoothing method | How to filter view factors (Character, “CROP” or “HIGH”; Default “CROP”)                                                           |
| Fij Filter cutoff    | Parameter for filter (Real > 0.0; Default 1.0e-8)                                                                                  |
| Fij Smoothing Tol    | Tolerance for smoothing (Real $\geq$ 0; Default 1.0e-4)                                                                            |
| Fij Smooth Iter      | Maximum allowable iterations to smoothing (Integer > 0; Default 50)                                                                |

#### Remarks:

1. This entry is for RC Network solver only.
2. Each entry type is designed for one specific radiation solver, except the very last two types, which are for SindaRad’s two options.

NEVADA  
TSS  
THERMICA  
TRASYS  
SINDARad RADK method  
SINDARad Q method

3. For more details about the parameters in the entry, please reference SINDA for *Patran User's Guide* and the *SINDARad User's Guide*.
4. MSC provides these radiation programs on Windows only.

## WALL

## Rigid Wall

Defines a rigid plane through which specified Lagrangian grid points cannot penetrate. Finite or infinite size (FINITE), Orthotropic friction (ORTHO) a mass and an initial velocity (MOVING) and Force output option (FORCES) can be defined. Used in SOL 700 only.

Format:

| 1    | 2      | 3  | 4  | 5   | 6  | 7  | 8  | 9      | 10 |
|------|--------|----|----|-----|----|----|----|--------|----|
| WALL | ID     | XP | YP | ZP  | NX | NY | NZ | BCGRID |    |
|      | METHOD | FS | FK | EXP |    |    |    |        |    |

Example:

|      |         |     |  |  |  |  |     |    |  |
|------|---------|-----|--|--|--|--|-----|----|--|
| WALL | 17      |     |  |  |  |  | 1.0 | 21 |  |
|      | PENALTY | 0.2 |  |  |  |  |     |    |  |

| Describer  | Meaning                                                                                                                                                                         |
|------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| ID         | Unique rigid-wall number. (Integer > 0; Required)                                                                                                                               |
| XP, YP, ZP | Coordinates of the origin of the wall. (Real; Default = 0.0)                                                                                                                    |
| NX, NY, NZ | A vector normal to the wall pointing towards the model. (Real; Default = 0.0)                                                                                                   |
| BCGRID     | Number of a BCGRID entry listing the points that cannot penetrate the wall. (Integer > 0; Required)                                                                             |
| METHOD     | Algorithm for contact processing. (Character; Default = PENALTY)                                                                                                                |
|            | PENALTY      Penalty method, allowing for extra boundary conditions, friction and output.                                                                                       |
|            | KINMATIC      Kinematic method, only included for compatibility reasons with older Dytran versions. This method allows no extra boundary conditions, no friction and no output. |
| FS         | Static coefficient of friction. See Remark 5. (Real $\geq$ 0; Default = 0.0)                                                                                                    |
| FK         | Kinetic coefficient of friction. See Remark 5. (Real $\geq$ 0; Default = 0.0)                                                                                                   |
| EXP        | Exponential decay coefficient. See Remark 5. (Real $\geq$ 0; Default = 0.0)                                                                                                     |

Remarks

1. A rigid plane of infinite size is generated that the grid points cannot penetrate. The plane is fixed in space.
2. The grid points can slide on the wall and separate from it.
3. A (moving) rigid plane of finite size can be modeled by using a rigid surface and a primary-secondary contact.

4. For the wall definition using penalty method, output can be requested by referencing it in a SET command in the Case Control Section. The keywords for output are WALLS and WALLOUT. Please check \$S700.
5. The coefficient of friction is given by:

$$\mu = \mu_k + (\mu_s - \mu_k)e^{-\beta v}$$

where

$\mu_s$  = Static coefficient of friction FS.

$\mu_k$  = Kinetic coefficient of friction FK.

$\beta$  = Exponential decay coefficient EXP.

$v$  = Relative sliding velocity at the point of contact.

## WEAR Specifies Values for Modeling Mechanical Wear in Deformable Contact Bodies

Specifies values for modeling mechanical wear in deformable contact bodies for SOL 600 only.

Format:

| 1    | 2       | 3       | 4       | 5       | 6    | 7    | 8 | 9 | 10 |
|------|---------|---------|---------|---------|------|------|---|---|----|
| WEAR | BID     | Model   | IUPD    |         |      |      |   |   |    |
|      | COEF    | HARD    | SEXP    | VEXP    | B    | SFGU |   |   |    |
|      | T(COEF) | T(HARD) | T(SEXP) | T(VEXP) | T(B) |      |   |   |    |

Example:

|      |     |      |     |     |      |     |  |  |  |
|------|-----|------|-----|-----|------|-----|--|--|--|
| WEAR | 3   | 2    | 1   |     |      |     |  |  |  |
|      | 3.6 | 200. | 1.1 | 1.2 | .087 | 1.0 |  |  |  |
|      | 22  | 23   | 24  | 25  | 26   |     |  |  |  |

| Describer | Meaning                                                                                                           |
|-----------|-------------------------------------------------------------------------------------------------------------------|
| BID       | BCBODY ID for which this wear entry applies. (Integer > 0; no Default; see Remark 2)                              |
| MODEL     | Wear model. (Integer > 0; Default = 1, see Remark 1)                                                              |
| IUPD      | Geometry update due to wear flag. (Integer; Default = 0)<br>0 Do not update the geometry<br>1 Update the geometry |
| COEF      | Wear coefficient. (Real; no Default)                                                                              |
| HARD      | Material hardness. (Real; no Default)                                                                             |
| SEXP      | Stress exponent. (Real; Default = 1.0)                                                                            |
| VEXP      | Velocity exponent. (Real; Default = 1.0)                                                                          |
| B         | Thermal activation constant. (Real; Default = 1.0)                                                                |
| SFGU      | Scale factor for geometry update. (Real; Default = 1.0)                                                           |
| T(COEF)   | TABL3D ID for the wear coefficient. (Integer; Default = 0, meaning no table variation)                            |
| T(HARD)   | TABL3D ID for the material hardness. (Integer; Default = 0, meaning no table variation)                           |
| T(SEXP)   | TABL3D ID for the stress exponent. (Integer; Default = 0; meaning no table variation)                             |
| T(VEXP)   | TABL3D ID for the velocity exponent. (Integer; Default = 0, meaning no table variation)                           |
| T(B)      | TABL3D ID for the thermal activation constant. (Integer; Default = 0, meaning no table variation)                 |

## Remarks:

1. The following wear models are available

$$\dot{w} = \frac{K}{H} \sigma v_{rel} \quad \text{Archard base model}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n \quad \text{Archard model; Bayer exponential form}$$

$$\dot{w} = \frac{K}{H} \sigma^m v_{rel}^n e^{\frac{-B}{T}} \quad \text{Archard model; exponential form with thermal activation}$$

2. BID may also point the ID in field 2 of BCPROP, BCMATL or BCBOX.

**WETELME**

Wetted element in OpenFSI by side in SOL 400

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2    | 3   | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|-----|------|---|---|---|---|---|----|
| WETELME | WEID | EID | SIDE |   |   |   |   |   |    |

Example:

|         |       |    |   |  |  |  |  |  |  |
|---------|-------|----|---|--|--|--|--|--|--|
| WETELME | 10001 | 34 | 3 |  |  |  |  |  |  |
|---------|-------|----|---|--|--|--|--|--|--|

| Describer | Meaning |
|-----------|---------|
|-----------|---------|

WEID      Wetted element identification number. (Integer > 0; no Default)

EID      Structural element identification number, which corresponds to a surface element CQUAD4, CQUAD8, CQUADR, CTRIA3, CTRIA6, CTRIAR; or a solid element CTETRA, CPENTA, or CHEXA. (Integer > 0; no Default)

SIDE      Side identification number of element EID. ( $1 \leq \text{Integer} \leq 6$ ; no Default)

Remarks:

1. Refer to the remarks [6.](#) and [7.](#) for the element side identification of CHBDYE entry.



**WETELMG****Wetted element in OpenFSI by type in SOL 400**

Defines a wetted element used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2    | 3    | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|---------|------|------|----|----|----|----|----|----|----|
| WETELMG | WEID | TYPE |    |    |    |    |    |    |    |
|         | G1   | G2   | G3 | G4 | G5 | G6 | G7 | G8 |    |

Example:

|         |       |       |     |    |  |  |  |  |  |
|---------|-------|-------|-----|----|--|--|--|--|--|
| WETELMG | 10001 | QUAD4 |     |    |  |  |  |  |  |
|         | 23    | 35    | 124 | 28 |  |  |  |  |  |

| Describer   | Meaning                                                                                                      |
|-------------|--------------------------------------------------------------------------------------------------------------|
| WEID        | Wetted element identification number. (Integer > 0; no Default)                                              |
| TYPE        | Wetted element type, which can be any of TRIA3, TRIA6, QUAD4, QUAD8, LINE2 or LINE3. (Character; no Default) |
| G1, ..., G8 | Grid point identification numbers for the wetted surface element WEID. (Integer > 0; no Default)             |

## WETLOAD Load set and associated wetted surface for OpenFSI in SOL 400

Defines the load set and associated wetted surface for OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2    | 3    | 4       | 5 | 6 | 7 | 8 | 9 | 10 |
|---------|------|------|---------|---|---|---|---|---|----|
| WETLOAD | WLID | WSID | SERV_ID |   |   |   |   |   |    |

Example:

|         |   |      |        |  |  |  |  |  |  |
|---------|---|------|--------|--|--|--|--|--|--|
| WETLOAD | 1 | 1001 | scafsi |  |  |  |  |  |  |
|---------|---|------|--------|--|--|--|--|--|--|

| Describer | Meaning                                                                                                                                                            |
|-----------|--------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| WLID      | Load set ID, referenced by the EXCITEID field in the TLOAD entry. (Integer > 0; no Default)                                                                        |
| WSID      | Wetted surface identification number. The wetted surface must be defined in the WETSURF Bulk Data entry. (Integer > 0; no Default)                                 |
| SERV_ID   | OpenFSI SCA service name associated with the wetted surface loads. The OpenFSI SCA service is defined using the CONNECT SERVICE FMS entry. (Character; no Default) |

Remarks:

1. To activate WETLOAD, the following entries must be present and referenced:
  - a. The load set ID WLID must be referenced by a TLOAD1 entry.
  - b. The TLOAD1 entry must be referenced by a DLOAD Case Control command or a DLOAD Bulk Data entry, which in turn is referenced by a DLOAD Case Control command.
  - c. The TLOAD1 entry must reference a TABLED1 entry.
  - d. The TABLED1 entry must have constant values and be the same for all TLOAD1 entries referenced by the WETLOAD entries used within the same service.

Procedures a ~ d are required in the Nastran load generator processes. Their values do not really influence the load calculation of WETLOAD in OpenFSI. Please refer to the remarks below to apply WETLOAD(s) to OpenFSI.
2. The SERV\_ID string must be 8 characters or less.
3. For multiple WETLOAD entries with same SERV\_ID, the wet surfaces in WSID will be merged together for the service and the WLID in the first WETLOAD entry will replace other entries WLID.
4. In the current implementation, all OpenFSI services in a model are active and participate in the calculation. The TLOAD1 and DLOAD selection in case control are not applied on OpenFSI services.
5. For OpenFSI services, the scale factors in DLOAD are always taken to be constant 1.0.

6. In the FMS Section, the following statement should be included: `CONNECT SERVICE SERV_ID  
ExternalCodeVendor.OpenFSI`

## WETSURF

Wetted surface used in OpenFSI in SOL 400

Defines a wetted surface used in OpenFSI fluid structure interaction simulations in SOL 400 only.

