MSC FlightLoads 2021.4

User's Guide

Corporate

5161 California Ave. Suite 200
University Research Park, Irvine, CA 92617
Telephone:(714) 540-8900
Toll Free Number: 1 855 672 7638
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa

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

Japan

Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific

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

Worldwide Web

www.mscsoftware.com

Support

https://simcompanion.hexagon.com/

Disclaimer

FL:V2021.4:Z:Z:Z:DC-UG-PDF

Contents

MSC FlightLoads and Dynamics User's Guide

1 Introduction

2 Getting Started

3 Aero Modeling

4 Aerodynamics

5 Aeroelasticity

6 Results Browser

7 Import/Export

A Panel Aerodynamics

B Splines

C Results Interface via XDB

D Aerodynamic and Aeroelastic Databases

E Static Aeroelastic Analysis

F Aero Mesh Interface File Format

1 References

1 Introduction

Loads in the Design of Flight Vehicles

Background

The calculation of flight loads is a critical part of air vehicle design. The structural design can only occur once the representative loads are provided to the designer. On the other hand, the prediction of accurate loads is a sophisticated and complex process that requires skilled and experienced engineers. They must integrate results from wind tunnel tests, computer simulations, historical data and empirical formulations into a number of loads cases that provide a realistic assessment of the flight vehicle's environment. Under these conditions, the vehicle must satisfy requirements imposed by regulatory agencies as part of the vehicle certification process.

Given the complexity and importance of the loads calculation, it has become a truism in air vehicle design that "the loads are always late." This means that the quantification of the loads is on the critical path in the development of a new or modified vehicle. It also implies that inaccurate initial loads that are corrected or updated after completion of the original structural design can have a serious negative effect on the overall development schedule. In the worst case, if the inaccurate loads are not detected until after the vehicle has entered flight testing, very costly redesign and retrofitting may have to occur or vehicle placards may be established to limit certain maneuvers, thus reducing operational performance.

MSC's Initiative

Due to their long history of successful application development for the aerospace industry, MSC Software has repeatedly been asked to assist in the development of an advanced flight loads calculation system. MSC Software's core MSC Nastran and MSC Patran products were developed to address analysis and design requirements for aerospace applications. MSC Software first introduced an aeroelastic capability into MSC Nastran in 1974, and this has been continuously maintained and enhanced.

Starting in late 1996, MSC Software assembled a development team dedicated to creating a system for providing timely and accurate flight loads information. This User's Guide documents the current release of MSC FlightLoads and Dynamics.

Architecture and Capabilities

MSC FlightLoads and Dynamics builds on the MSC Nastran and MSC Patran products, augmenting them with new system architecture concepts, enhanced data and model management, and integrated visualization tools that directly address flight loads and dynamics requirements.

Figure 1-1 provides a simplistic overview of the FLDS architecture.

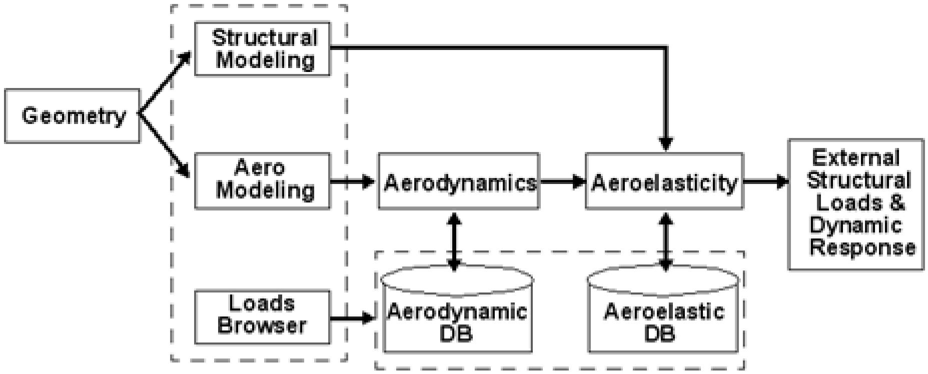


Figure 1-1 FLDS Architecture Overview

The FlightLoads system provides the capability to:

- Start with native geometry from user-preferred sources such as CAD applications, STEP AP203, or IGES files.
- Define the aerodynamic and structural models.
- Perform aerodynamic calculations.
- Analyze the combined structural-aerodynamic model to provide both component and total vehicle aeroelastic responses.
- View the results and produce external loads that can be passed to the stress group for detailed design and verification.
- Store intermediate results for subsequent retrieval in further analyses.

Integration of MSC Nastran and MSC Patran

MSC FlightLoads and Dynamics represents an integration of MSC Nastran and MSC Patran products, as shown in Figure 1-2.

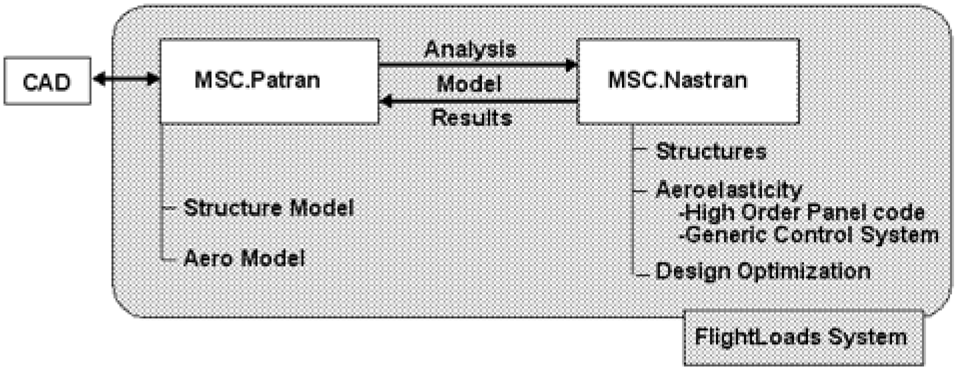


Figure 1-2 FlightLoads Integration of MSC Nastran and MSC Patran

The MSC Patran component of FlightLoads provides access to the user's CAD system for basic geometry information (STEP or IGES sources are also supported) and then creates the aerodynamic and structural models that are used in calculating the loads.

The MSC Nastran system handles the computationally intense calculations that produce basic loads information. These can then be passed back to the graphics package for visualization of the results, including the components that were used in building up the final solution. For loads determination, the end result is the creation of loads in the MSC Nastran bulk data format that can be applied to the structural model to provide detailed stress information.

About This Guide

This User's Guide describes in detail the numerous features of MSC FlightLoads and Dynamics. The guide is organized to emphasize the graphical nature of MSC FlightLoads and Dynamics. Following this Introduction there is an overview section, "Getting Started," that sets the stage for the remaining chapters, which lead the user through each of the system modules.

Separate chapters discuss:

- Aero Modeling
- Aerodynamics
- Aeroelasticity
- Results Browser
- Import/Export

Appendices provide theoretical and user information on the solutions central to the aeroelastic analyses, including the following:

- Panel Aerodynamics
- Splines
- Results Interface via XDB

- Aerodyanmic and Aeroelastic Databases
- Static Aeroelastic Analysis
- MSC Nastran Input File

This document is intended to provide a thorough introduction to MSC FlightLoads and Dynamics, but it can address only a small fraction of the issues that are invoked by the system. MSC has many other related documents and the user is advised to search out these relevant publications. Two especially important ones are:

- MSC Nastran Aeroelastic Analysis User's Guide
- MSC Patran User's Guide

The MSC Bookstore is on the World Wide Web at www.mscsoftware.com; select the engineering-e.com tab and the BooksMart button to display a comprehensive list of publications available from MSC Software.

2 Getting Started

Prerequisites

Software

MSC.FlightLoads and Dynamics is based on MSC.Nastran and MSC.Patran. The MSC.Nastran code must include the Aero I option. If supersonic analyses are of interest, it is also necessary to have Aero II. The MSC.Patran code must include the MSC.Nastran Preference.

Experience

It is assumed that the MSC FlightLoads user has some experience with both the underlying software and the analysis procedures involved in the system. It is also expected that the user has basic familiarity with flight loads concepts such as rigid and elastic loads, stability derivatives, control surfaces, maneuvers and other similar concepts. Some familiarity with static aeroelasticity in MSC.Nastran (SOL 144) and flutter in MSC Nastran (SOL 145) is beneficial.

Structural Model

One of the components of an aeroelastic model in MSC.FlightLoads and Dynamics is the structural model. In this manual, these structural models are presumed to exist. They can enter the MSC.FlightLoads and Dynamics system by import or by direct creation in MSC.FlightLoads and Dynamics (MSC.Patran) using the structures preference. This manual does not cover any structural modeling issues. Please refer to the MSC.Nastran Preference of MSC.Patran for information on structural modeling.

Terms

The following is a list of various terms and acronyms that you will need to know when using the MSC FlightLoads and Dynamics system and this manual.

FLD - MSC FlightLoads and Dynamics

kmin - Minimum Reduced Frequency

kmax - Maximum Reduced Frequency

Fmin - Minimum Cyclic Frequency

Fmax - Maximum Cyclic Frequency

Vmin - Minimum Velocity

Vmax - Maximum Velocity

v - velocity in consistent length units (length/s)

c - reference chord

b - reference span

S - reference area

ω - radian frequency

s - seconds

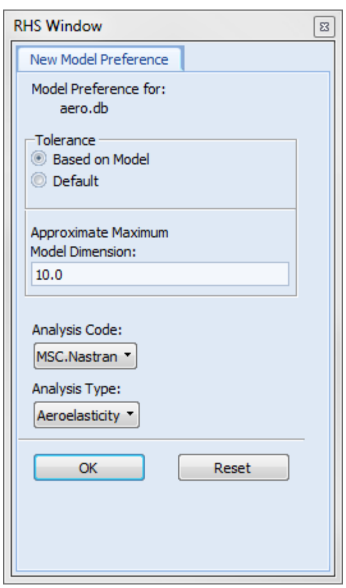
g - acceleration due to gravity in consistent length units (length/s/s)

dimensionless rate = $\frac{\omega b}{2v}$ for anti-symmetric; or $\frac{\omega c}{2v}$ for symmetric

Invoking MSC.FlightLoads and Dynamics

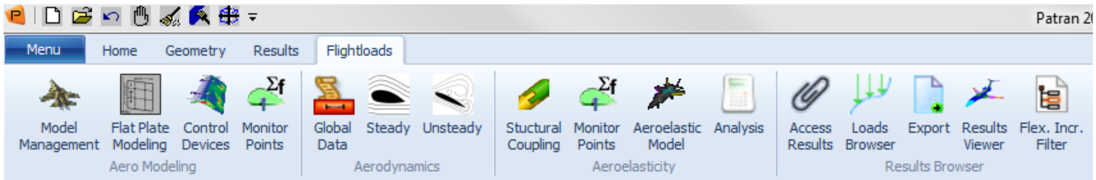
MSC.FlightLoads and Dynamics is invoked using the `p3fld` or the `mscfld` command. These two commands are the same as the command that invokes MSC.Patran except that it uses the **-ifile** option to replace the use of the init.pcl file with the init_fld.pcl file that MSC.FlightLoads and Dynamics needs. This displays the familiar MSC.Patran main form. Users can still run MSC.Patran from these two commands.