Format:

| 1       | 2     | 3      | 4      | 5     | 6     | 7     | 8     | 9     | 10 |
|---------|-------|--------|--------|-------|-------|-------|-------|-------|----|
| WETSURF | WSID  | WTAG   |        |       |       |       |       |       |    |
|         | WEID1 | WEID2  | WEID3  | WEID4 | WEID5 | WEID6 | WEID7 | WEID8 |    |
|         | WEID9 | WEID10 | -etc.- |       |       |       |       |       |    |

Alternate Format:

|         |       |        |       |      |     |  |  |  |  |
|---------|-------|--------|-------|------|-----|--|--|--|--|
| WETSURF | WSID  | WTAG   |       |      |     |  |  |  |  |
|         | WEID1 | “THRU” | WEID2 | “BY” | INC |  |  |  |  |

Example:

|         |       |       |     |    |    |    |    |    |  |
|---------|-------|-------|-----|----|----|----|----|----|--|
| WETSURF | 10001 | wall1 |     |    |    |    |    |    |  |
|         | 5     | THRU  | 21  | BY | 4  |    |    |    |  |
|         | 27    | 30    | 33  |    |    |    |    |    |  |
|         | 35    | THRU  | 44  |    |    |    |    |    |  |
|         | 67    | 68    | 70  | 72 | 77 | 82 | 86 | 79 |  |
|         | 89    | THRU  | 110 | BY | 3  |    |    |    |  |

| Describer         | Meaning                                                                                                                 |
|-------------------|-------------------------------------------------------------------------------------------------------------------------|
| WSID              | Wetted surface identification number. (Integer > 0; no Default)                                                         |
| WTAG              | Wetted surface tag name exported to an external code using the OpenFSI SCA interface. (Character; no Default)           |
| WEID1, WEID2, ... | Wetted element identification numbers defined using the WETELMG or WETELME Bulk Data entries. (Integer > 0; no Default) |
| THRU, BY          | Keywords to specify a range of wetted elements. (Character; no Default)                                                 |
| INC               | Increment to use with the “THRU” and “BY” keywords. (Integer; Default = 1)                                              |

## YLDHY

## Hydrodynamic Yield Model

Defines a yield model with zero yield stress. Used in SOL 700 only.

Format:

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-----|---|---|---|---|---|---|---|----|
| YLDHY | YID |   |   |   |   |   |   |   |    |

Example:

|       |     |  |  |  |  |  |  |  |  |
|-------|-----|--|--|--|--|--|--|--|--|
| YLDHY | 200 |  |  |  |  |  |  |  |  |
|-------|-----|--|--|--|--|--|--|--|--|

| Describer | Meaning                                                                            |
|-----------|------------------------------------------------------------------------------------|
| YID       | Unique yield-model number referenced from a MATDEUL entry. (Integer > 0; Required) |

Remark:

1. This yield model should be used for fluids that have no shear strength.
2. YID must unique among all YLDxx entries in one model.

## YLDJC

## Johnson-Cook Yield Model

Defines a Johnson-Cook yield model where the yield stress is a function of effective plastic strain, strain rate, and temperature. Used in SOL 700 only.

Format:

| 1     | 2   | 3     | 4     | 5   | 6   | 7   | 8    | 9    | 10 |
|-------|-----|-------|-------|-----|-----|-----|------|------|----|
| YLDJC | YID | A     | B     | n   | C   | m   | EPS0 | CP   |    |
|       | 100 | 200E6 | 50.E6 | 0.1 | .15 | .95 | 1.   | 285. |    |

Example:

|       |       |       |  |  |  |  |  |  |  |
|-------|-------|-------|--|--|--|--|--|--|--|
| YLDJC | TMELT | TROOM |  |  |  |  |  |  |  |
|       | 1500. | 273.  |  |  |  |  |  |  |  |

| Describer | Meaning                                                                            |
|-----------|------------------------------------------------------------------------------------|
| YID       | Unique yield-model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A         | Static yield stress. (Real ≥ 0.0; Required)                                        |
| B         | Hardening parameter. (Real; Default = 0.0)                                         |
| n         | Hardening exponent. (Real; Default = 1.0)                                          |
| C         | Strain-rate parameter. (Real; Default = 0.0)                                       |
| m         | Temperature exponent. (Real; Default = 1.0)                                        |
| EPS0      | Reference strain rate. (Real > 0.0; Default = 1.0)                                 |
| CP        | Specific heat. (Real > 0.0; Default = 1.E20)                                       |
| TMELT     | Melt temperature. (Real; Default = 1.E20)                                          |
| TROOM     | Room temperature. (Real; Default = 293.0)                                          |

Remark:

1. The yield stress is computed from

$$\sigma_y = (A + B\varepsilon_p^n) \left( 1 + C \ln \left( \frac{\dot{\varepsilon}}{\varepsilon_0} \right) \right) (1 - T^{*m})$$

where

$$\begin{aligned} \varepsilon_p &= \text{effective plastic strain} \\ T^* &= \frac{(T - T_r)}{(T_m - T_r)} \end{aligned}$$

|                    |   |                        |
|--------------------|---|------------------------|
| $\dot{\epsilon}$   | = | effective strain rate  |
| $\dot{\epsilon}_0$ | = | referenced strain rate |
| $T$                | = | temperature            |
| $T_r$              | = | room temperature       |
| $T_m$              | = | melt temperature       |

and  $A$ ,  $B$ ,  $n$ ,  $C$ , and  $m$  are constants.

2. The reference strain rate is per unit time.
3. YID must unique among all YLDxx entries in one model.

## YLDMC

## Mohr-Coulomb Yield Model

Defines a Mohr-Coulomb yield model. Used in SOL 700 only.

Format:

|       |     |    |    |    |   |   |   |   |    |
|-------|-----|----|----|----|---|---|---|---|----|
| 1     | 2   | 3  | 4  | 5  | 6 | 7 | 8 | 9 | 10 |
| YLDMC | YID | Y1 | Y2 | Y3 |   |   |   |   |    |

Example:

|       |   |       |       |      |  |  |  |  |  |
|-------|---|-------|-------|------|--|--|--|--|--|
| YLDMC | 1 | 10.E5 | 20.E5 | 1.E4 |  |  |  |  |  |
|-------|---|-------|-------|------|--|--|--|--|--|

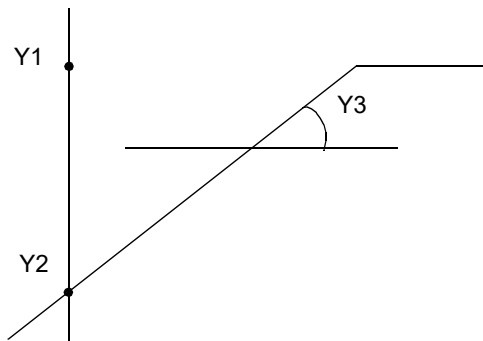
| Describer | Meaning                                                                                                              |
|-----------|----------------------------------------------------------------------------------------------------------------------|
| YID       | Unique yield-model number referenced from MATDEUL for Eulerian elements with shear strength. (Integer > 0; Required) |
| Y1        | Cap yield stress. (Real; Required)                                                                                   |
| Y2        | Cohesion. (Real; Required)                                                                                           |
| Y3        | Internal friction angle. (Real; Required)                                                                            |

Remarks:

1. The yield stress depends on the pressure as

$$\sigma_y = \text{MIN}(Y1, (Y2 + Y3 \cdot P))$$

where Y1, Y2, Y3 are constants and  $P$  is the pressure.



2. This yield model is applicable only for Eulerian materials with shear strength.
3. YID must unique among all YLDxx entries in one model.



## YDMSS

## Multi-Surface Yield Model for Snow

Defines the yield model for snow material. This entry must be used in combination with MATDEUL, EOSPOL and SHREL. Used in SOL 700 only.

Format:

| 1     | 2   | 3     | 4  | 5    | 6    | 7   | 8    | 9   | 10 |
|-------|-----|-------|----|------|------|-----|------|-----|----|
| YDMSS | YID | KC    | T  | CC   | AC   | BC  | FC   | FTU |    |
|       | 7   | 0.149 | 82 | 1E-5 | 0.09 | 0.2 | 0.99 | 82  |    |

Example:

|       |       |     |  |  |  |  |  |  |  |
|-------|-------|-----|--|--|--|--|--|--|--|
| YDMSS | ALP0  | DS  |  |  |  |  |  |  |  |
|       | -0.37 | 0.0 |  |  |  |  |  |  |  |

| Describer | Meaning                                                                              |
|-----------|--------------------------------------------------------------------------------------|
| YID       | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required)   |
| KC        | Parameter related to the angle of friction. (Real > 0; Required)                     |
| T         | Equivalent value of the snow cohesion. see Remark 5. (Real > 0; Required)            |
| CC        | Shape of the yield surface. See Remark 4. (Real > 0; Required)                       |
| AC        | Hardening parameter for compression. See Remark 4. (Real > 0; Required)              |
| BC        | Hardening parameter for compression. See Remark 4. (Real > 0; Required)              |
| FC        | Factor to avoid singularity. See Remark 4. (0 < Real < 1; Default = 0.99)            |
| FTU       | Hydrostatic tensile strength. See Remark 6. (Real > 0; Default = T/3)                |
| ALP0      | Initial compressive volumetric plasticity strain. See Remark 4. (Real < 0; Required) |
| DS        | Softening modulus. See Remark. (Real ≥ 0; Default = 0.0)                             |

Remarks:

1. This material model can be used to model Snow material.
2. In addition to deviatoric plastic strain there is also volumetric plastic strain. This volumetric strain is stored in the output variable VOLPLS. The deviatoric strain is stored in the variable EFFPLS.
3. For detail description of each parameter in this model, the user should refer to the Theory Manual in which the mechanical properties of snow are described.
4. If CC is set to zero then the material behaves as a Drucker-Prager model. Ac, Bc, Fc\* and ALP0 will be ignored.

5. The T value must be consistently converted from the cohesion data,  $C_{DP}$ , of the Drucker-Prager model as follows:

$$T = C_{DP}/K_C$$

6. FTU, hydrostatic tensile strength, may not be greater than T divided by 3. Otherwise it will be set to that value.
7. The softening modulus is used to update the hardening parameter  $q_t$ , see Theory Manual. It can be requested as output using FTU variable. The corresponding accumulated-plastic-volumetric-tensile-strain variable is SOFTE.
8. This material model is valid for the Euler with Strength solver and the Multi-material Euler with Strength solver.
9. YID must unique among all YLDxx entries in one model.

## YLDAPOL

## Polynomial Yield Model

Defines a polynomial yield model where the yield stress is a function of effective plastic strain. Used in SOL 700 only.

Format:

| 1       | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9    | 10 |
|---------|-----|---|---|---|---|---|---|------|----|
| YLDAPOL | YID | A | B | C | D | E | F | SMAX |    |

Example:

|         |   |        |  |  |  |  |  |  |  |
|---------|---|--------|--|--|--|--|--|--|--|
| YLDAPOL | 7 | 180.E6 |  |  |  |  |  |  |  |
|---------|---|--------|--|--|--|--|--|--|--|

| Descriptor | Meaning                                                                    |
|------------|----------------------------------------------------------------------------|
| YID        | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A          | Initial yield parameters. (Real > 0; Required)                             |
| B          | Coefficient B. (Real; Default = 0.0)                                       |
| C          | Coefficient C. (Real; Default = 0.0)                                       |
| D          | Coefficient D. (Real; Default = 0.0)                                       |
| E          | Coefficient E. (Real; Default = 0.0)                                       |
| F          | Coefficient F. (Real; Default = 0.0)                                       |
| SMAX       | Maximum yield stress. (Real; Default = 1.E20)                              |

Remarks:

- The yield stress is computed from

$$\sigma_y = \text{MIN}(\sigma_{\max}, A + B\varepsilon_p + C\varepsilon_p^2 + D\varepsilon_p^3 + E\varepsilon_p^4 + F\varepsilon_p^5)$$

where

$$\begin{aligned} \varepsilon_p &= \text{effective plastic strain} \\ \sigma_{\max} &= \text{maximum yield stress} \end{aligned}$$

and  $A$ ,  $B$ ,  $C$ ,  $D$ ,  $E$  and  $F$  are constants.

- YID must be unique among all YLDxx entries in one model.

## YLDRPL

## Rate Power Law Yield Model

Defines a rate power law yield model where the yield stress is a function of effective plastic strain and strain rate. Used in SOL 700 only.