When a new database is opened, the New Model Preferences form displays and the user can access FlightLoads by selecting Aeroelasticity as the Analysis Type; this allows for immediate aeroelastic modeling and analysis. If the user chooses an Analysis Type of Structural or Thermal, FlightLoads is subsequently accessed in an existing database through the Preferences/Analysis menu. FlightLoads coexists with the available structural and thermal analyses in the MSC.Nastran Preference as an aeroelastic analysis.

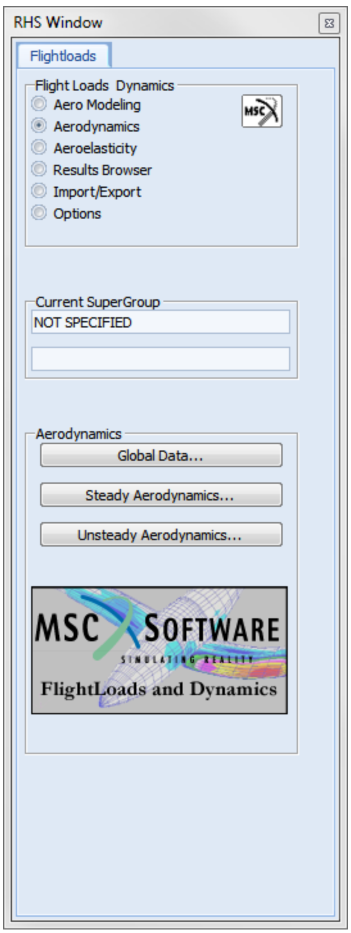


Parameter	Description
Analysis Type	As an alternative, if the database has been opened with an analysis type other than Aeroelasticity, the Preferences Option on the main form can be chosen, and the Aeroelasticity Analysis type selected.

After the Aeroelasticity preference is picked, a menu customized for MSC.FlightLoads and Dynamics displays.



Many of the options available in a Structural analysis type (such as LBCs and Materials) disappear from the main menu. The Geometry switch is available to assist in the definition of the aerodynamic model. If the FlightLoads switch is selected, the MSC.FlightLoads and Dynamics main menu displays.



Parameter	Description
Flight Loads Dynamics	This main menu has six modules. Select an option to display the main form for that module.

Note:	When selected, a window displays with the version number and information about the various parts of the product.

Most of this MSC FlightLoads and Dynamics User's Guide is devoted to an explanation of the menus that are accessed from this main menu. The next five chapters of this manual gives detailed information on each of the options. Note that the Options module is as part of the Model Management section of Chapter 3.

A general description of each of the MSC FlightLoads and Dynamics main options follows.

Aero Modeling

Chapter 3 - Aero Modeling module allows the user to define the aerodynamic geometry, including wings, bodies and control surfaces. The release supplies the aerodynamics currently available in MSC.Nastran to perform static aeroelastic analysis, namely the Doublet- Lattice (subsonic) and ZONA51 (supersonic) aerodynamics and flutter. See Panel Aerodynamics (App. A). Aero Modeling includes the management of various aerodynamic models, the creation and subsequent processing of the aerodynamic lifting surfaces and bodies, the definition of control systems, and various other model visualization and verification tools.

Chapter 4 - Aerodynamics module allows the user to define the aerodynamics as steady or unsteady and set global data.

Chapter 5 - Aeroelasticity module is used to couple and subsequently analyze the aerodynamic and structural models. MSC.FlightLoads and Dynamics provides for model evolution (i.e., beam - stick to 3D FEM Structural models) and for the coexistence of multiple aerodynamic mesh representations. Data reuse is also supported. Aerodynamics and aeroelastic data can be archived for subsequent reuse in analysis. A variety of static aeroelastic analyses can be performed, including flexible trim, rigid trim and the calculation of flexible load increments.

Chapter 6 - Results Browser is a key feature of MSC.FlightLoads and Dynamics. It allows the user to view external loads on the aerodynamic and structural models, providing insight into the flight environment. The graphical display of these loads is extremely useful in spotting modeling errors or areas for model refinement. External loads data can reside in an aerodynamic or aeroelastic database, an MSC.Patran database or an MSC.Nastran results file.

Chapter 7 - Import/Export is used to support the extensive legacy information that exists for MSC.Nastran aeroelasticity. The user can now import aerodynamic and spline models from an existing MSC.Nastran bulk data file and subsequently manipulate this data using the Aero Modeling module.

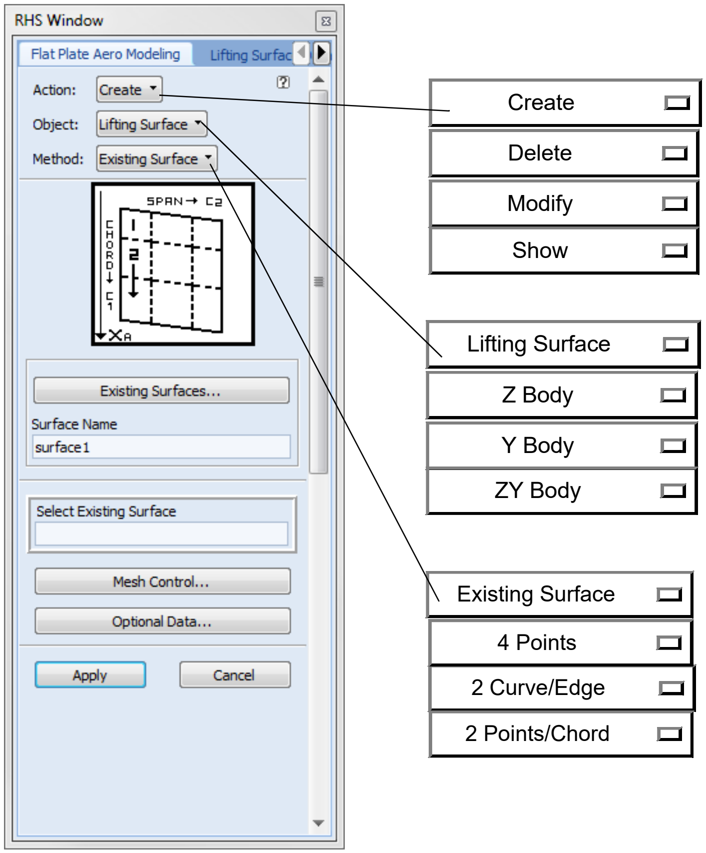
Note:	During typical usage, the user first populates the MSC.Patran database with the structural model. This is done by using the File/Import function within the MSC FlightLoads Import function. Spline data cannot be imported unless the corresponding structural nodes are already in the database.

Graphical User Interface

The previous section introduced the high level MSC.FlightLoads and Dynamics menus. This section provides a brief general description of the use of forms in the system. The conventions discussed here are those of MSC.Patran.

Working with FlightLoads Forms

The application chapters of this guide show most of the forms provided in the MSC.FlightLoads application and describe how to complete and execute each form. Therefore, this section provides only a brief general description of how to complete the forms. One of the Flat Plate Aero Modeling forms is shown as an example.



To use this and the other forms, the user starts at the top of the form and works to the bottom. First, select an Action then an Object and last, a Method from three pulldown menus. The fields on the lower portion of the form displays, depending on user selections.

To complete the fields in the lower portion of the form, the user may either click on toggle buttons, select from pulldown menus, type surface names and IDs or other numerical data, depending on the requested information. Special selection menus also display to help select entities in the viewport; this displays their numeric IDs in the currently selected form field. After all the fields on the form are completed, click on Apply to execute the desired operation.

Working with Subforms

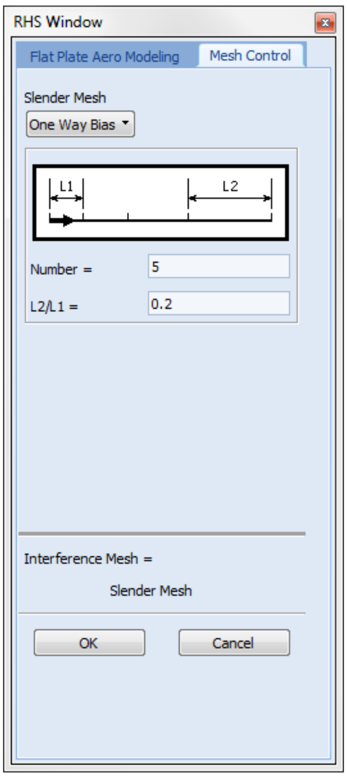
Clicking on an ellipsis field (those that end with "...") displays a subform. The available subforms from the Create/Lifting Surface form are shown on the preceding page.

- Existing Surfaces

- Mesh Control
- Optional Data

Each of these subforms allows input of additional data. The layout of the subforms is unique to the requested data and typically does not follow the standard Action/Object/Method convention.

The fields on particular subforms may vary depending on user input. For example, there are four different layouts for the Mesh Control subform, depending on the selected option from the pulldown menus on the subform. A sample Mesh Control subform is shown below.

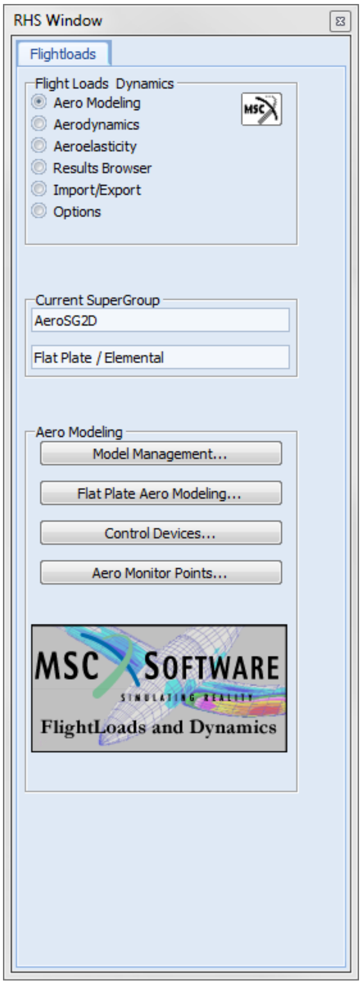


3 Aero Modeling

Introduction

The Aero Modeling module is where you create, modify and manage the aerodynamic model(s) that you subsequently couple to a structure for aeroelastic analysis. All modeling functions for aerodynamics are found within this module. The Aero Modeling module is divided into four areas that are available by selecting one of the ellipsis buttons at the bottom of the form:

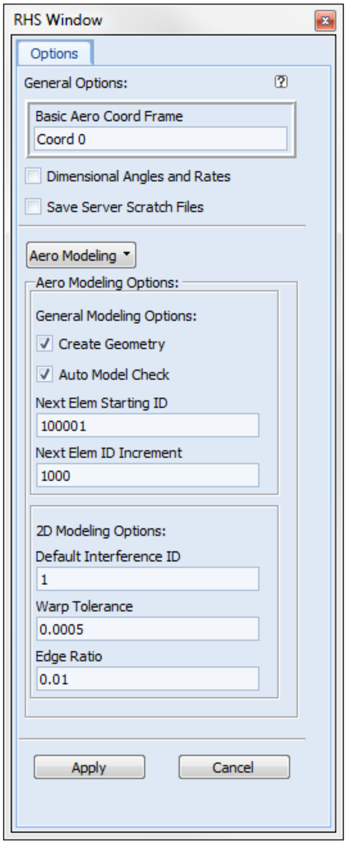
- Model Management - Collects groups of aerodynamic entities (lifting surfaces and bodies) into SuperGroup(s). Each SuperGroup is an aerodynamic model (section 3.3).
- Flat Plate Aero Modeling - Provides the geometric definition and meshing for the aerodynamic model (section 3.4).
- Control Devices - Provides the control device (e.g., flap, elevator, aileron, etc.) information for the model, as well as operational limits (section 3.5).
- Aero Monitor Points - Provides the ability to create, delete, modify, or show the aero monitor points (section 3.6).
- Options - Provides basic default parameters and system behavior for model creation (section 3.2).