Format:

| 1      | 2   | 3 | 4 | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|---|---|---|---|---|---|---|----|
| YLDRPL | YID | A | B | n | m | C |   |   |    |

Example:

|        |   |        |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|
| YLDRPL | 7 | 180.E6 |  |  |  |  |  |  |  |
|--------|---|--------|--|--|--|--|--|--|--|

| Describer | Meaning                                                                    |
|-----------|----------------------------------------------------------------------------|
| YID       | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A         | Initial yield parameters. (Real > 0; Required)                             |
| B         | Hardening parameter. (Real; Default = 0.0)                                 |
| N         | Hardening exponent. (Real; Default = 1.0)                                  |
| M         | Strain rate exponent. (Real; Default = 1.0)                                |
| C         | Minimum yield stress. (Real; Default = 1.E20)                              |

Remarks:

1. The yield stress is computed from

$$\sigma_y = \text{MAX}(C, A + B \varepsilon_p^n \dot{\varepsilon}^m)$$

where

|                     |   |                          |
|---------------------|---|--------------------------|
| $\varepsilon_p$     | = | effective plastic strain |
| $\dot{\varepsilon}$ | = | effective strain rate    |

and  $A$ ,  $B$ ,  $n$ ,  $m$  and  $C$  are constants.

2. YID must unique among all YLDxx entries in one model.

## YLD SG

## Steinberg-Guinan Yield Model

Defines the Steinberg-Guinan yield model where the yield stress is a function of effective plastic strain, pressure and temperature. Used in SOL 700 only.

Format:

| 1      | 2     | 3     | 4  | 5  | 6  | 7 | 8 | 9  | 10 |
|--------|-------|-------|----|----|----|---|---|----|----|
| YLD SG | YID   | A1    | A2 | A3 | A4 | H | B | CP |    |
|        | TMELT | TROOM |    |    |    |   |   |    |    |

Example:

|        |      |      |        |     |     |  |  |  |  |
|--------|------|------|--------|-----|-----|--|--|--|--|
| YLD SG | 7    | 8E+6 | 100E+6 | 110 | 0.5 |  |  |  |  |
|        | 1500 | 273  |        |     |     |  |  |  |  |

| Descriptor | Meaning                                                                            |
|------------|------------------------------------------------------------------------------------|
| YID        | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A1-A4      | Yield parameters. (Real > 0; Required)                                             |
| H, B       | Yield parameters. See Remark 4. (Real; Default = 0.0)                              |
| CP         | Specific heat. (Real > 0; Default = 1.E20)                                         |
| TMELT      | Melt temperature. (Real; Default = 1.E20)                                          |
| TROOM      | Room temperature. (Real; Default = 293.0)                                          |

Remarks:

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$A_T = A_1(1 + A_3 \varepsilon_p)^A$$

$$\sigma_y = \min(A_2, A_T) \left[ 1 - H(T - T_r) + Bp \left( \frac{p}{p_{ref}} \right)^{\frac{1}{3}} \right] \quad T < T_m$$

$$\sigma_y = 0, T \geq T_m$$

And  $A_1, \dots, A_4, H$  and  $B$  are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.

## YLDTM

## Tanimura-Mimura Yield Model

Defines the Tanimura-Mimura yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

Format:

| 1     | 2     | 3     | 4   | 5 | 6 | 7    | 8    | 9  | 10 |
|-------|-------|-------|-----|---|---|------|------|----|----|
| YLDTM | YID   | A     | B   | C | D | M    | EPSM | CP |    |
|       | TMELT | TROOM | SCR | E | K | EPS0 |      |    |    |

Example:

|       |   |        |         |       |     |     |       |  |  |
|-------|---|--------|---------|-------|-----|-----|-------|--|--|
| YLDTM | 7 | 45.6E6 | 19.5E6  |       |     | 1.0 | 0.001 |  |  |
|       |   |        | 4000.e6 | 2100. | 0.5 | 1.0 |       |  |  |

| Describer | Meaning                                                                    |
|-----------|----------------------------------------------------------------------------|
| YID       | Unique yield model number referenced from MATDEUL. (Integer > 0; Required) |
| A         | Static yield parameter. (Real > 0; Required)                               |
| B         | Hardening parameter. (Real; Default = 0.0)                                 |
| C         | Strain rate parameter C. (Real; Default = 0.0)                             |
| D         | Strain rate parameter D. (Real; Default = 0.0)                             |
| M         | Temperature exponent. (Real; Default = 0.0)                                |
| EPSM      | Quasi-static strain rate. (Real > 0; Default = 1.0)                        |
| CP        | Specific heat. (Real > 0; Default = 1.E20)                                 |
| TMELT     | Melt temperature. (Real; Default = 1.E20)                                  |
| TROOM     | Room temperature. (Real; Default = 293.0)                                  |
| SCR       | Critical yield stress. (Real > 0; Default = 1.0)                           |
| E         | Strain rate parameter E. (Real; Default = 0.0)                             |
| K         | Strain rate exponent. (Real; Default = 1.0)                                |
| EPS0      | Reference strain rate. (Real > 0; Default = 1.0)                           |

Remarks:

1. This material model can be used to model metals for a wide range of strain rates.
2. The yield stress is computed from

$$\sigma_Y = \left[ A + B\varepsilon_P + (C + D\varepsilon_P) \left( 1 - \frac{A + B\varepsilon_P}{\sigma_{cr}} \right) \ln \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_s} \right) \right] (1 - T^{*m}) + E \left( \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \right)^k$$

where

|                       |   |                           |
|-----------------------|---|---------------------------|
| $\varepsilon_P$       | = | effective plastic strain  |
| $\sigma_{cr}$         | = | critical yield stress     |
| $\dot{\varepsilon}$   | = | effective strain rate     |
| $\dot{\varepsilon}_s$ | = | quasi-static strain rate  |
| $\dot{\varepsilon}_0$ | = | reference strain rate     |
| $T^*$                 | = | $(T - T_r) / (T_m - T_r)$ |
| $T$                   | = | temperature               |
| $T_r$                 | = | room temperature          |
| $T_m$                 | = | melt temperature          |

and  $A, B, C, D, m, E$  and  $k$  are constants.

3. The reference and quasi-static strain rate are per unit time.
4. YID must unique among all YLDxx entries in one model.



## YLDUDS

## User-defined Yield Model for Elements.

Specifies that a user subroutine is being used to define a simple yield model. Use in SOL700 only.

Format:

| 1      | 2   | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|-----|-------|-------|---|---|---|---|---|----|
| YLDUDS | YLD | GROUP | UNAME |   |   |   |   |   |    |

Example:

*In FMS Section of the MSC Nastran input stream:*

```
CONNECT SERVICE myyld 'SCA.MDSolver.Obj.Uds.Dytran.Materials'
```

*In Bulk Data:*

| 1      | 2  | 3     | 4     | 5 | 6 | 7 | 8 | 9 | 10 |
|--------|----|-------|-------|---|---|---|---|---|----|
| YLDUDS | 12 | myyld | EXYLD |   |   |   |   |   |    |

| Describer | Meaning                                                                                    |
|-----------|--------------------------------------------------------------------------------------------|
| YLD       | Unique yield model ID. (Integer > 0; Required)                                             |
| GROUP     | The group name used for the FMS section CONNECT SERVICE statement. (Character; no Default) |
| UNAME     | User subroutine name associated with the entry. (Character; default=EXYLD)                 |

Remarks:

1. Only can be used for SOL 700.
2. The YLD must be referenced by a MATDEUL or MAT1 entry.
3. This yield model is applicable only for Euler elements with shear strength
4. UNAME can be:

| Subroutine Name | Function                          |
|-----------------|-----------------------------------|
| EXYLD           | Standard user defined yield model |

5. The yield strength can depend on the amount of failure or damage of the Euler element and on the VOLPLS and SOFTE element variables. This amount of damage can be either specified by the damage variable of the FAILJC entry or by a more general failure estimate by using the FAILUDS entry with UNAME=EXFAIL2. The VOLPLS and SOFTE variables can also be filled by the YLDUDS entry and FAILUDS entry with UNAME=EXFAIL2.

6. The damage variable is determined by either FAILJC or FAILUDS entry with UNAME=EXFAIL2. In UNAME=EXYLD, the yield stress can be reduced depending on the magnitude of the damage variable. In that case the NOFAIL option should be set on the FAILJC or FAILUDS entry with UNAME=EXFAIL2.

## YLDVM

## von Mises Yield Model

Defines a bilinear or piecewise-linear yield model with isotropic hardening, using the von Mises yield criterion. Used in SOL 700 only.

Format:

| 1     | 2     | 3     | 4    | 5 | 6 | 7 | 8 | 9 | 10 |
|-------|-------|-------|------|---|---|---|---|---|----|
| YLDVM | YID   | YIELD | EH   |   |   |   |   |   |    |
|       | TABLE | TYPE  | TABY | D | P |   |   |   |    |

Example:

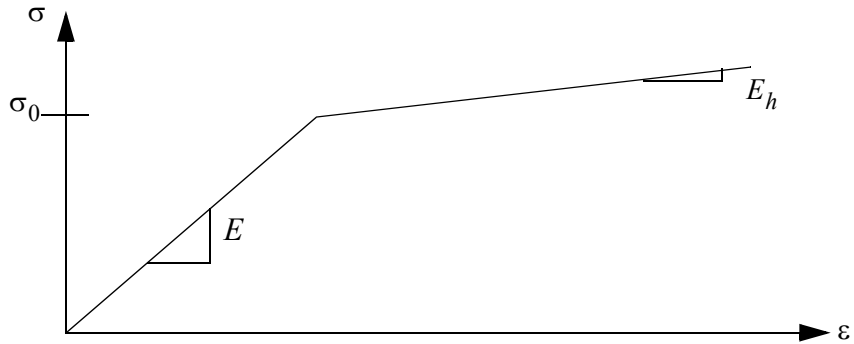
|       |    |        |         |  |  |  |  |  |  |
|-------|----|--------|---------|--|--|--|--|--|--|
| YLDVM | 32 | 250.E6 | 2000.E6 |  |  |  |  |  |  |
|       |    |        |         |  |  |  |  |  |  |

| Descriptor | Meaning                                                                                                                                                                                                                                                                    |
|------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| YID        | Unique yield-model number referenced from MATDEUL. (Integer > 0; Required)                                                                                                                                                                                                 |
| YIELD      | Yield stress. (Real; Required)                                                                                                                                                                                                                                             |
| EH         | Hardening modulus. (Real; Default = 0.0)                                                                                                                                                                                                                                   |
| TABLE      | Number of a TABLED1 entry giving the variation of effective stress (y-value) with effective strain (x-value). See Remark 4. (Integer > 0)                                                                                                                                  |
| TYPE       | The type of stress and strain defined in TABLED1. (Character; Default = TRUE)<br>ENG            Engineering stress and strain.<br>TRUE           True stress and strain.<br>PLAST         True stress and plastic strain.<br>PMOD         Plastic modulus and true stress. |
| TABY       | Number of TABLED1 entry giving the variation of the scale factor for the yield stress (y-value) with the strain rate (x-value). Strain-rate effects are also specified using the Cowper-Symonds relation (see input parameters D and P). See Remark 6. (Integer > 0)       |
| D          | Factor D in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real ≥ 0)                                                                                                                                                                                         |
| P          | Factor P in the Cowper-Symonds rate enhancement equation. See Remark 6. (Real ≥ 0)                                                                                                                                                                                         |

Remarks:

1. A bilinear stress-strain characteristic is used by specifying YIELD and EH:

where the yield stress  $\sigma_y$  is given by



$$\sigma_y = \sigma_0 + \frac{EE_h}{E - E_h} \varepsilon_p$$

where

|                 |                                               |
|-----------------|-----------------------------------------------|
| $\sigma_0$      | = yield stress specified in the YIELD field   |
| $E$             | = Young's modulus                             |
| $E_h$           | = hardening modulus specified in the EH field |
| $\varepsilon_p$ | = equivalent plastic strain                   |
| $\sigma_y$      | = yield stress                                |

2. A piecewise linear, stress-strain characteristic is used by specifying TABLE and TYPE (beams and shells only)

$$\sigma_{ij} = [(\sigma_i - \sigma_{i-1})(\varepsilon - \varepsilon_{i-1}) / (\varepsilon_i - \varepsilon_{i-1})] + \sigma_{i+1}$$

The stress-strain characteristic used internally in the solver is in terms of true stress and equivalent plastic strain. However, for convenience, the stress-strain characteristic can be input in any of the following ways:

True stress/true strain (TYPE = TRUE)

Engineering stress/engineering strain (TYPE = ENG)

True stress/plastic strain (TYPE = PLAST)

Plastic modulus/true stress (TYPE = PMOD)

3. With Lagrangian and Eulerian solid elements, only an elastic perfectly plastic yield model is currently used. Only the YIELD field is used.
4. If TABLE is blank or zero, a bilinear stress-strain curve is assumed. If TABLE has a value, it refers to a TABLED1 entry giving the stress-strain curve for the material.
5. If TABLE is defined, the value of YIELD is left blank, since it is determined from the stress-strain curve.
6. If TABY is blank or zero and  $D$  and  $P$  are blank or zero, the yield stress does not vary with strain rate. If TABY has a value, then it references a TABLED1 entry, which gives the variation of the scale factor applied to the yield stress with strain rate. ( $D$  and  $P$  must be blank or zero.)  
 If TABY is blank or zero and  $D$  and  $P$  are defined, the enhancement of the yield stress with strain rate is calculated as

$$\frac{\sigma_d}{\sigma_y} = 1 + \left( \frac{\dot{\epsilon}_p}{D} \right)^{1/p}$$

Where  $\sigma_d$  is the dynamic stress,  $\sigma_y$  is the static yield stress (YIELD), and  $\dot{\epsilon}_p$  is the equivalent plastic strain rate.

7. If TYPE is PLAST or PMOD, Young's modulus must be defined. If TYPE is ENG or TRUE and Young's modulus is defined it will override the value calculated from the stress-strain curve.
8. Note that for values exceeding the maximum x-value of either of the TABLED1 entries (see TABLE and TABY fields), linear extrapolation is used based upon the last two points specified in the TABLED1.
9. YID must unique among all YLDxx entries in one model.

## YLDZA

## Zerilli-Armstrong Yield Model

Defines the Zerilli-Armstrong yield model where the yield stress is a function of effective plastic strain, strain rate and temperature. Used in SOL 700 only.

Format:

| 1     | 2   | 3 | 4 | 5 | 6 | 7 | 8    | 9 | 10 |
|-------|-----|---|---|---|---|---|------|---|----|
| YLDZA | YID | A | B | n | C | m | EPS0 |   |    |
|       | D   |   |   |   |   |   |      |   |    |

Example:

|       |     |        |       |     |  |  |  |  |  |
|-------|-----|--------|-------|-----|--|--|--|--|--|
| YLDZA | 7   | 200.E6 | 50.E6 | 0.1 |  |  |  |  |  |
|       | 0.0 |        |       |     |  |  |  |  |  |

| Describer | Meaning                                                                            |
|-----------|------------------------------------------------------------------------------------|
| YID       | Unique yield model number referenced from a MATDEUL entry. (Integer > 0; Required) |
| A         | Static yield parameters. (Real > 0; Required)                                      |
| B         | Hardening parameters. (Real ≥ 0; Default = 0.0)                                    |
| n         | Hardening exponent. (Real; Default = 0.0)                                          |
| C         | Strain rate parameter. (Real; Default = 1.0)                                       |
| m         | Temperature exponent. (Real; Default = 1.0)                                        |
| EPS0      | Reference strain rate. (Real > 0; Default = 1.0)                                   |
| CP        | Specific heat. (Real > 0; Default = 1.E20)                                         |
| D         | Bcc parameter. See Remark 4. (Real; Default = blank)                               |

Remarks:

1. This material model can be used to model to model Fcc (iron and steels) and Bcc (aluminum and alloys) metals.
2. The yield stress is computed from  
for Fcc metals:

$$\sigma_y = (A + B\varepsilon_p^n) \varepsilon \left[ -mT + CT \ln \left( \frac{\dot{\varepsilon}}{\varepsilon_0} \right) \right] \text{ for Fcc metals}$$

$$\sigma_y = (A + B\varepsilon_p^n) + De \left[ -mT + C \ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right) \right] \text{ for Bcc metals}$$

where

|                                         |   |                          |
|-----------------------------------------|---|--------------------------|
| $\varepsilon_p$                         | = | effective plastic strain |
| $\dot{\varepsilon}$                     | = | effective strain rate    |
| $\dot{\varepsilon}_0$                   | = | reference strain rate    |
| $T$                                     | = | temperature              |
| $A, B, n, C, m,$ and $D$ are constants. |   |                          |

3. The reference strain rate are per unit time.
4. In case the Bcc parameter  $D$  is not supplied, it is assumed that a Fcc metal is defined.





# A

## Configuring the Runtime Environment

- [Specifying Parameters](#)
- [User-Defined Keywords](#)
- [Resolving Duplicate Parameter Specifications](#)
- [Customizing Command Initialization and Runtime Configuration Files](#)
- [Symbolic Substitution](#)

## Specifying Parameters

MSC Nastran execution is controlled by a variety of parameters, either keywords or special Nastran statements, both required and optional. The purpose of this section is to describe how and where these parameters may be specified, not to describe these parameters in detail. This is done in subsequent sections. The MSC Nastran parameters may be specified on the command line, in a command initialization (INI) file, in runtime configuration (RC) files and, for some parameters, from environment variables. The information from these sources is consolidated at execution time into a single set of values. Much of this information is passed to analysis processing in a **control file**, built using the templates ([Customizing the Templates, 53](#)). (The records in this control file are echoed to the .log file.) Examples of INI and RC files are given in the [User-Defined Keywords, 3355](#) and [Customizing Command Initialization and Runtime Configuration Files, 3363](#).

## Command Initialization and Runtime Configuration Files

Although the purposes of the INI and RC files are somewhat different, the format of each file is the same. All INI and RC files are processed twice, once (the "first" pass) to extract parameters (keywords and other information) that are to be used for all MSC Nastran jobs, and once (the "second" pass) to extract parameters specific to a particular job. This is accomplished by separating the INI and RC files into a series of "sections" identified by a "section header" and "subsections" within sections, identified by a subsection "header." There are two types of sections: "unconditional" and "conditional." Subsections are always "conditional."

- An unconditional section is one that starts with the name of the section enclosed in square brackets ("[" , "]" ). Section names may not contain any embedded blanks but may be separated from the square brackets by any number of blanks. As currently implemented, there are three valid unconditional names: "General", "Solver" and "Nastran". (These section names are case-insensitive.) In addition, there is an implicit "unnamed" section that consists of all parameters in the INI or RC file that appear before the first named section or subsection. There is no special meaning assigned to any of the unconditional sections. Their use is optional; the section names are intended to be used for descriptive purposes.
- A conditional section or subsection is one that starts with an expression in the form:

`<keyword><operator><value>`

enclosed in section header identification characters. For a conditional section, the section header identification characters are square brackets ("[" , "]" ), just as for unconditional sections. For a subsection, the section header identification characters are "less than" and "greater than" ("<" , ">") characters. Keywords and values may not contain any embedded blanks but may be separated from each other and from the enclosing section header identification characters (the square brackets or "less than"->greater than" characters) by any number of blanks. In the expression:

|                               |                                                                                                                                                        |
|-------------------------------|--------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>&lt;keyword&gt;</code>  | represents any valid internal keyword (see <a href="#">Keywords, 184</a> ) or user-defined keyword (see <a href="#">User-Defined Keywords, 3355</a> ). |
| <code>&lt;operator&gt;</code> | specifies the comparison to be performed between <code>&lt;keyword&gt;</code> and <code>&lt;value&gt;</code> as follows:                               |
| <code>=</code>                | equal (either string or numeric)                                                                                                                       |

|         |                                                                       |
|---------|-----------------------------------------------------------------------|
| !       | not equal (either string or numeric)                                  |
| !=      | not equal (either string or numeric)                                  |
| <       | numerically less than                                                 |
| <=      | numerically less than or equal                                        |
| >       | numerically greater than                                              |
| >=      | numerically greater than or equal                                     |
| <value> | specifies the appropriate keyword value to be used in the comparison. |

Keywords and values may be specified in any case.

Parameters in unconditional sections, but not in subsections (which are always conditional) within unconditional sections, are processed on the first pass through an INI or RC file. On the second pass, these parameters are ignored (they are not reprocessed). Parameters in conditional sections and subsections are ignored on the first pass. Parameters in conditional sections and subsections whose expressions evaluate to "true" are processed on the second pass through an INI or RC file, thus allowing conditional expressions to reference all of the valid keywords. Note that for subsections within conditional sections, both the conditional expression for the section *and* the conditional expression for the subsection must evaluate to "true" before parameters in the subsection are processed.

Parameter specifications in, either unconditional or conditional sections, may be continued, if necessary, by specifying a backslash (“\”) character as the last non-blank character of the line. Note for Windows users, if the parameter value itself ends with a backslash, the statement must have additional characters, such as a comment, after the value specification. For example, a specification such as:

```
sdir=e:\
```

will not work properly. Instead, write the statement as:

```
sdir=e:\ $ Specify the scratch directory
```

In addition to parameters, INI and RC files may contain “comment” records. There are two types of comment records: ignored and printed.

- Ignored comments are records that start with a semi-colon (“;”) or pound sign (“#”). These records are completely ignored. When running in Windows, there is a special form of ignored comments that may be specified in an INI file (but not in RC files). These are records that start with "REM", short for "REMARK". The test for "REM" is case-insensitive.
- Printed comments are records that start with the currency symbol (“\$”). These records are passed on as part of the analysis information but are otherwise ignored.

**Note:** Although sectioning within INI and RC files was first introduced in MSC Nastran 2004, valid INI and RC files from prior versions of MSC Nastran are fully compatible with this new format. Since sections were not supported in previous versions (except for INI files on Windows, which allowed unconditional sections), all parameters will be in the "unnamed" implicit section (or, on Windows, in named unconditional sections) and will be processed on the first pass through the file. No information will be extracted from these files on the second pass.

- Command Initialization (INI) File

This file is used to define keywords that are to be set whenever the nastran command is executed. Typical keywords in the unconditional sections include the installation base directory and the version of MSC Nastran. Conditional sections and subsections might include keywords such as "rcmd" and "rsdirectory" in sections that are conditional upon the value of the "node" keyword.

Default installation directories are:

LINUX: *install\_dir/prod\_ver/arch/nastran.ini*

Windows: *install\_dir\prod\_ver\arch\nastran.ini*

Starting with MSC Nastran 2011, there are two possible RC files that may be defined in each of the locations that are searched for RC files. The first name is a version independent name and the second name is a version dependent name, where the version number is indicated by *<vernum>* in the file name and the version number for MSC Nastran is 2022.1. The list below specifies the INI and RC files that MSC Nastran uses. Any or all of these files may be omitted. [Table 1-1](#) lists the keywords that are generally set in the unconditional sections of the command initialization file. [Table 1-2](#) lists the keywords that are generally set in RC files.

*prod\_ver* is the msc version (msc20221 for MSC Nastran 2022.1).

In addition, the default *install\_dir* of MSC Nastran 2022.1 is as follows:

Linux: /msc/MS\_C\_Nastran/2022.1

Windows: C:\Program Files\MSC.Software\MS\_C\_Nastran\2022.1.

- System RC Files

These files are used to define parameters that are applied to all MSC Nastran jobs using this installation structure. Many of the parameters that might be specified in the INI file could, alternatively, be specified in this file.

LINUX: *install\_dir/conf/nastranrc* and  
*install\_dir/conf/nast<vernum>rc*

Windows: *install\_dir\conf\nastran.rcf* and  
*install\_dir\conf\nast<vernum>.rcf*

- Architecture RC Files

This files are used to define parameters that are applied to MSC Nastran jobs using this architecture.

LINUX: *install\_dir/conf/arch/nastranrc* and  
*install\_dir/conf/arch/nast<vernum>rc*

Windows: *install\_dir\conf\arch\nastran.rcf* and  
*install\_dir\conf\arch\nast<vernum>.rcf*

- Node RC Files

These files are used to define parameters that are applied to MSC Nastran jobs running on this node. Alternatively, the parameters in this file could be specified in a conditional section in one of the previous files, using *nodename* as the value of the "s.hostname" keyword in the conditional expression.

LINUX: *install\_dir/conf/net/nodename/nastranrc* and  
*install\_dir/conf/net/nodename/nast<vernum>rc*

Windows: *install\_dir\conf\net\nodename\nastran.rcf* and  
*install\_dir\conf\net\nodename\nast<vernum>.rcf*

- User RC Files

These files are used to define parameters that are applied to MSC Nastran jobs run by an individual user.