Parameter	Description
Aero Modeling	Five Aero Modeling Application Sections are available.

Aero Modeling Options

The Aero Modeling Options form is discussed first. It allows the user to define default data values and system behavior, as shown in the form illustration. Generally, the defaults provided for these options are adequate. An important exception to this is the Basic Aerodynamic Coordinate Frame. The flow is in this coordinate system's X-direction and any planes of symmetry are defined as this coordinate system's origin, X-Z plane and Y-Z plane.



Parameter	Description
The Basic Aero Coord Frame	positive X-axis describes the positive flow direction.
Dimensional Angles and Rates	The database always stores the data in rads, however, you may enter your calculations in either dimensional units by toggling this option ON.
Create Geometry	Automatically creates geometry when it is not present. Default is ON. (The aerodynamic mesh is automatically associated to this geometry.)
Auto Model Check	Perform basic model checks at construction time. Default is ON.

Parameter	Description
Next Elem ID Increment	This number is the next highest available number, rounded to the increment place plus 1 (e.g., 9567 -> 10001). This number is also accessible from the Optional Data subform from the Aero Modeling form.
1000	After creation of an aero surface or body, increment the Next Elem Starting ID to the next available digit indicated by this number (. i.e., 11580 would increment to 12001). This increment can provide gaps between surface and body IDs, which can be later filled in by a more refined mesh without causing ID overlap.
2D Modeling Options:	This invokes a subform which allows the user to change the current aero SuperGroup.
Default Interference ID	Default Interference Group ID (IGID) for surfaces and bodies. Any surfaces and bodies sharing an IGID have aerodynamic interaction. Default is 1. This number is also accessible from the Aero Modeling “Optional Data” and "Additional Data" subforms.
Warp Tolerance	This value defines the allowable skew of an aero surface.
Edge Ratio	This value determines the largest allowable distance between corner points of an aero surface before scaling of the edges is needed.

Model Management

An aerodynamic model is made up of a collection of lifting surfaces and, optionally, bodies. In FlightLoads terminology, each individual lifting surface or body is an Aero Group. A SuperGroup is then a collection of these Groups into a model (Aerodynamic Configuration) that is to be used for analysis. The model database can contain any number of SuperGroups. The user must then identify which SuperGroup is being used in the current analysis (see Aeroelastic Model definition in the Aeroelasticity module). FlightLoads uses a default SuperGroup if the user has not explicitly created one.

Overview

When Model Management is selected, the first form displayed is Create/SuperGroup/Flat Plate. Shown adjacent to this form are all the different possible Actions, Objects, and Methods for Model Management:

Most of the Model Management forms are shown and annotated in the following pages, grouped by Object as follows:

- SuperGroup Forms (Create, Delete, Modify, Set Current)
- Aero Groups Forms (Rebuild)
- Orphan Groups Forms (Show)

Definitions for the Model Management Objects

A SuperGroup is the name of an aerodynamic model which is a collection of Aero Groups. Any number of SuperGroups can be defined in a single database. Aero Groups cannot be shared by SuperGroups. A SuperGroup name is limited to 8 characters; this name is used to qualify model data, by configuration, on an

MSC.Nastran Aerodynamic or Aeroelastic Database and corresponds to the "AECONFIG" Case Control name in MSC.Nastran subcases.

Aero Groups are individual aerodynamic surface or body macroelements. (In MSC.Nastran terminology, these are CAERO1 or CAERO2 bulk data entries.)

Orphan Groups are Aero Groups that are not assigned to any SuperGroup. This can happen, for example, if a SuperGroup is deleted.

Model Management Forms

This subsection provides annotated illustrations for the Model Management forms. Note that the **Type** for all of these forms is Flat Plate which refers to the Doublet Lattice/ZONA51 aerodynamic modeling methods that are characterized as being 2D panel methods (flat surfaces in 3D-space) as opposed to 3D methods (general surfaces in 3-space). The 3D version of the form is exactly the same as the Flat Plate versions shown on the next few pages. The available set of forms are:

SuperGroup Forms:

- Create/SuperGroup
- Delete/SuperGroup
- Modify/SuperGroup
- Show/SuperGroup
- Set Current/SuperGroup

Aero Groups Forms:

- Rebuild/Aero Groups

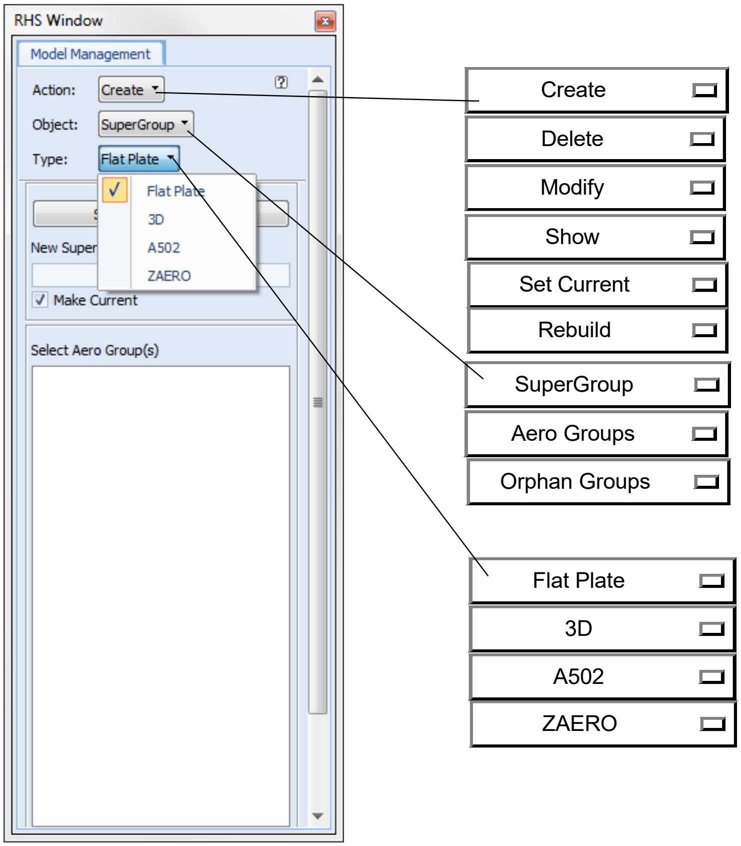
Orphan Groups Forms:

- Show/Orphan Groups

SuperGroup Forms

Create/SuperGroup

This form allows the user to provide a name for the SuperGroup that is meaningful within the design task. The components (Aero Groups) that make up the SuperGroup can also be identified. If this form is not invoked, the default SuperGroup name (AeroSG2D, for Aerodynamics SuperGroup 2D and AeroSG3D, for Aerodynamics SuperGroup 3D) is used and all of the Aero Groups automatically belong to this Group.



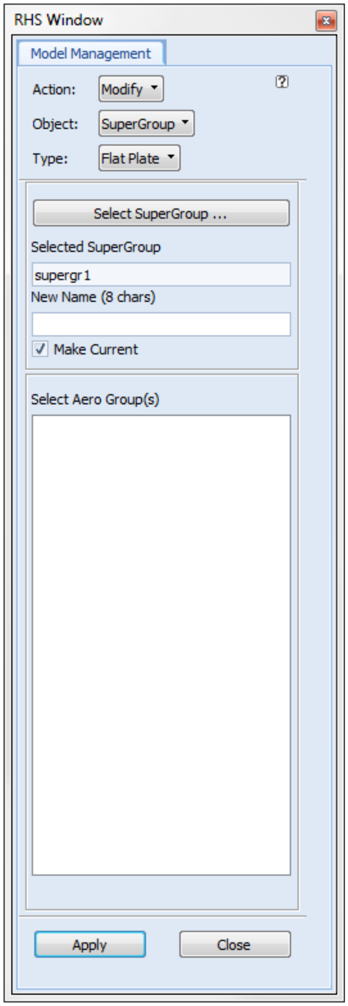
Delete/SuperGroup

This form (not shown) deletes the selected SuperGroup(s). The associated Aero Groups are not deleted but become "Orphan". The current SuperGroup does not appear within the Select SuperGroup(s) box and cannot be deleted.

Note: Only SuperGroups of the given type are displayed within the select SuperGroup(s) box.

Modify/SuperGroup

This form allows the user to change the Aero Group members in the selected SuperGroup. After the user selects from the existing SuperGroups, the form displays all the available Aero Groups with the current members highlighted. If a member is deselected, it becomes an "Orphan". If a member is selected, it becomes part of the SuperGroup and is removed from any SuperGroup it currently resides in and it loses its current "Orphan" status. If the name of an existing SuperGroup is to be modified, the user can either enter the desired new name or select the old name from existing SuperGroups and modify it within the New Name box.



Parameter	Description
Select SuperGroup	Lists all previously created SuperGroups of a specified Type. Select a single SuperGroup from the list of previously created SuperGroups of the specified type.
Selected SuperGroup	Original SuperGroup name.

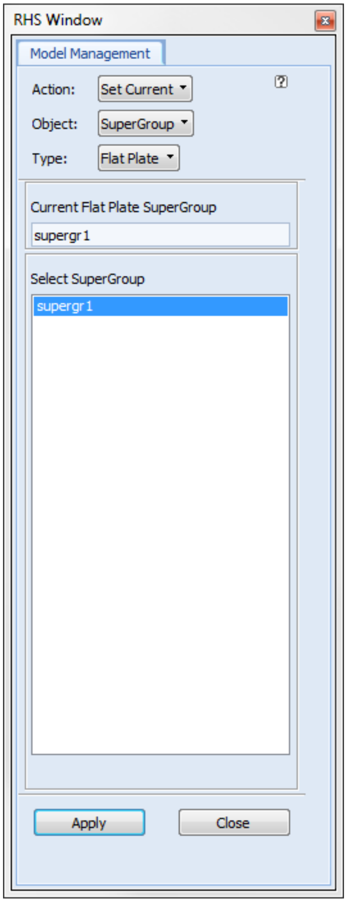
Parameter	Description
New Name	New SuperGroup name limited to 8 characters. This name is used as the configuration qualifier on MSC.Nastran Aerodynamic and Aeroelastic Databases.
Select Aero Group(s)	Listbox of Aero Group(s). The user may select either none, one or multiple member groups. Only the Groups of the currently selected Type are shown.
Make Current	The user can optionally Make Current a newly created SuperGroup.
Apply	Click Apply to create the SuperGroup. If any Groups are already members of a SuperGroup, the user is asked a "Yes for All, Yes, No, No For All" question to replace the relationship.

Show/SuperGroup

This form (not shown), displays which Aero Groups are associated with the selected SuperGroup.

Set Current/SuperGroup

This form lists all the created SuperGroups and highlights the current SuperGroup. The user can now choose to set another Current SuperGroup.



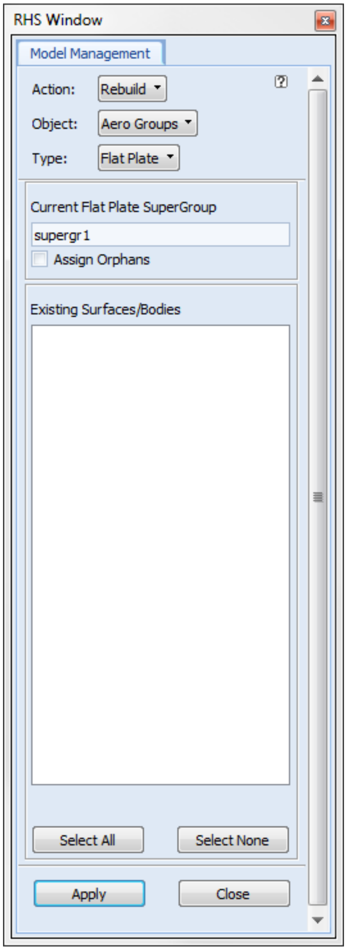
Parameter	Description
Type	The type of the current SuperGroup.
Select SuperGroup	Lists all of the previously created SuperGroups of the specified Type with the currently set SuperGroup highlighted.

Aero Groups Forms

Rebuild/Aero Groups

MSC.FlightLoads and Dynamics completely manages all Aero Groups and SuperGroups, including their contents. These Groups are, however, also available under the Group application in MSC.Patran but SHOULD NOT BE CHANGED THERE. In the situation where an Aero Group becomes modified, this form allows the user to restore the Aero Groups.

If there are orphaned Aero Groups, they can either be placed in the Current SuperGroup or left as orphans, depending on the selection of the "Assign Orphans to SuperGroup" toggle.



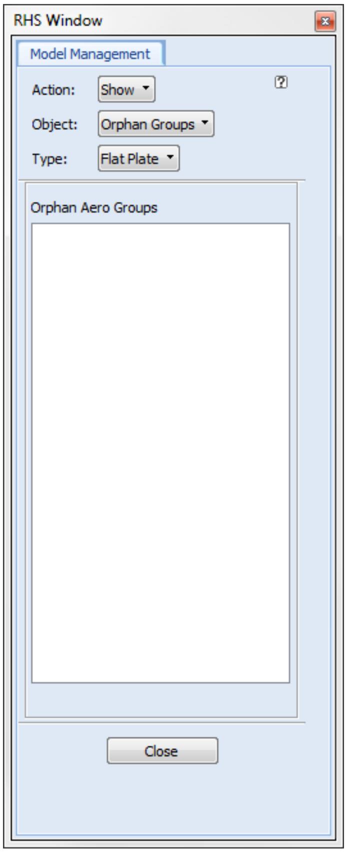
Parameter	Description
Existing Surfaces/Bodies	List of existing aero surfaces and bodies. Users may rebuild one or more Aero Groups. All of Type are initially highlighted.

Note: Current SuperGroups are shown according to Type.

Orphan Groups Forms

Show/Orphan Groups

This form allows the user to view any Orphan Aero Groups. These can be subsequently reassigned to a SuperGroup.



Parameter	Description
Orphan Aero Group(s)	Read Only" list of Orphan Aero Groups.

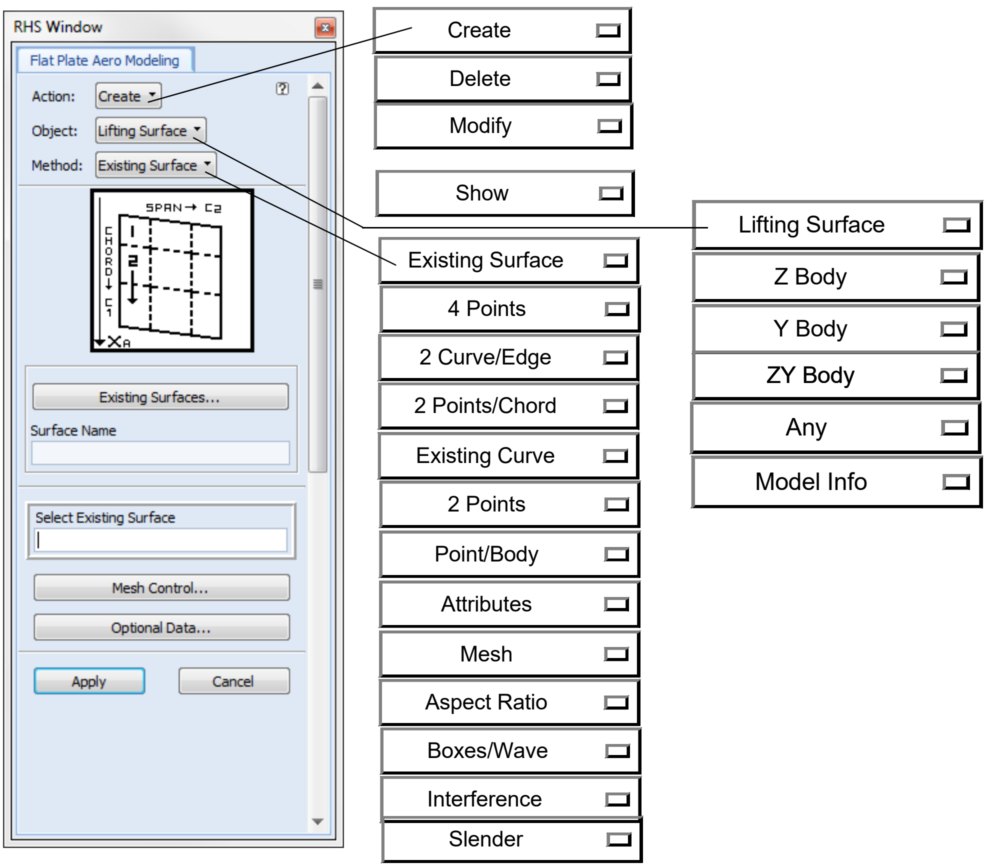
Flat Plate Aero Modeling

This section describes how to create and manipulate Aero Groups. Both Lifting Surfaces and Bodies are supported to provide a graphical user interface to the Doublet Lattice/ZONA51 aerodynamic capability of MSC.Nastran. Panel Aerodynamics (App. A) of this document provides theoretical and modeling guideline information on these aerodynamic methods.

Note: This button is disabled if the current SuperGroup is a 3D SuperGroup.

Overview

When the user selects Flat Plate Aero Modeling, the first form displayed is Create/Lifting Surface/Existing Surface. Shown adjacent to this form below are all the different Actions, Objects, and Methods options available:



Most of the Flat Plate Aero Modeling forms are shown and annotated in the following pages, grouped by Object as follows:

- Lifting Surface Forms (Create, Delete, Modify, Show)
- Body Forms (Create, Delete, Modify, Show)
- Any (Delete, Modify)
- Model Info Form

Definitions for the Flat Plate Aero Modeling Objects

A Lifting Surface is the representation of a winglike surface that provides the primary lifting capability for the wing surface. Both the Doublet Lattice Method (subsonic) and ZONA51 (supersonic) aerodynamic codes support lifting surfaces modeled as a trapezoid with inboard and outboard edges that are **parallel** to the direction of the flow (the X-coordinate of the aerodynamic coordinate system).

The Doublet Lattice Method (but not ZONA51) also supports the ability to model bodies, such as fuselages, external fuel tanks and other "stores" and/or engine nacelles. The theory requires the user to distinguish the bodies based on the types of motion they can sustain. A Z Body can only move in the Z-direction of the aerodynamic coordinate system. A Y Body can only move in the Y-direction of the aerodynamic coordinate system.

A ZY Body can move in the Z and Y directions. This option would typically be selected in an asymmetric analysis and for bodies that are not on the plane of symmetry (such as engine nacelles).

Flat Plate Aero Modeling Forms

This subsection provides annotated illustrations for the Flat Plate Aero Modeling forms. The available set of forms are:

Lifting Surface Forms

Create Forms

- Create/Lifting Surface/Existing Surface
- Create/Lifting Surface/4 Points
- Create/Lifting Surface/2 Curve/Edge
- Create/Lifting Surface/2 Points/Chord
- Create/Lifting Surface Subforms
  - Existing Surfaces
  - Mesh Control
  - Optional Data

Delete Forms

- Delete/Lifting Surface

Modify Forms

- Modify/Lifting Surface

Show Forms

- Show/Lifting Surface /Attributes
- Show/Lifting Surface /Mesh
- Show/Lifting Surface /Aspect Ratio
- Show/Lifting Surface/Boxes/Wave
  - Non-Dimensional Option
- Show/Lifting Surface Subforms
  - Fringe Attribute

Body Forms

Z Body, Y Body, and ZY Body Forms

- Create/Body/Existing Curve
- Create/Body/2 Points
- Create/Body/Point/Body
- Create/Body Subforms
  - Existing Bodies
  - Mesh Control
  - Additional Body Data
- Modify/Body
  - Mesh Control
- Delete/Body
- Show/Body/Attributes
- Show/Body/Mesh
- Show/Body/Interference
- Show/Body/Slender
- Delete/Any
- Modify/Any
- Show Model Information
- Show/All Model Information

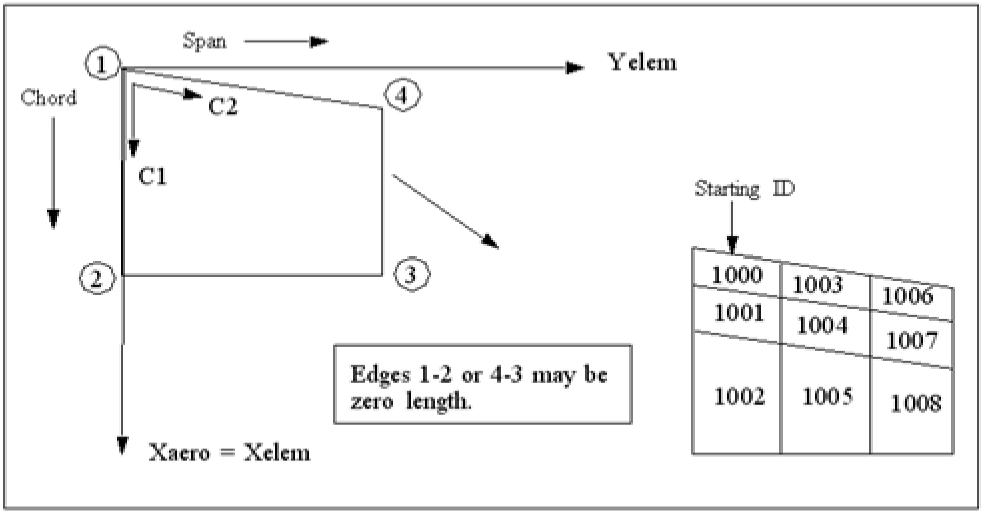
Lifting Surface Methods

A lifting surface is a trapezoidal flat plate that has inboard and outboard edges aligned with the X-axis of the aerodynamic coordinate system as shown in the sketch below. FlightLoads provides the user with four alternative ways of creating these trapezoids as described in the following pages. Each of the Create/Lifting Surface Methods has three subforms:

1. Existing Surfaces lists previously created surfaces.

2. Mesh Control presents the methods for creating the mesh of boxes on which the analysis is performed.
3. Optional Data allows the user to modify the element numbering for the mesh, specify the Interference Group ID, associated bodies, and reference coordinate frames.