LINUX: `$HOME/.nastranrc` and  
`$HOME/.nast<vernum>rc`

Windows: `%HOMEDRIVE%%HOMEPATH%\nastran.rcf` and  
`%HOMEDRIVE%%HOMEPATH%\nast<vernum>.rcf`

- Local RC Files

These files should be used to define parameters that are applied to MSC Nastran jobs that reside in the input data file's directory. This RC file is in the same directory as the input data file. If the "rcf" keyword ([page 215](#)) is used, this local file is ignored.

LINUX: `.nastranrc` and  
`.nast<vernum>rc`

Windows: `nastran.rcf` and  
`nast<vernum>.rcf`

Please note that the LINUX shorthand "~", to refer to your or another user's home directory, cannot be used in an RC file. In addition, environment variables are only recognized within the context of a logical symbol definition.

Also, note that, on LINUX systems, the leading period (“.”) on the User RC Files and Local RC Files file names cannot be deleted even if alternate names are specified using the “a.urc” and “a.urcb” keywords as described below.

The file names listed above may be changed by the user using the “a.rc”, “a.rcb”, “a.urc” and “a.urcb” keywords, noting that the directories in which the files are located may not be changed.

- The “a.rc” keyword can be used to change the names of the version dependent RC file names for the System RC Files, the Architecture RC Files and the Note RC File. The default for this keyword is “nast<vernum>rc” for LINUX and “nast<vernum>.rcf” for Windows.
- The “a.rcb” keyword can be used to change the names of the version-independent RC file names for the System RC Files, the Architecture RC Files and the Node RC Files. The default for this keyword is “nastranrc” for LINUX and “nastran.rcf” for Windows.
- The “a.urc” keyword can be used to change the names of the version dependent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rc” keyword with a leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rc” keyword.
- The “a.urcb” keyword can be used to change the names of the version-independent RC file names for the User RC Files and the Local RC Files. For LINUX, the default for this keyword is the value of the “a.rcb” keyword with the leading period (“.”) added. For Windows, the default for this keyword is the value of the “a.rcb” keyword.

In addition to keyword specifications, the following MSC Nastran statements (from the NASTRAN and FMS Sections) may appear in RC files and conditional sections in an INI file: NASTRAN, ACQUIRE, ASSIGN, CONNECT, DBCLEAN, DBDICT, DBFIX, DBLOAD, DBLOCATE, DBSETDEL, DBUNLOAD, DBUPDATE, DEFINE, ECHOOFF, ECHOON, ENDJOB, EXPAND, INCLUDE, INIT, PROJ, RESTART and RFINCLUDE. Except for minimal checking of the NASTRAN and PARAM statements, the syntax of these statements is not validated. These records are simply passed on for use in MSC Nastran analysis processing.

INI files and RC files also may contain PARAM statements that specify values that affect MSC Nastran analysis processing. The values associated with PARAM names may be specified using PARAM statements in INI files and RC files or by using PARAM keywords, defined using the PARAM keywords feature as described in [User-Defined Keywords, 3355](#). PARAM statements must be specified in "free-field format", i.e., in the Case Control PARAM format (PARAM,name,value), not in Bulk Data fixed-field format. Please see [Parameters](#) (Ch. 6) in the *MSC Nastran Quick Reference Guide* for more information on PARAM names and statements and their usage.

## Environment Variables

Several keywords may have their values set from associated environment variables. When this is the case, the environment variable takes precedence over any INI or RC file keyword specification. A command-line specification will over-ride the environment variable specified value. This same precedence rule applies to user-defined keywords that may have their initial values taken from environment variables, as described in the next section. A list of the keywords and their associated environment variables, along with a description of each keyword, may be obtained by using the following command:

`nast_ver help env`

## User-Defined Keywords

In addition to the internally defined keywords (see [Keywords, 184](#)), MSC Nastran allows users to define their own keywords. There are two classes of user-defined keywords:

- General keywords. These are intended for use in INI file or RC file conditional section clauses, in user modifications to the run template files (nastran.dmp, nastran.lcl, nastran.rmt or nastran.srv) and, for LINUX, in customized queue commands (“submit” keyword).
- PARAM keywords. These are keywords associated with a PARAM name. Using descriptive keywords to set a PARAM value may be more convenient than specifying the PARAM statement in an RC file. Also, keywords are not limited to a maximum of eight characters, as PARAM names are, and may be more descriptive of the action being affected or requested.

User-defined keywords are supported by the “help” and “whence” functions.

## General Keywords

These keywords are defined in the file specified by the "0.kwds" keyword. While the file is not delivered, an administrator could create the file. The default file names are:

```
LINUX:          install_dir/prod_ver/arch/nastran.kwds
Windows:       install_dir\prod_ver\arch\nastran.kwds or
               install_dir\bin\nast20221.kwds
               The file used is the first one found.
```

The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';', or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword records. These records consist of the keyword name along with an optional value descriptor and comment in the form:

```
keyword_name[,attributes] : value_descriptor comment
```

where:

|                               |                                                                                                                                                                                                                                                                                                                                                                                     |
|-------------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>keyword_name</code>     | is the name to be assigned to the user keyword. This name may not contain any embedded blanks and may not be the same as any internal keyword or previously specified user-defined keyword. It is also case-insensitive except in the case when its initial value may be set from an environment variable with the same name.                                                       |
| <code>attributes</code>       | specifies optional attributes to be assigned to the keyword defined by <code>keyword_name</code> . Currently, the only defined attribute is:<br><code>argv keyword</code> and its value is to be added to the "r.argv" keyword value<br>Any number of blanks may separate <code>keyword_name</code> , the separating command and the attributes specification.                      |
| <code>value_descriptor</code> | is optional. If specified, it should be as described in <a href="#">Value Descriptors, 3357</a> and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted.. The default value descriptor is "string". This field may also specify that the initial value of this keyword be taken from an environment variable with the same name. |
| <code>comment</code>          | is an optional comment field. If present, it must be separated from <code>value_descriptor</code> or <code>keyword_name</code> by blanks or must begin with a comment character.                                                                                                                                                                                                    |

There may be any number of leading blanks in the record and before and after the separating colon.

General keywords and the values assigned to them only affect MSC Nastran processing if:

- there are customized INI and RC files that have conditional sections, using these keywords in expressions, that specify other keywords and statements (e.g., NASTRAN and PARAM statements) that modify MSC Nastran processing to meet the requirements of a user's site and installation.
- they are used in customized templates ([Customizing the Templates, 53](#)).
- for LINUX systems, they are used in customized queue commands defined using the "submit" keyword ([Customizing Queue Commands \(LINUX\), 51](#)).

## PARAM Keywords

These keywords are defined in the file specified by the "0.params" keyword. The default file names are:

LINUX:            *install\_dir***|***prod\_ver**|arch*/nastran.params

Windows:        *install\_dir***\***prod\_ver**\arch*\nastran.params



The records in this file consist of:

- Comment records. These are records that start with a comment character (hash, '#', semi-colon, ';', or currency symbol, '\$') and are completely ignored.
- Blank or null records. These records are ignored.
- Keyword-name records. These records consist of the keyword name, the associated PARAM name, along with an optional value descriptor and comment in the form:

keyword\_name : param\_name : value\_descriptor comment

where:

|                               |                                                                                                                                                                                                                                                                                                |
|-------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <code>keyword_name</code>     | is the name to be assigned to the PARAM keyword. This name is case-insensitive, may not contain any embedded blanks and may not be the same as any internal keyword, general user-defined keyword or previously specified PARAM keyword.                                                       |
| <code>param_name</code>       | is the PARAM name to be associated with <code>keyword_name</code> . This name is case-insensitive, may be a maximum of eight characters, must begin with an alphabetic character and may not contain any embedded blanks. Also, it may not be the same as any previously specified PARAM name. |
| <code>value_descriptor</code> | is optional. If specified, it should be as described in <a href="#">Value Descriptors</a> and may not contain any embedded blanks. If this field is not present, the separating colon may be omitted. The default value descriptor is "string".                                                |
| <code>comment</code>          | is an optional comment field. If present, it must be separated from <code>value_descriptor</code> or <code>param_name</code> by blanks or must begin with a comment character.                                                                                                                 |

There may be any number of leading blanks in the record and before and after the separating colons.

Keyword names that are the same as PARAM names are allowed, as long as the keyword name is not an internal or general user-defined keyword name.

Values associated with PARAM names, whether set using PARAM keywords or set using PARAM statements (statements having the form `PARAM,name,value`), directly affect MSC Nastran analysis processing.

## Value Descriptors

Value descriptors enable limited syntax checking for values assigned to general and PARAM user-defined keywords. For general keywords, they may also specify that the initial value of the keyword be set from the value associated with the environment variable having the same name as the keyword. There are two types of syntax checking available: value must be one of a list of entries or value must be numeric. Also, the two forms can be combined. These are specified as follows:

List: { "val1", "val2", ..., "valn" }

That is, the acceptable values are enclosed in double quotes (") and separated from each other by commas. The specification, including the various acceptable values, may not contain any embedded blanks. Values are case-insensitive and any partial specification is acceptable and will be replaced by the full value. For example, if a keyword may only have the values "preliminary", "check" and "final", the value descriptor would be:

```
{"Preliminary", "Check", "final"}
```

and a value specification of "Ch" would be accepted and replaced by "check".

Numeric: `number`

Values will be checked to see if they are valid numbers, either integer or floating point. For example, valid keyword value specifications could be: "1", "-3.247", "4.e-5", "3.75-4", "4.24x" and "-4-5" are invalid specifications.

**Note:** This checking does *not* support the NASTRAN "nnnsee" numeric format, where the 'e' between the number and the signed exponent ("see") is missing.

Complex value: `number, number`

This format is only supported for PARAM keyword value descriptors. Values will be checked to see if they consist of two valid numeric values, separated by a comma.

Combined: `{"val1", "val2", ..., "valn", number}`

**Note:** This "combined" format does not support complex numbers.

In addition, for general keywords, if the value descriptor starts or ends with the string "env", specified in any case and separated from the rest of the value descriptor with a comma (unless the value descriptor is only "env"), the keyword value will be set using the value associated with the environment variable having the same name as the keyword. The environment value will be subjected to the same syntax-checking rules that an INI file, RC file or command line specification would be, with a warning message generated if syntax checking fails. This occurs even if the keyword is specified on the command line. Note that, for LINUX systems, since environment variable names are case-sensitive, the keyword name must be specified exactly the same as the environment variable name. This is the only time that the keyword name is case-sensitive. For Windows systems, since environment variable names are not case-sensitive, this restriction does not apply. Keyword values set from environment variables over-ride keyword values set in INI or RC files but do not over-ride keyword values set on the command line.

If a value descriptor is omitted or is not one of these formats, no syntax checking will be performed.

## Examples:

1. The following value descriptor would accept a value of "test", "final" or a number:

```
{"Test", "Final", Number}
```

Acceptable values would be: `te` (replaced by `test`), `FIN` (replaced by `final`), `7`, `14.5`, `3.e-4`, `-5`

2. The following value descriptor would accept only the strings "abc", "def", "ghi" and "glm":

```
{"abc", "def", "ghi", "glm"}
```

Acceptable values would be: `g` (replaced by `ghi`), `aB` (replaced by `abc`), `g1` (replaced by `glm`), `D` (replaced by `def`)

3. The following value descriptor, only valid for a PARAM keyword, would only accept a complex number specification:

```
number, number
```

Acceptable values would be: `1`, `2`, `7.54`, `3.14`

4. The following value descriptors, only valid for a general keyword, would accept only the strings "qrs", "test", and "xyz". In addition, the value descriptor requests that the keyword value be set from the environment.

```
enV, {"qrs", "test", "xyz"}
```

or

```
{"qrs", "test", "xyz"}, Env
```

Acceptable values would be: `q` (replaced by `qrs`), `xY` (replaced by `xyz`), `T` (replaced by `test`)

## Resolving Duplicate Parameter Specifications

MSC Nastran processing information is obtained by scanning the various INI and RC files, the system environment, and the Nastran command line in the following order:

1. Nastran command line, first pass. Only "program options", i.e., "-x" options, are processed during this command line scan. For example, this is when the "-i *ini\_file\_name*" program option is processed.
2. Environment variables, first pass. During this pass, the only keywords whose values are set are those that may only be specified as environment variables. This includes keywords such as HOME (for LINUX), HOMEDRIVE and HOMEPATH (for WINDOWS) and PWD.
3. INI file, first pass, if this file exists. During this pass, only unconditional sections are processed. Generally, the only keywords processed in this pass are: 0.kwds, 0.params, accmd, acvalid, rcmd, rsdirectory, sysmsg and version (although rcmd and rsdirectory probably should be in conditional sections scanned during the second pass).
4. Environment variables, second pass. During this pass, only those keywords that may only be set in global sections of the INI file or as environment variables are processed. This includes keywords such as MSC\_ARCH, MSC\_BASE and MSC\_VERSD.
5. Nastran command line, second pass. The only general use keywords processed during this command line scan are: dmparallel, jid, jidpath, jidtype, node, pause, rcf, username, version and whence. The processing of other command line keywords is deferred until later command line scans.