The user may wish to invoke this latter form in order to force the numbering to conform to values that are suited to the analysis environment. Following is a list of the Flat Plate Aero Modeling forms and subforms related to lifting surfaces:

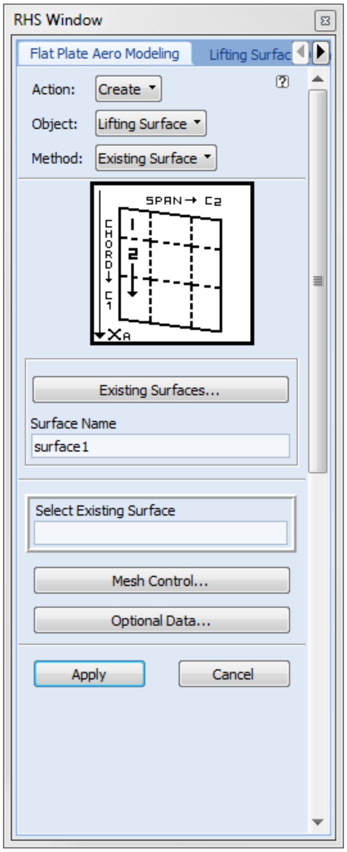


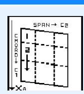
Note:	1. Edges 1-2 and 4-3 must be parallel to the Xaero axis. 2. Points 1-4 must be planar. 3. Edge 1-2 or 4-3 can be of zero length. 4. The mesh normals are assumed to align with the Zaero axis.

Lifting Surface Forms

Create/Lifting Surface/Existing Surface

Existing Surface is the default method for creating lifting surfaces. This form allows the user to select an existing surface that was either accessed as CAD geometry or created using the Geometry application. The surface corners are used to construct the lifting surface, ignoring any surface curvature. On this form, the user selects the existing surface and provides a lifting surface name. The Lifting Surface name becomes an Aero Group after successful creation.

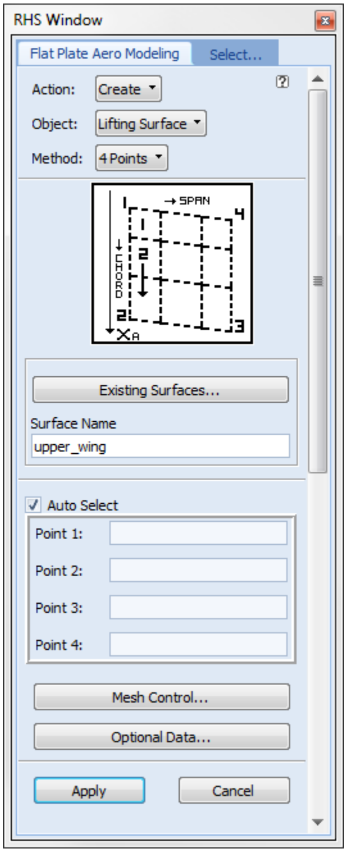


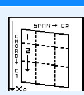
Parameter	Description
	This icon serves as a reminder for the assumed surface and coordinate system orientations by MSC.Nastran.
Existing Surfaces	Invokes the Existing Surfaces subform which lists all previously created surfaces.

Parameter	Description
Surface Name	New Lifting Surface Name. This name must be unique.
Select Existing Surface	Specify the Existing Surface here by picking a surface in the viewport.
Mesh Control	Invokes the Span and Chord Mesh Control subform.
Optional Data	Invokes the Optional Data subform, which allows the user to change: - starting element id - interference group id - associated bodies - surface mesh reference coordinate frame

Create/Lifting Surface/4 Points

Under this option, a trapezoidal or triangular region is created by defining four points. The points can be structural nodes, geometric points or x,y,z locations as shown in the example below. Note that all four points need to be defined before Apply can be invoked. The four points must lie in a plane and the plane can not be inclined to the incoming flow. A three-sided surface is created when two of the defined four points are the same.

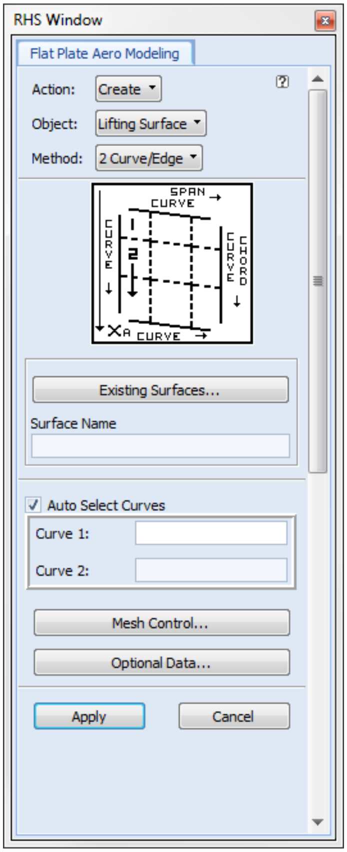


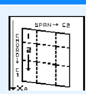
Parameter	Description
	This icon serves as a reminder for the assumed surface and coordinate system orientations by MSC.Nastran.
Existing Surfaces	Invokes the Existing Surfaces subform which lists all previously created surfaces.
Surface Name	New Lifting Surface Name. This name must be unique.

Parameter	Description
Points 1,2,3,4	Selection fields for the four points. **Note:** Points 1 & 2 or 3 & 4 may be the same points to yield a 3-sided surface.
Auto Select	Auto Select automatically places cursor focus into the next point select box after successful point selection.
Mesh Control	Invokes the Span and Chord Mesh Control subform.
Optional Data	Invokes the Optional Data subform, which allows the user to change: - starting element id - interference group id - associated bodies - surface mesh reference coordinate frame

Create/Lifting Surface/2 Curve/Edge

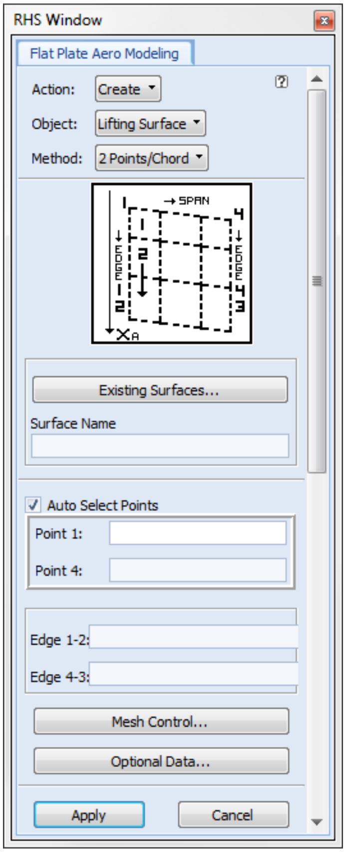
Under this option, a trapezoidal or triangular region is created from two curves or edges. Typically, these two curves would define the inboard and outboard edges or the leading and trailing edges. A four-sided surface is created when two opposing curves/edges are selected. A three-sided surface is created when two intersecting edges are selected (i.e., inboard and trailing edges).

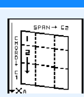


Parameter	Description
	This icon serves as a reminder for the assumed surface and coordinate system orientations by MSC.Nastran.
Existing Surfaces	Invokes the Existing Surfaces subform which lists all previously created surfaces.
Surface Name	New Lifting Surface Name. This name must be unique.
Curves 1,2	2 Curve/Edge selection fields.
Mesh Control	Invokes the Span and Chord Mesh Control subform.
Optional Data	Invokes the Optional Data subform, which allows the user to change: - starting element id - interference group id - associated bodies - surface mesh reference coordinate frame

Create/Lifting Surface/2 Points/Chord

This option conforms most closely to what displays on the MSC.Nastran bulk data entry for the lifting surface. Points 1 and 4 define the locations of the inboard leading edge and outboard leading edge, respectively. Edge 1-2 is the length of the inboard chord while Edge 4-3 provides the length of the outboard chord.



Parameter	Description
	This icon serves as a reminder for the assumed surface and coordinate system orientations by MSC.Nastran.
Existing Surfaces	Invokes the Existing Surfaces subform which lists all previously created surfaces.
Surface Name	New Lifting Surface Name. This name must be unique.

Parameter	Description
Points 1, 4	2 leading edge Points Selection fields.
Edges 1-2, 4-3	Associated chord dimensions, assumed to be in the flow direction.
Mesh Control	Invokes the Span and Chord Mesh Control subform.
Optional Data	Invokes the Optional Data subform, which allows the user to change: - starting element id - interference group id - associated bodies - surface mesh reference coordinate frame

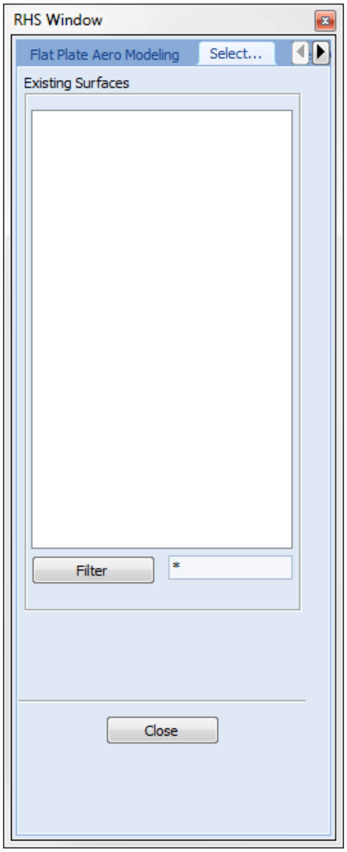
Create/Lifting Surface Subforms

Three subforms can be invoked from the Create/Lifting Surface Form:

- Existing Surface
- Mesh Control
- Optional Data

Existing Surfaces

This form simply lists all existing lifting surfaces. Selecting a surface does *not* result in any information being transferred back to the Create/Lifting Surface form.

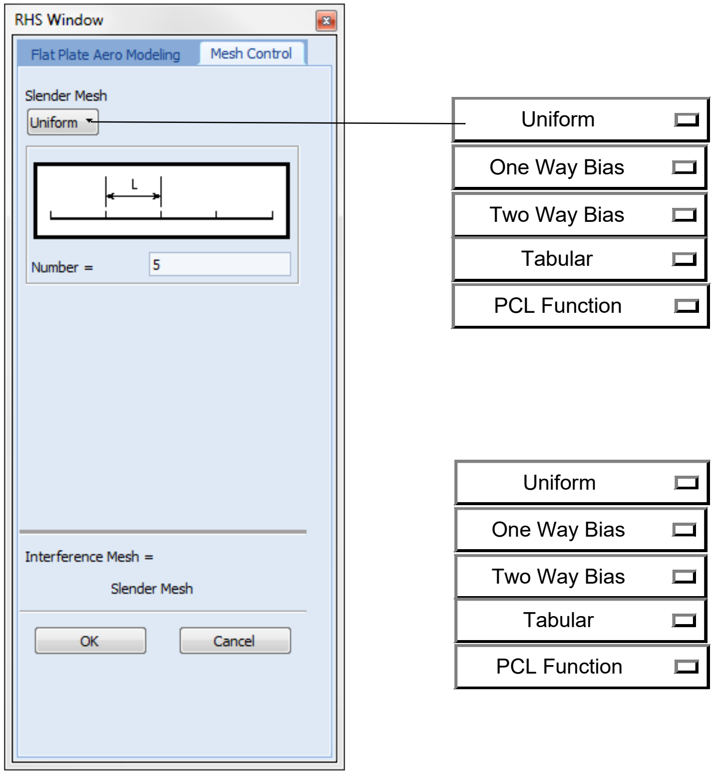


Mesh Control

As described in Panel Aerodynamics (App. A), the flat plate aerodynamic methods solve for the pressures at a discrete set of points contained within a set of boxes that are meshed onto the lifting surface. The boxes are arranged in strips parallel to the freestream and it is the user's task to define the chord-wise distribution of these boxes within a given strip and to define the span-wise strips themselves. This is done with the Mesh Control Subform described here. Guidelines on creating the mesh are included in Panel Aerodynamics (App. A).

On the Mesh Control Subform, the user can select a Mesh Control Type from the pulldown menu for both the Span mesh and the Chord mesh (the types do not have to be the same in the two directions). After a Mesh Control Type is selected, the form changes to support the chosen method. The example form shown here is for the Uniform Mesh Control Type. As its name implies, this creates equal length or width boxes of the number specified in the Number box.

The Span and Chord directions are defined in the parent form icon. Chord is always in the flow direction while Span is along the wing and perpendicular to the chord.



The following page contains illustrations of the form fields that appear for each of the options for Mesh Control Type.

Uniform

This form allows the user to specify an equally distributed mesh along the chosen direction.

One Way Bias

This form allows the user to specify the ratio of the lengths of the first and last boxes and the intermediate boxes are distributed accordingly. This option may be of use in defining the spanwise box distribution since a value less than 1.0 would concentrate boxes at the outboard edge where additional boxes are advised to capture the rapidly changing pressure distribution at the wing tip.

Two Way Bias

This form allows the user to specify the ratio of the middle box to the first and last boxes. Intermediate boxes are distributed accordingly. This option may be of use in defining the chordwise box distribution since a value less than 1.0 would concentrate boxes near the leading and trailing edges, where the pressure gradients are the highest.

Tabular Mesh

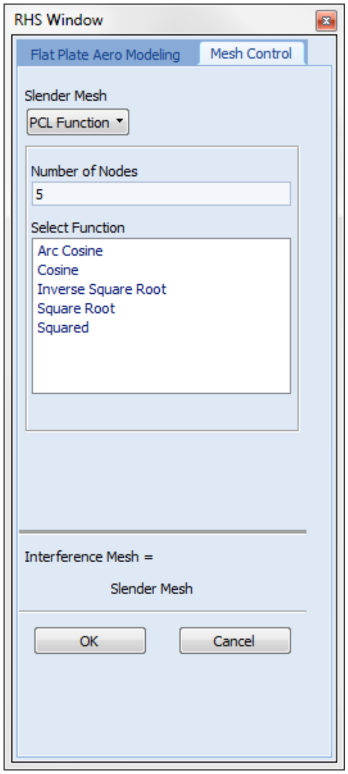
This form allows the user to explicitly define the box cut locations. Note that there is one more cut in each direction than there are boxes. The input values are in fractions of chord or span and can therefore range in value from 0.0 to 1.0. It is not necessary that the endpoints be 0.0 or 1.0, although this would be the most typical case. Exceptions are: 1) creating multiple lifting surfaces that represent the same trapezoidal area but specify different ranges for the mesh distribution (so that there is no overlap in meshes) or, 2) the lifting surface may extend to the fuselage centerline, and only the exposed portion of the wing, (that starts at the outer radius of the fuselage), would be meshed.

PCL Function

This option allows the user to select from one of several available options for performing the meshing. Only the listed PCL functions are available. The Cosine distribution is often the ideal selection for the chordwise pattern. Note that this option asks for the number of nodes, which is one more than the number of boxes in the mesh. The PCL functions are:

1. ArcCosine
2. Cosine
3. Inverse Square Root
4. Square Root
5. Squared

Mesh Control Subform (continued)



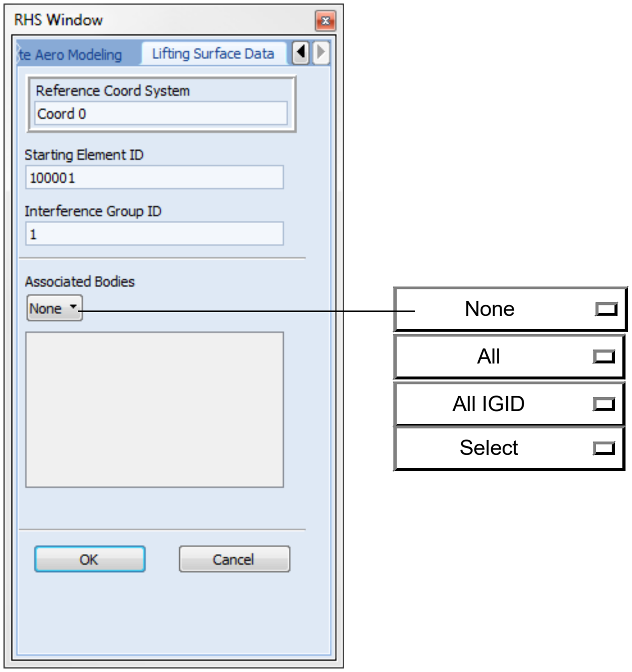
Note: The Tabular spreadsheet is initially set to 25 rows. This can be expanded by the user as required, just continue to add data.

Optional Data

The Optional Data subform invoked from the Create/Lifting Surface form allows the user to modify default values. In particular, it may be desirable to input the lifting surface geometry in a local coordinate system specific to the surface.

This coordinate system can be specified on the form. A different numbering scheme may be desired for labeling the mesh; if so, all mesh box ID's must be unique. Panel Aerodynamics (App. A) provides rules for insuring uniqueness. If there are bodies in the model, this subform allows the user to selectively define associated bodies of the lifting surface (see Panel Aerodynamics (App. A)).

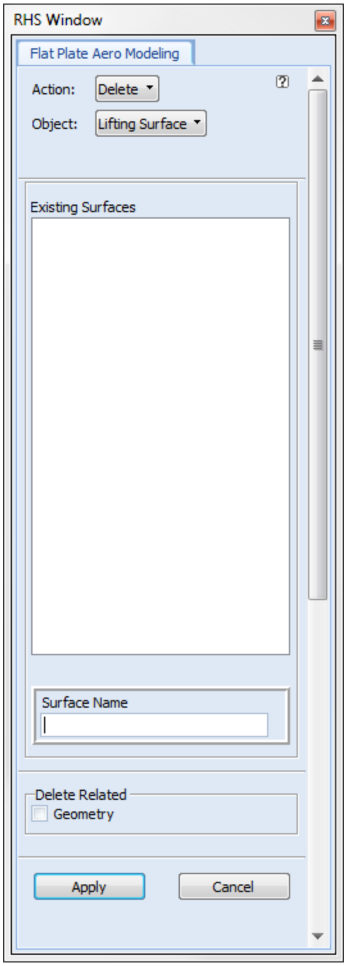
The interference group ID is used to indicate which lifting surfaces and bodies interact with each other; this is only available for the subsonic aerodynamics. Usually, it is desired to have all the Aero Groups share a single IGID; however, in some cases several may exist to either reduce computation time or to perform a study on the importance of interference (between a wing and tail, for example).



Parameter	Description
Reference Coord System	The Reference Coordinate System is used to describe the aero mesh geometry on the MSC.Nastran CAERO1 entry. If left blank, the default Base Aero CS from the Options form is used. A Cartesian system required.
Starting Element ID	Surface starting node and element ID. This value is obtained from the Aero Modeling Options form.
Interference Group ID	Initially = current IGID specified on Aero Modeling Options form.
Associated Bodies	A flag is stored with the property set indicating that at analysis time, ALL, ALL IGID, NONE or Selected bodies are to be used. **Note:** If Select is chosen, then the select list below is shown. This is the list of existing "Flat Plate" bodies (Z, Y, ZY). One or more bodies must be selected.

Delete/Lifting Surface

This form allows the user to select which Lifting Surfaces are to be deleted.

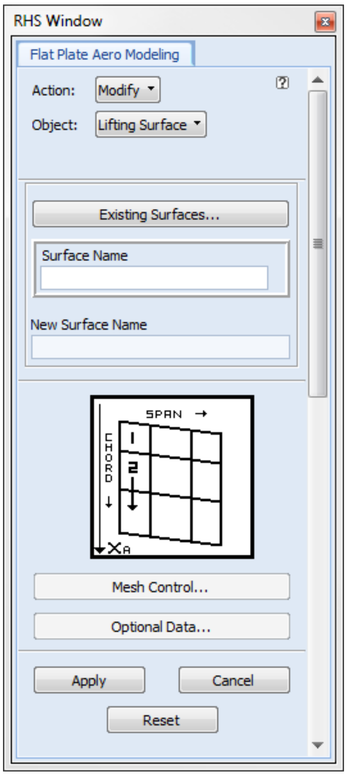


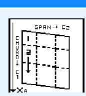
Parameter	Description
Existing Surfaces	Identifies surfaces to delete by selecting from the Existing list, or (see next)
Surface Name	Place the cursor in the Surface Name databox, then graphically select the surface. If multiple surfaces are graphically selected, then the cycle pick menu displays.

Note: Delete Related Geometry is set to OFF by Default.

Modify/Lifting Surface

This form is used to modify an existing Lifting Surface. All associated data are available for modification. If the mesh distribution is altered, the original starting ID is retained if possible.

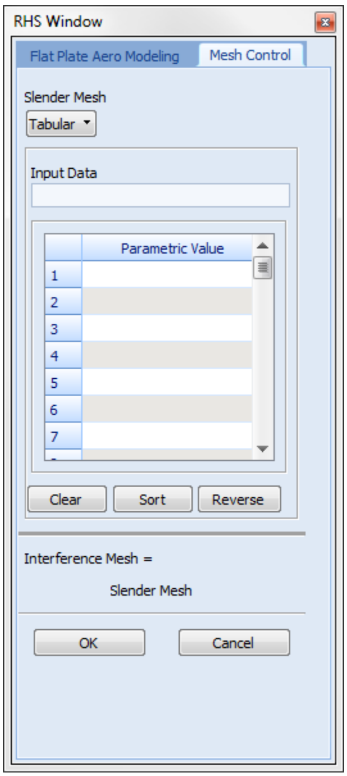


Parameter	Description
	This icon serves as a reminder for the assumed surface and coordinate system orientations by MSC.Nastran.
Mesh Control	The Mesh Control form is initially set to Tabular input with the appropriate Span and Chord values. However, any mesh construction may be subsequently used.