This is the time that the user-defined keyword definition files (for both general use and PARAM keywords), if any, are processed and the keyword specifications defined by these files are added to the keywords tables. The keywords defined in these files may be used just as internal keywords are used. (See [User-Defined Keywords, 3355](#).)

6. System RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
7. Architecture RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
8. Node RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
9. User RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
10. Local RC files, first pass, if these files exist. During this pass, only unconditional sections are processed.
11. Environment variables, third pass. During this pass, only "general" user-defined keywords that have been flagged to be set from environment variables are processed. (This pass will be skipped if there are no "general" user-defined keywords.)
12. Nastran command line, third pass. Only "general" user-defined keywords are processed during this command line scan. (This pass will be skipped if there are no "general" user-defined keywords.)

At this point, all keyword values that can be used in conditional section expressions are known.

13. INI file, second pass, if this file exists and has conditional sections. During this pass, only the conditional sections are processed.
14. System RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
15. Architecture RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
16. Node RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
17. User RC files, second pass, if these files exist and have conditional sections. During this pass, only the conditional sections are processed.
18. Local RC files, second pass, if these files exist and have conditional sections and if they are not ignored. During this pass, only the conditional sections are processed.
19. Environment variables, fourth pass. During this pass, all keywords that may be set from environment variables and that have not been processed previously are now processed.
20. Nastran command line, fourth pass. All keywords not processed during the previous passes are now processed. For example, this is when user-defined PARAM keyword specifications are processed.  
At this point, all information necessary to generate the "control file" has been collected. This file is generated when the "script templates" (see [Customizing the Templates, 53](#)) are processed.
21. NASTRAN, FMS and PARAM statements in the input file.

If duplicate keywords are encountered, the *last* specification found is the one used. That is, the above list specifies the precedence order, from lowest precedence (number 1) to highest (number 21). The only case in which the last keyword specification is not used is when keywords are "locked", i.e., when a specification of the form

```
lock=keyword
```

is processed. After this "lock" request is processed, any requests to set *keyword*, whether from INI files, RC files, environment variables or command line arguments, are quietly ignored. That is, processing proceeds as if any *keyword* specifications specified after the "lock=*keyword*" request do not exist. Once a keyword has been "locked," there is no way to "unlock" it. (Note that it is valid to "lock" the `lock` keyword itself.)

If duplicate NASTRAN and FMS statements are encountered, they are simply passed on for use in MSC Nastran analysis processing in the order in which they were encountered.

Thus, the general rule for resolution is:

- Information specified in NASTRAN input data files always takes precedence over any other values.
- Command line parameters have the next highest precedence.
- Environment variables associated with keywords and that have non-null values are next.
- RC file parameter specifications are next.
- INI file parameter specifications are last.

Generally, the only exceptions to this precedence ordering are "general" user-defined keyword specifications. The command line values take precedence over values specified in unconditional INI file and RC file sections but have lower precedence than values specified in conditional INI file and RC file sections. Because the primary purpose for general user-defined keywords is for conditional section selection, changing a general user-defined keyword in a conditional section *may* lead to unexpected results. Such specifications should be used with care. Also, because user-defined PARAM keywords on the command line are not processed until the last command line scan, PARAM keywords should not be used in INI file and RC file conditional section expressions since command line specified values will not be in effect when these expressions are evaluated.

Because PARAM values may be specified either using PARAM statements or using PARAM keywords, they require further explanation. PARAM statements and PARAM keywords referring to the same PARAM name are considered equivalent definitions for the PARAM name. As such, the last specification, regardless of whether it was a PARAM statement or a PARAM keyword, is the one that is used to establish the value associated with the PARAM name.

## Customizing Command Initialization and Runtime Configuration Files

Table 1-1 lists the keywords that are generally set in the unconditional sections of the command initialization file.

Table 1-1      Command Initialization File Keywords

| Keyword         | Purpose                                                                                   |
|-----------------|-------------------------------------------------------------------------------------------|
| <b>0.kwds</b>   | Alternate name for user-defined keywords definition file.                                 |
| <b>0.params</b> | Alternate name for PARAM keywords definition file                                         |
| <b>acct</b>     | Enables job accounting, see <a href="#">Enabling Account ID and Accounting Data, 40</a> . |
| <b>acvalid</b>  | Activates account ID validation, see <a href="#">Enabling Account ID Validation, 40</a> . |
| <b>MSC_BASE</b> | Defines the installation base directory.                                                  |
| <b>version</b>  | Specifies the default version of MSC Nastran to be run.                                   |

Most of the command line keywords can be set in any of the RC files. Table 1-2 lists keywords that are generally set in the system, architecture, or node RC files:

Table 1-2      RC File Keywords

| Keyword              | Preferred RC File | Purpose                                                                                                                   |
|----------------------|-------------------|---------------------------------------------------------------------------------------------------------------------------|
| <b>accmd</b>         | System            | Command line to invoke accounting logger program.                                                                         |
| <b>acct</b>          | System            | Enables job accounting.                                                                                                   |
| <b>acvalid</b>       | System            | Enables account ID (acid) validation.                                                                                     |
| <b>authorize</b>     | System            | Specifies the licensing method.                                                                                           |
| <b>buffsize</b>      | System            | Set the default buffsize. Suggested values are in <a href="#">Table 4-5</a> .                                             |
| <b>lock</b>          | Any               | Prevent further changes to a keyword's value.                                                                             |
| <b>memory</b>        | Node              | Specifies a default memory allocation                                                                                     |
| <b>memorymaximum</b> | Node              | Specifies a maximum "memory" request. May be specified as a percentage of RAM e.g.<br><code>memorymax=0.5xPhysical</code> |
| <b>ncmd</b>          | Architecture      | Specifies the notify command when "notify=yes" is set.                                                                    |
| <b>news</b>          | System            | Controls the display of the news file at the beginning of the .f06 file.                                                  |

Table 1-2 RC File Keywords (continued)

| Keyword            | Preferred RC File | Purpose                                                                                                                                                  |
|--------------------|-------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>post</b>        | Architecture      | LINUX: Specifies commands to be run after each job is completed.                                                                                         |
| <b>ppcdelta</b>    | Architecture      | LINUX: Specifies the value that is subtracted from the "CPU" keyword value to determine the NQS per-process CPU time limit.                              |
| <b>ppmdelta</b>    | Architecture      | LINUX: Specifies the value that is added to the "memory" keyword value to determine the NQS per-process memory limit.                                    |
| <b>pre</b>         | Architecture      | LINUX: Specifies commands to be run before each job begins.                                                                                              |
| <b>prmdelta</b>    | Architecture      | LINUX: Specifies the value that is added to the "ppm" value to determine the NQS per-request (per-job) memory limit.                                     |
| <b>qoption</b>     | Architecture      | LINUX: Specifies a string of additional queuing options to be set in the queue submittal command.                                                        |
| <b>rcmd</b>        | Any               | Specifies the remote Nastran command to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression. |
| <b>real</b>        | Node              | Specifies the "REAL" parameter to limit virtual memory usage.                                                                                            |
| <b>rsdirectory</b> | Any               | Specifies the scratch directory to be used when "node" is specified. Should be in a conditional section using "node" in the conditional expression.      |
| <b>scratch</b>     | Any               | Specifies the default job status as scratch or permanent.                                                                                                |
| <b>sdirectory</b>  | Node              | Specifies a default scratch directory.                                                                                                                   |
| <b>submit</b>      | Architecture      | LINUX: Defines queues and their associated submittal commands.                                                                                           |
| <b>sys<i>n</i></b> | Any               | Specifies system cells. Can also be specified using the synonym keywords, e.g., buffsize is equivalent to sys1.                                          |



## Examples

The following (relatively simplistic) examples illustrate how unconditional and conditional sections could be used.

### Example 1:

Assumptions: There are three computer nodes, sysnode1, sysnode2 and sysnode3, that may be accessed.

On sysnode1:

- MSC Nastran 2014 and MSC Nastran 2022.1 are installed:
  - MSC Nastran 2014 is accessed using "/local/msc/bin/nast2014"
  - MSC Nastran 2022.1 is accessed using "/local/msc/bin/nast20221"
  - The scratch directory is /local/temp

On sysnode2:

- Only MSC Nastran 2014 is installed and is accessed using "/local1/msc/bin/nast2014"
- The scratch directory is /local1/temp

On sysnode3:

- MSC Nastran 2014 and MSC Nastran 2022.1 are installed:
  - MSC Nastran 2014 is accessed using "/local2/msc/bin/nast2014"
  - MSC Nastran 2022.1 is accessed using "/local2/msc/bin/nast20221"
- The scratch directory is /local2/temp

All of this information could be specified in an INI file, identical on all three nodes, as follows:

```
;
; This is the MSC Nastran Command Initialization File
; The default version is to be set to 2022.1
;
version=2022.1
; Define conditional sections giving the appropriate sdir
; values when MSC Nastran is run locally.

[ s.hostname = sysnode1 ]
sdir=/local/temp
[ s.hostname = sysnode2 ]
sdir=/local1/temp
[ s.hostname = sysnode3 ]
sdir=/local2/temp

; Define conditional sections giving the appropriate
; remote access keywords when a "node" value,
; requesting remote execution, is specified.
;
[ node = sysnode1 ]
rsdir=/local/temp
< version = 2014.0 >
rcmd=/local/msc/bin/nast2014
```

```

< version = 2022.1 >
rcmd=/local/msc/bin/nast20221

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast20140

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2022.1 >
rcmd=/local2/msc/bin/nast20221

;
; This is the end of the Command Initialization file
;

```

Alternatively, the information could be split between an INI file and a system RC file, identical on all three nodes, as follows:

In the INI file:

```

;
; This is the MSC Nastran Command Initialization File
; The default version is to be set to 2022.1
;
version=2022.1

; Define conditional sections giving the appropriate
; remote access keywords when a "node" value,
; requesting remote execution, is specified.
;
[ node = sysnode1 ]
rsdir=/local/temp
< version = 2014.0 >
rcmd=/local/msc/bin/nast2014
< version = 2022.1 >
rcmd=/local/msc/bin/nast20221

[ node = sysnode2 ]
rsdir=/local1/temp
< version = 2014.0 >
rcmd=/local1/msc/bin/nast20140

[ node = sysnode3 ]
rsdir=/local2/temp
< version = 2014.0 >
rcmd = /local2/msc/bin/nast2014
< version = 2022.1 >
rcmd=/local2/msc/bin/nast20221

;
; This is the end of the Command Initialization file;

```

In the system RC file, identical on all three nodes:

```

;
; This is the MSC Nastran system RC file.
;

```



```

; Define conditional sections giving the appropriate sdir
; values when MSC Nastran is run locally.

[ s.hostname = sysnode1 ]
sdir=/local/temp
[ s.hostname = sysnode2 ]
sdir=/local1/temp
[ s.hostname = sysnode3 ]
sdir=/local2/temp

;
; This is the end of the system RC file
;

```

### Example 2:

Assumptions: User keywords defining "run type" and "data complexity" are needed and AUTOSPC, AUTOSPCR, BAILOUT and ERROR PARAM values are to be set based on these keywords.