Note: 'Mesh Control' and 'Optional Data' buttons remain disabled until an Existing Surface is selected. Both are then enabled, and the Existing Surfaces button and Surface Name databox are disabled.

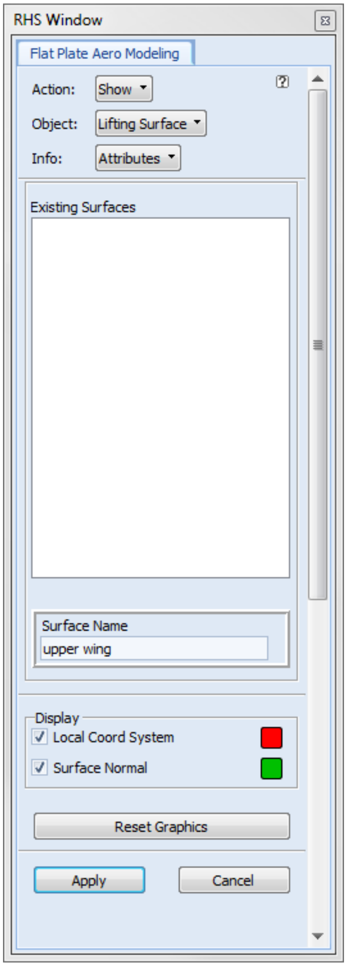
After the surface is identified, its name displays in the New Surface, and the Mesh Control and Optional Data buttons are enabled. A New Surface can be provided as well as a modified Mesh and Optional Data. If the Mesh Control button is selected, a subform displays that allows the user to change the mesh distribution (see Mesh Control form below).

The existing data are presented in a tabular form and each node point can be individually altered by highlighting the old data and entering the new value in the Input Data window. The user can also select any of the other meshing methods and enter data in an identical way as was done during the Create step.



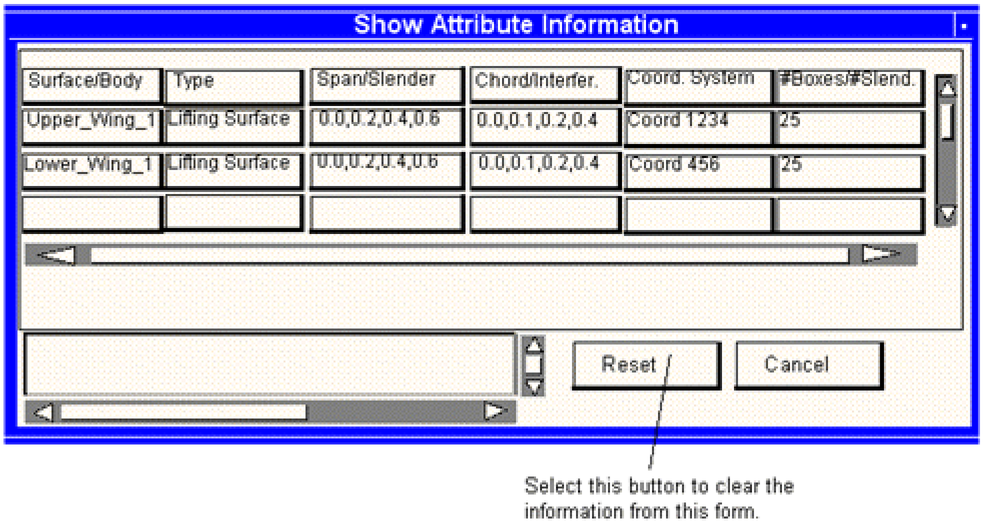
Show/Lifting Surface/Attributes

The Show/Lifting Surface/Attribute form is used to display Lifting Surface Attributes (type, span and chord parametric mesh distributions, and reference coordinate system).



Parameter	Description
Existing Surfaces	Select one surface to display desired information, or (see next)
Surface Name	Place the cursor in the Surface Name databox, then graphically select the surface. If multiple surfaces are graphically selected, then the cycle pick menu displays.
Local Coord System	displays the positive direction (C1) for each aero element.
Surface Normal	displays the positive lift direction for each aero element.
Apply	Take the selected Surface Name and display it and its associated data in the appropriate subform.

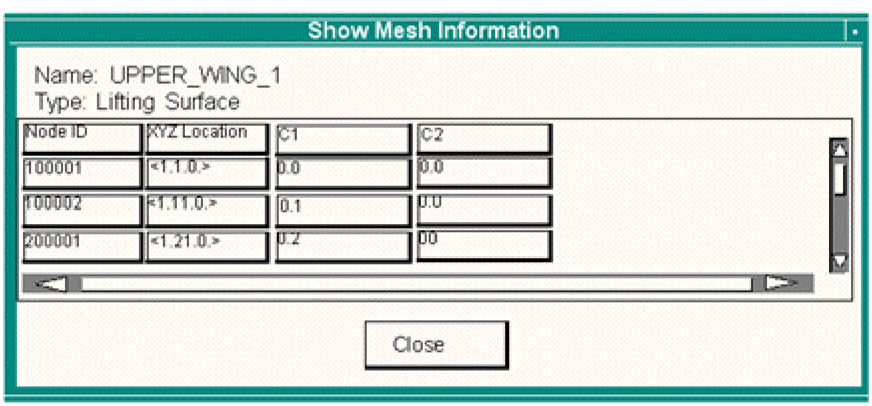
This subform is displayed when the Apply button on the Show/Lifting Surface/Attributes form is selected. Note that this form does not remove the data from the previous viewings but rather appends the new data at the end.



Show/Lifting Surface/Mesh

This form (not shown), looks just like the Show/Lifting Surface/Attributes form. This form is used to display the Mesh Information (grid locations in the isoparametric, nondimensional chord and span coordinates C1 and C2, respectively) for the selected Lifting Surfaces.

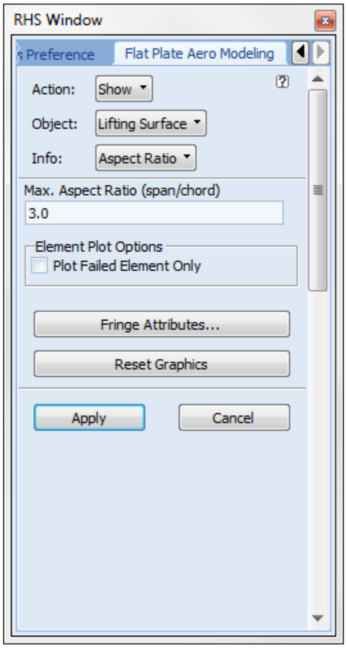
This subform is displayed when the Apply button on the form is selected.



Note:	The Aspect Ratio is defined as the span dimension for each box divided by the chord dimension. You can use the aspect ratio logic under the previously mentioned Finite Element form.

Show/Lifting Surface/Aspect Ratio

This form computes and shows the individual Box Aspect Ratio (mean span/mean chord) for all Lifting Surfaces in the current group. It can also be used to display only those elements that fail the plot criteria. This is useful to assess the model quality using the recommended modeling practices of MSC Nastran. By default, the aerodynamic boxes are fringed according to their aspect ratio values. Users may, optionally, check failure criteria where any element values greater than a specified value (default=3.0) are color coded according to a user-selected color (default is red).



Parameter	Description
Maximum Aspect Ratio	Maximum Aspect Ratio the user wishes to see. It is this number that is used to determine the failure criteria.
Plot Failed Element Only	If this toggle is selected, then only those elements that fail the Maximum Aspect Ratio criteria are plotted.
Fringe Attributes	It controls the way colors are assigned to segments of the range of aspect ratios. **Note:** The same form as for the Finite Element: Verify/Quad/Aspect form in MSC.Patran.

Note:	The Aspect Ratio is defined as the span dimension for each box divided by the chord dimension. You can use the aspect ratio logic under the previously mentioned Finite Element form.

Show/Lifting Surface/Boxes/Wave

In unsteady aerodynamics, the number of aerodynamic model degrees of freedom in one wavelength (distance the flow travels during one unsteady cycle) is a critical parameter in determining chordwise mesh refinement.

The wavelength is a function of the freestream velocity, v, and the frequency of the unsteady, harmonic oscillatin, f

```
wavelength = V/f, f in Hertz
```

So the number of boxes in a wavelength can be computed for each box as the wavelength divided by the mean chordlength.

Boxes/wave x

$$\frac{v}{(f)(local - chord)}$$

but, in unsteady aerodynamics, we often use a nondimensional frequency, *k*

$$k = \frac{2\pi fc}{2v}$$

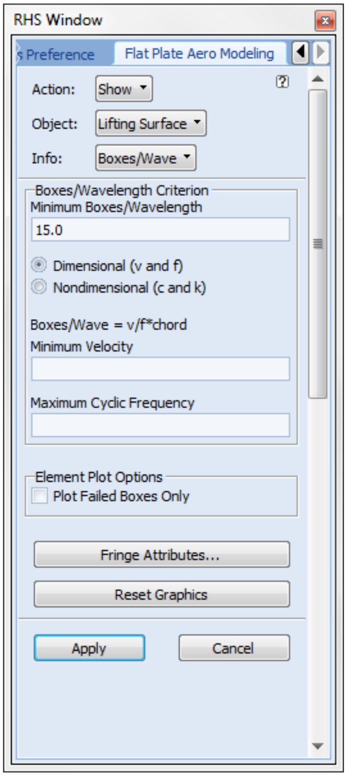
where *C* is a reference length and *v* is a reference freestream velocity at which we compute the aerodynamics. In these nondimensional terms, the user can also define the boxes in wavelength as

```
Boxes/Wave =
```

$$\frac{\pi c}{k(local - chord)}$$

To ensure converge, you must look at the minimum number of boxes/wavelength given the intend frequency and velocity range of the analysis. This corresponds to the maximum k value or, dimensionally, to the combination of the minimum velocity and highest frequency of interest. In modeling terms, one typically would choose the lowest velocity of interest and the highest natural frequency in the retained set of normal modes. (Note that the extremely low velocities demand very high numbers of elements.)

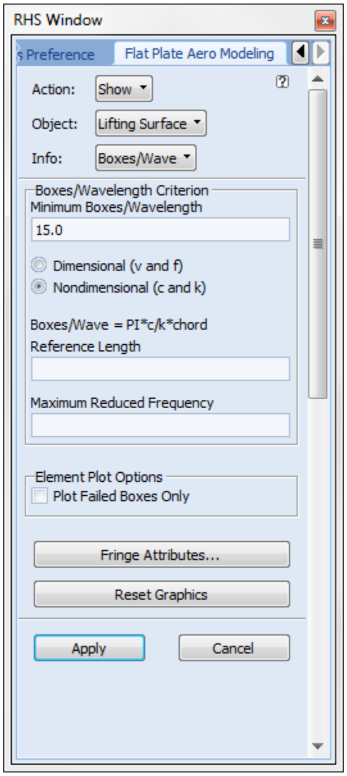
This form computes and shows the boxes per wavelength for all individual Lifting Surface boxes in the current group. This can be done by using the Dimensional equation (Boxes/Wave = v/f* chord) or by using the Non-dimensional equation (Boxes/Wave = PI* c/k* chord). It can also be used to display only those boxes that fail the recommended criteria as used in MSC.Nastran.