The nastran.kwds file could be:

```

; User Keywords
Runtype:{"prelim","development","final"};Analysis stage
      Level :      number      # Data complexity level
;

```

The nastran.params file could be:

```

; PARAM keywords

Set_AutoSPC : AutoSPC      : {"Yes","No"}
Set_AutoSP_CR : AUTOSPCR : {"yes","no"}
Bai_lout_Val_ue : bailout : number
Set_Err_ör : Error : number
;

```

Then, the system RC file could contain:

```

; RC file
[ runtype = prelim ]
set_autospc = yes
bailout_value = -1
set_error = 0
set_autosp_cr = yes

[ runtype = development ]
set_autospc=yes
bailout_value=0
set_error=-1

[runtype=final]
set_autospc=no
param,bailout,0
param,error,-1
param,autospcr,no

[level < 3]
; basic data complexity parameters
[level >= 3]
<level>8>

```

```
; advanced data complexity parameters  
<level<=8>  
; intermediate data complexity parameters  
; End of RC file
```

# Symbolic Substitution

## Introduction

Symbolic Substitution is a capability added to MSC Nastran that allows a user to effectively modify a Nastran data file using command line and RC file keyword specifications without actually editing the file. This capability is very similar to “environment variable” expansion that happens in various command prompt shells such as the Linux Bourne, Korn and C shells and the Windows Command Prompt shell when scripts are processed. It is also analogous in some ways to the capabilities provided by programming language preprocessors, for example, the CPP preprocessor used by the various C/C++ compilers. The key feature of symbolic substitution is that these modifications do *not* affect the actual data file but present the data read from the data file to the processing program as if it was the modified data that was being processed.

Generally, symbolic substitution means that a data record is scanned to see if it contains special data strings (that identify the “symbolic” variables) that specify “symbolic substitution” requests. If such strings are found, the record is modified to replace the special data strings with user-defined substitution (replacement) strings (the values currently associated with the “symbolic” variables, i.e., the variable “values”) and it is this modified record that is actually processed. This symbolic substitution happens before any other processing of the record occurs, thus making it transparent to the rest of the program processing the data record. In the case of MSC Nastran, this symbolic substitution processing will happen immediately after a record is read from the Nastran data file and before any other processing (with the possible exception of special processing required to satisfy licensing requirements) is performed.

## Simple Examples

Two very simple examples illustrate how this capability could be used in Nastran data files. Note that the details of the syntax are completely described in the following sections and may be ignored for now. Also note that the examples do not deal with things such as managing the output from multiple Nastran runs. These issues, involving, among other techniques, using command line or RC file keywords such as "out=", "append=" and "old=yes", are beyond the scope of this document.

### Example 1:

Suppose you want to make several tests where the thickness of a PSHELL element is to be varied. You could do this by defining the thickness of the PSHELL element as a "symbolic variable" (identified using the string "%thickness%"), setting a default value (using the "%defrepsym" statement) and specifying the desired thickness on the command line (using the "REPSYM=" keyword). A very simple data file (sym.dat) could be (where most of the BULK entries are in an include file named "model.bdf", not shown here):

```
%defrepsym thickness=5.0
SOL 103
CEND
TITLE = 1st perturbation, t = %thickness%
ECHO = NONE
SUBCASE 1
    METHOD = 100
    SPC = 1
```

```

        DISP = ALL
    BEGIN BULK
    EIGRL,100,,,6
    PARAM,POST,0
    PARAM,GRDPNT,0
    $PBEAML Properties
    PBEAML 2      1      I
           70.0    60.0    60.0    3.3    5.    5.
    $
    $PSHELL Properties
    $
    pshell,1,1,%thickness%,1,,1
    $
    include 'model.bdf'
    enddata

```

If the test is run using the following command line:

```
nast20221 sym repsym=thickness=1.0 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 1.0
```

and

```
pshell,1,1,1.0,1,,1
```

If the test is run using the following command line:

```
nast20221 sym repsym=thickness=3.5 ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 3.5
```

and

```
pshell,1,1,3.5,1,,1
```

If the test is run without specifying any REPSYM setting for "thickness", e.g., using the following command line:

```
nast20221 sym ...
```

the test will run as if the "TITLE" and "pshell" records are:

```
TITLE = 1st perturbation, t = 5.0
```

and

```
pshell,1,1,5.0,1,,1
```

## Example 2:

Suppose you have a test that contains two superelements, where the only difference between the data for each superelement is the area of a PBAR element. Instead of having two different definitions, you could have a single definition of the data in an include file, where the area of the PBAR is specified as a symbolic variable. The include file (called "bar.bdf") could be:

```

%defrepsym area=1.
grid,2,,1.0,0.0,0.0
grid,3,,2.0,0.0,0.0

```

```

grid,4,,3.0,0.0,0.0,,123456
cbar,2,2,2,3,0.,1.,0.
cbar,3,2,3,4,0.,1.,0.
pbar,2,2,%area%,1.,1.,1.
mat1,2,1.e7,,.3

```

and the actual input file could be:

```

sol 101
cend
title=simple part se
echo=both
subcase 1
load=1
disp=all
elforce=all
begin bulk
grid,1,,0.0,0.0,0.0
grid,2,,1.0,0.0,0.0
cbar,1,1,1,2,0.,1.,0.
pbar,1,1,1.,1.,1.,1.
mat1,1,1.e7,,.3
force,1,1,,1.,1.,1.,1.
$
begin super=1
%setrepsym area=1.
include 'bar.bdf'
$
begin super=2
%setrepsym area=2.
include 'bar.bdf'
enddata

```

The first "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,1.,1.,1.,1.
```

and the second "include 'bar.bdf'" statement will be processed as if the pbar record is

```
pbar,2,2,2.,1.,1.,1.
```

## Detailed Specifications

The use of the Symbolic Substitution capability is defined by a number of “rules”. These “rules” are given in the following sections and provide the complete specification. Following the rules, there is information about requesting report information and about error handling. Finally, there are some (again simple) examples showing usage.

### Symbolic Substitution Rules

The following rules define the symbolic substitution user interface. The descriptions start with the rules for variable naming, followed by the rules for defining the replacement width information, followed by the various keywords and statements used to control symbolic substitution.

### Variable Naming

The rules for naming symbolic substitution variables are:

- Symbolic variable names are not case-sensitive, are a maximum of 32 characters long and may not contain leading, trailing or embedded blanks or special characters including (“\_”). Variable names must start with an alphabetic character followed by zero or more alphabetic or numeric characters. For example:
  - The variable name "VaRiaBLe1" is the same as "VARIABLE1" and "variable1"
  - The following variable names are valid:
    - abcdef
    - abc123
    - Name1
  - The following variable names are not valid:
    - 123abc      Does not start with an alphabetic character
    - a bcd        Contains an embedded blank
    - abc&        Contains an invalid character ('&')
    - /def         Does not start with an alphabetic character
    - \_abc123     Uses an underscore in the name.
- Unless symbolic variable values are quoted, they are not case-sensitive and may not contain leading, trailing or embedded blanks or percent ("%") characters. The quoting rules are given below.

### Substitution Field Width Specification

The ability to control the appearance of any symbolic substitution is an important requirement when generating data for a program such as MSC Nastran. The result of a symbolic substitution request is identified as a *field*. Substitution field width information can be taken by default, specified in the data file or specified using command line and/or RC file keywords. These methods are explained below.

The rules for defining substitution field width information are:

- Symbolic variable substitution is, by default, *exact*. That is, the number of characters occupied by the symbolic symbol replacement is exactly the same as the replacement value. However, this default replacement processing can be controlled by specifying the substituted field *width*, the field *precision* and the *justification* within the field. This information is specified using the syntax

`-w.p`

where the ‘-’, ‘w’ and ‘p’ are all optional and have the following meanings.

- The field width specification (w) defines the *minimum* number of characters the field is to have as a decimal integer value. If the replacement value has fewer characters than the field width, it will be padded with spaces on the left (by default) or on the right (if the left justification flag is specified). If the replacement value has more characters than the field width and if no precision



value was specified, the entire replacement string will be used. A field width value of 0 (zero) is equivalent to omitting the width specification. Note that a negative width value will be processed as if the “left-justification” flag was specified (see below) since a negative field width is meaningless.

- The field precision specification (*p*) defines the *maximum* number of characters the field is to have. The format is a period (.) followed by a decimal integer value. If the replacement value length exceeds the precision value, only the last *p* (by default) or the first *p* (if the left justification flag is set) characters of the replacement value will be used. A field precision value of 0 (zero) (or a negative value) is equivalent to omitting the precision specification.
  - If both field width and field precision are specified and are positive, the precision value cannot be less than the width value. If it is, it will be reset to the field width.
  - The ‘-’ character is the “left-justification” flag and specifies that the replacement value is to be left-justified within the field. If this character is omitted, the replacement value will be right-justified within the field.
- For example, the width, precision and justification of a typical field in the Bulk Data portion of a Nastran data file is:
 

```
-8.8
```

 meaning that the field is exactly eight characters wide and that data is to be left-justified within the field. For a wide-format Bulk Data record, this specification would be:
 

```
-16.16
```

 The specification for an exact replacement, i.e., where the replaced field is exactly the size of the replacement value, is:
 

```
0.0
```
  - To simplify width specification for Nastran widths, the following (case-insensitive) synonyms for common widths are available and may be used wherever a width specification can be used:

|                    |                                      |
|--------------------|--------------------------------------|
| <code>exact</code> | is equivalent to <code>0.0</code>    |
| <code>bulk</code>  | is equivalent to <code>-8.8</code>   |
| <code>wide</code>  | is equivalent to <code>-16.16</code> |

It is very important to note that there are two distinct portions to a Nastran data file, that part that is before the first BEGIN statement and that has “free format”, and that part that is after the first BEGIN statement (the Bulk Data Section) and often has fixed format fields. Because of this, two different sets of field width information are maintained for use when field width information is not explicitly specified as part of a symbolic substitution request, one for use before the first BEGIN statement and one for use after the first BEGIN statement.

### Defining Variable Values and Width Information

Symbol names and associated values and symbol width specifications may be set using keywords on the command line or in RC files and may be set using special statements in the Nastran data file itself. Each keyword and statement is explained in detail.

### *Using Command Line or RC File Keywords*

#### **Setting Variable Value Using REPSYM**

Symbolic variables and associated values may be set on the Nastran command line or in RC files using the keyword

```
repsym=<varname>=<varvalue>
```

where <varname> specifies the name of the symbolic variable and <varvalue> specifies the value to be associated with the variable name. For example,

```
repsym=abc=1.23e-5
```

#### **Setting Variable Width Information Using REPWIDTH**

Symbolic variable substitution default width information may be set on the Nastran command line or in RC files using the keyword

```
repwidth=<widthinfo1>,<widthinfo2>
```

where <widthinfo1> specifies the default width information for the portion of the Nastran data file before the BEGIN statement and <widthinfo2> specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a `-w.p` specification or as one of the synonyms, as described previously. If either width specification is omitted, the current default for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if <widthinfo2> is specified. For example,

```
repwidth=12,bulk
```

specifies that symbolic substitution default width is to be 12.0 before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered and

```
repwidth=,bulk
```

specifies that symbolic substitution default width is to be EXACT (or 0.0, the default) before the BEGIN statement is encountered and -8.8 after the BEGIN statement is encountered.

Just as with other Nastran command line or RC file keywords, the REPSYM and REPWIDTH keywords are not case-sensitive.

### *Using Special Statements in a Nastran Data File*

#### **Setting Values Using setrepsym**

Symbolic variables and associated values may be set in a Nastran data file using the following statement:

```
%setrepsym <varname>=<varvalue>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following <varvalue>, where the start of the comment is indicated by a '\$' (blank, currency symbol). The setrepsym string is not case-sensitive and at least one blank must separate this string from the <varname> specification. For example,

```
%setrepsym abc=1.23e-5
```

#### **Clearing ("Unsetting") Values Using unsetrepsym**

A symbolic variable value set using the `%setrepsym` statement may be cleared ("unset") in a Nastran data file using the following statement:

```
%unsetrepsym <varname>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a '\$'. The `unsetrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the variable `abc`, use

```
%unsetrepsym abc
```

### Setting Default Values Using `defrepsym`

Default variable values can be set in a Nastran data file using the following statement:

```
%defrepsym <varname>=<varvalue>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varvalue>`, where the start of the comment is indicated by a '\$'. The `defrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. The specified value will be used *only* if a value for `<varname>` was not previously set, i.e., by a `repsym` keyword on the command line or in an RC file or by a `%setrepsym` statement previously specified in the data file that has not been unset by a `%unsetrepsym` statement. For example,

```
%defrepsym abc=2.46e+2
```

### Clearing ("Unsetting") Default Values Using `undefrepsym`

The default value for a symbolic variable may be cleared ("unset") in a Nastran data file using the following statement:

```
%undefrepsym <varname>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<varname>`, where the start of the comment is indicated by a '\$'. The `undefrepsym` string is not case-sensitive and at least one blank must separate this string from the `<varname>` specification. For example, to clear the default value associated with variable `abc`, use

```
%undefrepsym abc
```

### Setting Width Information Using `setrepwidth`

Symbolic variable substitution default width information may be set in a Nastran data file using the following statement:

```
%setrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following `<widthinfo2>`, where the start of the comment is indicated by a '\$'. The `setrepwidth` string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. `<widthinfo1>` specifies the width information for the portion of the Nastran data file before the `BEGIN` statement and `<widthinfo2>` specifies the width information for the portion of the Nastran data file after the `BEGIN` statement. Each is specified using a `-w.p` specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the

separating comma is required if the Bulk Data Section width value is to be set, i.e., if `<widthinfo2>` is specified. For example,

```
%setrepwidth 0.0,wide
```

specifies that the symbolic substitution width specification is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

### Clearing ("Unsetting") Width Information Using `unsetrepwidth`

Symbolic variable substitution width information set using the `%setrepwidth` statement may be cleared in a Nastran data file using the following statement:

```
%unsetrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the `unsetrepwidth` string, where the start of the comment is indicated by a '\$'. The `unsetrepwidth` string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both width specifications.

### Setting Default Width Information Using `defrepwidth`

Default symbolic variable substitution width information may be set in a Nastran data file using the following statement:

```
%defrepwidth <widthinfo1>,<widthinfo2>
```

where the '%' character *must* be in column 1 and nothing else may appear in the record (except for optional comments following `<widthinfo2>`), where the start of the comment is indicated by a '\$'. The `defrepwidth` string is not case-sensitive and at least one blank must separate this string from the width specifications. There may not be any blanks within the width specifications. `<widthinfo1>` specifies the default width information for the portion of the Nastran data file before the BEGIN statement and `<widthinfo2>` specifies the default width information for the portion of the Nastran data file after the BEGIN statement. Each is specified using a `-w.p` specification or as one of the synonyms, as described above. If either width specification is omitted, the current width information for that section is not changed. Note that the separating comma is required if the Bulk Data Section width value is to be set, i.e., if `<widthinfo2>` is specified. For example,

```
%defrepwidth 0.0,wide
```

specifies that default symbolic substitution is to be 0.0 before the BEGIN statement and is to be -16.16 after the BEGIN statement.

### Clearing ("Unsetting") Default Width Information Using `undefrepwidth`

Default symbolic variable substitution width information may be cleared in a Nastran data file using the following statement:

```
%undefrepwidth
```

where the '%' character *must* be in column 1 and nothing else may appear in the record except for optional comments following the `undefrepwidth` string, where the start of the comment is indicated by a '\$'. The `undefrepwidth` string is not case-sensitive and must be followed by at least one blank. This statement does not have any arguments and clears both default width specifications.

### *General Information For Special Statements*

The %setrepsym, %unsetrepsym, %defrepsym, %undefrepsym, %setrepwidth, %unsetrepwidth, %defrepwidth and %undefrepwidth statements are deleted, logically, from the data file and will never be processed by the rest of Nastran unless an error is encountered while they are being processed. This is discussed in the [Error Handling, 3380](#).

### Requesting Symbolic Substitution

Symbolic variable substitution will occur when a string having the form

```
%<varname>,<widthinfo>:<varvalue>%
```

is found anywhere within a Nastran data file, except that this string may *not* span records, i.e., the substitution request must be on a single record (line). The leading and trailing '%' characters are required as is the <varname> field. The <widthinfo> field is optional. If it is omitted, the comma (,) separating it from the <varname> field may be omitted and the rules for determining what width specification will be used are discussed below. The <varvalue> field is optional and provides a way of specifying a default value, i.e., the “local default value”, as described below. If it is omitted, the colon (:) separating it from the <varname> (or <widthinfo>) field may be omitted. The rules for determining what symbolic value will be used as the substitution value are discussed below. For example, if the symbolic variable abc is to be replaced by its current value with no special processing (or if default width processing is to be used), the substitution request would be:

```
%abc%
```

If the symbolic variable is to be replaced by its current value, with the minimum field width to be 12 characters and with the value always to be left-justified, the substitution request would be:

```
%abc,-12%
```

### Quoting Rules For Symbolic Variable Values

- If a symbolic variable value is case-sensitive, if it contains leading, trailing or embedded blanks or if it contains percent characters, tab characters or other special characters, it must be quoted. (Note that "escape" sequences such as '\t' or '\n' are not given any special treatment; that is, they are left as is.)
  - If the value is part of a repsym keyword command-line specification, the quoting rules of the command shell being used apply.
  - If the value is part of a repsym keyword specified in an RC file, it must be enclosed in single quotes (').
  - If the value is part of a %setrepsym or %defrepsym record or if it specified as the “local default value” in a symbolic substitution request, quoting a symbolic variable value means enclosing the value in one of the following pairs of characters:

| Starting Quote Character | Ending Quote Character |
|--------------------------|------------------------|
| "                        | "                      |
| '                        | '                      |
| /                        | /                      |
| \                        | \                      |

| Starting Quote Character | Ending Quote Character |
|--------------------------|------------------------|
| [                        | ]                      |
| {                        | }                      |
| (                        | )                      |

If the first non-blank character encountered in a variable value specification is one of the starting quote characters, the variable value *must* be ended by the associated ending quote character. The actual variable value will be the (possibly null) string between (but not including) the starting and ending quote characters. If the variable value starts with one of the starting quote characters, it must be quoted using an alternate quote character.

### General Rules For Symbolic Variable Substitution

- Nested symbolic substitution is not supported. Even if the value associated with a symbolic variable name is, itself, in the format of a symbolic variable substitution request, that request will be ignored. That is, after symbolic variable substitution has occurred, the substituted string is *not* re-scanned.
- Determining what symbolic variable value will be used when a variable substitution request is encountered depends on where the variable value associated with the specified variable name was set. The *first* value encountered in the following hierarchy is the value that will be used:
  - A value specified in the Nastran data file using the `%setrepsym` statement, if there is one active, i.e., if it has not been deactivated by a `%unsetrepsym` statement.
  - A value specified on the Nastran command line or in RC files using the `repsym` keyword.
  - As part of the variable symbol substitution request, using the local default value, if there is one.
  - A value specified in the Nastran data file using the `%defrepsym` statement, if there is one active, i.e., if it has not been deactivated by a `%undefrepsym` statement.

This precedence follows normal MSC Nastran ordering, i.e., "the data file wins," while still providing great flexibility. Also, the ordering of the last two items in this hierarchy allows a user to set all defaults except for special cases and follows the idea that the specification "closest" to the use is the one used. If no replacement value is found, the substitution request will be ignored and the record will be unchanged.

- Determining what symbolic width specification will be used when a variable substitution request is encountered depends on where the width information has been specified and on the part of the Nastran data file that is being processed, i.e., is the variable substitution request before or after the first `BEGIN` statement. The *first* width specification value encountered in the following hierarchy is the specification that will be used:
  - A value specified in the symbolic substitution request itself, i.e., if a `<widthinfo>` entry was specified as part of the symbolic substitution request.
  - A value specified on a `%setrepwidth` statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by an `%unsetrepwidth` statement.

- A value specified on the Nastran command line or in RC files using the `repwidth` keyword corresponding to the current section in the Nastran data file.
- A value specified in the Nastran data file using the `%defrepwidth` statement corresponding to the current section in the Nastran data file, if there is one active, i.e., if it has not been deactivated by a `%undefrepwidth` statement.
- The program default value of exact (0 . 0).

This precedence also follows normal Nastran ordering, i.e., "the record wins followed by the data file wins," while still providing great flexibility.

- When running in licensing "Interlock" mode, i.e., in CRC validation mode, the following restrictions will be in effect. If a restriction is violated, Nastran processing will be terminated.
  - The `%setrepsym`, `%unsetrepsym`, `%defrepsym` and `%undefrepsym` statements are not allowed. Also, specifying a default value within the symbolic substitution request is not allowed. That is, symbolic variable values may only be set using the `repsym` keyword on the command line or in an RC file. Note that the `%setrepwidth`, `%unsetrepwidth`, `%defrepwidth` and `%undefrepwidth` statements *are* allowed.
  - A maximum of two symbolic substitution specifications are allowed per record and a maximum of ten symbolic substitution requests are allowed in the entire input data file.
  - Interlock CRC calculations will be made on the input record *before* symbolic substitution occurs. Note that any alterations to the record made as part of the CRC calculation processing will not affect symbolic substitution processing.

### Requesting Symbolic Substitution Replacement Information Using REPINFO

- A report of what symbolic substitutions were made is generated at the end of Nastran processing, with the level of detail in the report controlled by an "information level" flag set using the `repinfo=n` keyword, where `n` is an integer number that specifies the level of detail desired. The meanings the various values for `n` are as follows:

- 0 suppress the report altogether
- 1 report the various values assigned using the `repsym` keyword
- 2 same as 1 except add the various values assigned using the `setrepsym` statement
- 3 same as 2 except add the various values assigned using the `defrepsym` statement
- 4 same as 3 except add the various values assigned as local default values
- 5 same as 1 except add all locations where the specified `repsym` value was used
- 6 same as 2 and 5 except add all locations where the specified `setrepsym` value was used
- 7 same as 3 and 6 except add all locations where the specified `defrepsym` value was used
- 8 same as 4 and 7 except add all locations where local default values were used.

The report is written to the .f06 file. If there is not enough dynamic memory available to save the report information, the `repinfo` level may be reduced. When running in MSC Nastran, the default is `repinfo=1`. Otherwise, `repinfo=0` will be forced.

- Just as with other Nastran command line or RC file keywords, the `REPINFO` keyword is not case-sensitive.

## Error Handling

If an error is encountered processing a `setrepsym`, `unsetrepsym`, `defrepsym`, `undefrepsym`, `setrepwidth`, `unsetrepwidth`, `defrepwidth` or `undefrepwidth` statement, a comment string will be added to the record giving the error information and the record will be passed to Nastran (or the application reading the data file) as if the record was a normal Nastran data record. If an error is encountered in a record containing a symbolic substitution request, the symbolic substitution request will not be processed and, if `repinfo=1` or greater is in effect, a message giving information about the error will be written to the .log file. It is expected that the statements in error will not be valid Nastran statements and so will be flagged as an error.

## Examples

1. The value on an “OPTION” statement is to be settable using the command line, taking a default value of “OPT1val” (case-sensitive) if no command line value is set. The OPTION statement could be

```
OPTION=%Option:'OPT1val' %
```

and the command line parameter that would be used to set OPTION to a different value, OP2VAL (not case-sensitive), would be

```
RepSym=Option=op2val
```

2. An INCLUDE file contains records that are to be used four times in the Bulk Data Section of a Nastran data file, with the only difference being the value in Field 3 of one record. The first time the file is used, this field must contain the value 1.234, the second time this field must contain the value 4.567 and the last two times this field must contain the value -12.578. In all cases, the replacement field must be eight characters wide and the data must be left-justified in the field. Assuming that the symbolic variable is DATFL3 and that the include file name is incl.data, this could be done as follows:

In the include file, specify the following statements before the record to be modified:

```
%DefRepSym datfl3=-12.578
```

then the record to be modified could be specified as follows:

```
FL1      FL2      %datfl3%FL4      FL5      FL6
```

and, for completeness, specify the following record after the record to be modified:

```
%Undefrepsym datfl3
```

Then the data file would contain:

```
. . .
%setrepsym DATFL3=1.234
%DefRepWidth ,bulk
include 'incl.data'
```



```
. . .  
%setrepsym DATFL3=4.567  
include `incl.data'  
%Unsetrepsym datfl3  
. . .  
include `incl.data'  
. . .  
include `incl.data'
```