Parameter	Description
Minimum Boxes/Wavelength	Minimum Boxes per Wavelength that is acceptable for this model.
Boxes/Wave = v/f*chord	Equation that is used in computing the boxes chord length.
Minimum Velocity	Minimum velocity (v) to be used with this model.
Maximum Cyclic Frequency	Maximum cyclic frequency (f) to be used with this model.
Plot Failed Boxes Only	If this toggle is selected, then only those boxes that are greater than the computed Boxes/Wave is plotted.
Fringe Attributes	It controls the way colors are assigned to segments of the range of aspect ratios. **Note:** The same form as for the Finite Element: Verify/Quad/Aspect form in MSC.Patran.

Non-dimensional Option

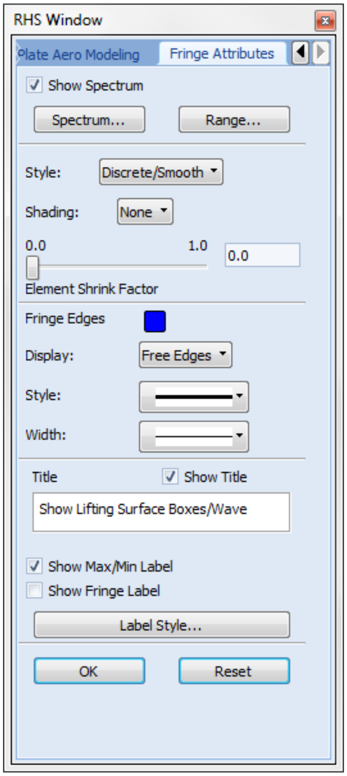
This form is similar to the one on the previous page and only those things that are different for the Non-dimensional options are described below.



Parameter	Description
Boxes/Wave = v/f*chord	Equation that is used in computing the boxes chord length.
Reference Length	Reference length (c) that is used.
Maximum Reduced Frequency	Reduced frequency (k) that is used.

Fringe Attributes

This Fringe Attributes form allows the user to better control the fringe plot resulting from showing box chords or boxes per wavelength.



Note:	This form is the same as the one for Finite Element: Verify/Quad/Aspect in the structural preference of MSC Patran.

Body Forms

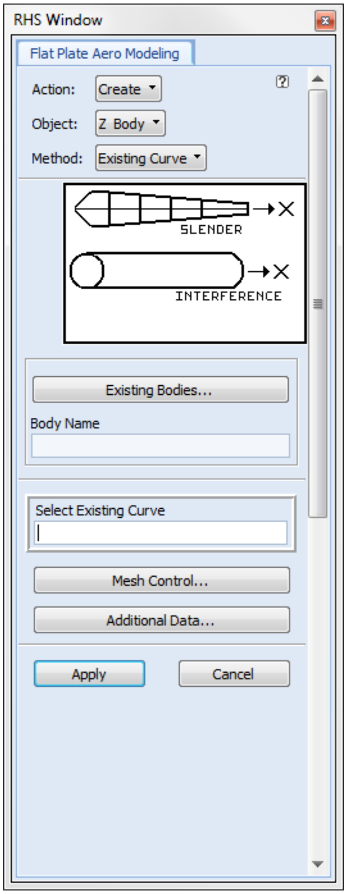
Z Body, Y Body and ZY Body

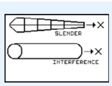
Following is a list of the Flat Plate Aero Modeling forms and subforms related to Z Body, Y Body and ZY Body objects.

All of the forms shown in this section are valid for any of the three Body types; for simplicity, Z Body is shown as the object on all of the forms.

Create/Body/Existing Curve

Existing Curve is the default method for creating Bodies. This form allows the user to select an existing curve. The curve end points are used to construct the Body, ignoring curvature. The newly created Body becomes an Aero Group.

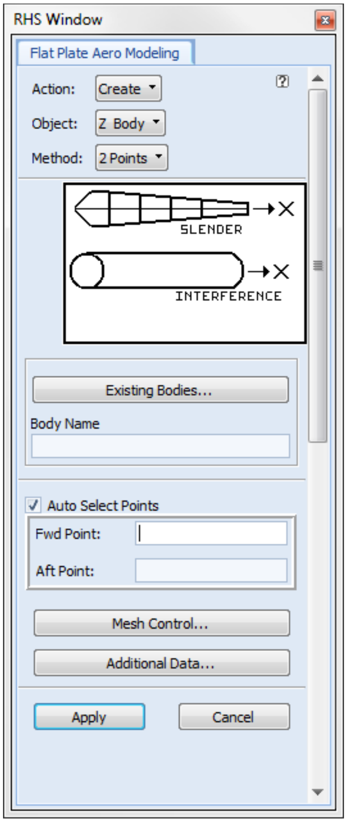


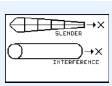
Parameter	Description
Method	Default Method is Existing Curve.
	This icon serves as a reminder of the assumed coordinate system.
Existing Bodies	Lists all previously created bodies.

Parameter	Description
Body Name	Name for the new Body must be unique.
Select Existing Curve	Specify Existing Curve here by picking a curve in the viewport or typing its name.
Mesh Control	Invokes the Slender Body Mesh Control subform.
Additional Data	Additional data input required for: - Body 1/2 widths - theta values

Create/Body/2 Points

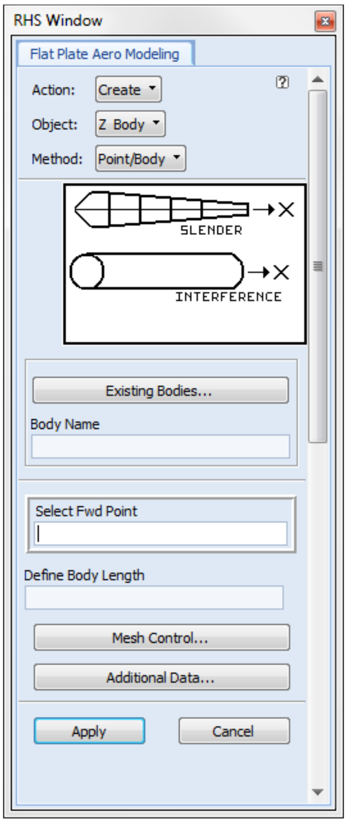
This form allows bodies to be created using two points. The points can be structural nodes, geometric points or X, Y, Z locations. The points must define a body parallel to the X-axis of the Aero System.



Parameter	Description
Method	Default Method is Existing Curve.
	This icon serves as a reminder of the assumed coordinate system.
Existing Bodies	Lists all previously created bodies.
Body Name	Name for the new Body must be unique.
Forward and aft points	Fields to select the forward and aft body points.
Mesh Control	Invokes the Slender Body Mesh Control subform.
Additional Data	Additional data input required for: - Body 1/2 widths - theta values

Create/Body/Point-Body

This form supports the body definition found in the MSC.Nastran input file for CAERO2 entries. The user is requested to define the body forward point and the body length.



Parameter	Description
Existing Bodies	Lists all previously created bodies.
Body Name	Name for the new Body must be unique.

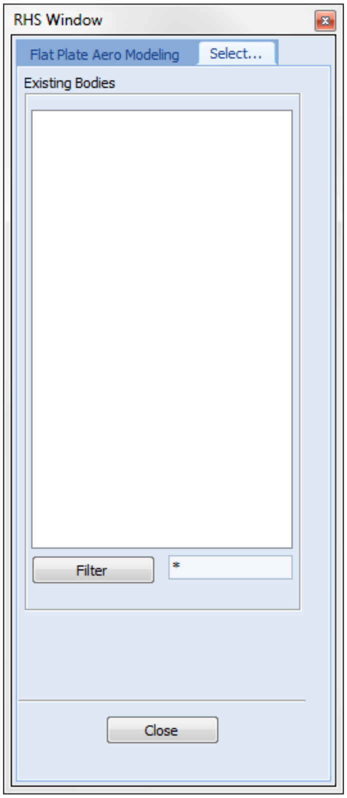
Parameter	Description
Select Forward Point	Field to select forward body point.
Define Body Length	Field to define body length. **Note:** The body length is assumed to be in the positive flow direction.
Mesh Control	Invokes the Slender Body Mesh Control subform.
Additional Data	Additional data input required for: - Body 1/2 widths - theta values

Create/Body Subforms

Three subforms are available from the Create/Body form: Existing Bodies, Mesh Control, and Additional Data. Each are discussed in more detail in the following sections. These subforms are applicable for all bodies (Z, Y, ZY).

Existing Bodies

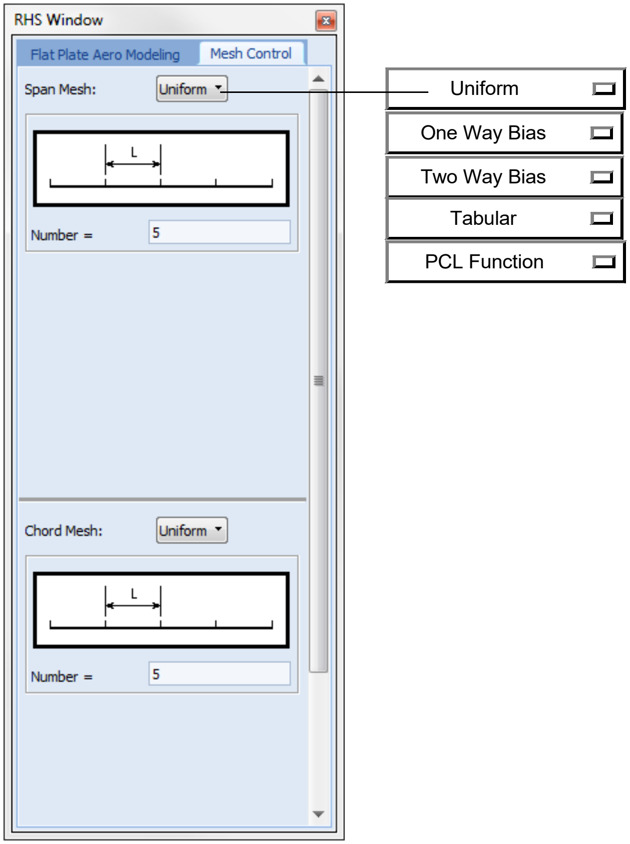
This form is used to list all existing bodies. Selection of a body does *not* result in any information being transferred back to the parent form.



Mesh Control

On this form, the user selects a Mesh Control Type from the menu for the Slender Body. The Interference Body mesh is fixed to be equal to the slender mesh. After a Mesh Control Type is selected, the form changes to support the chosen method.

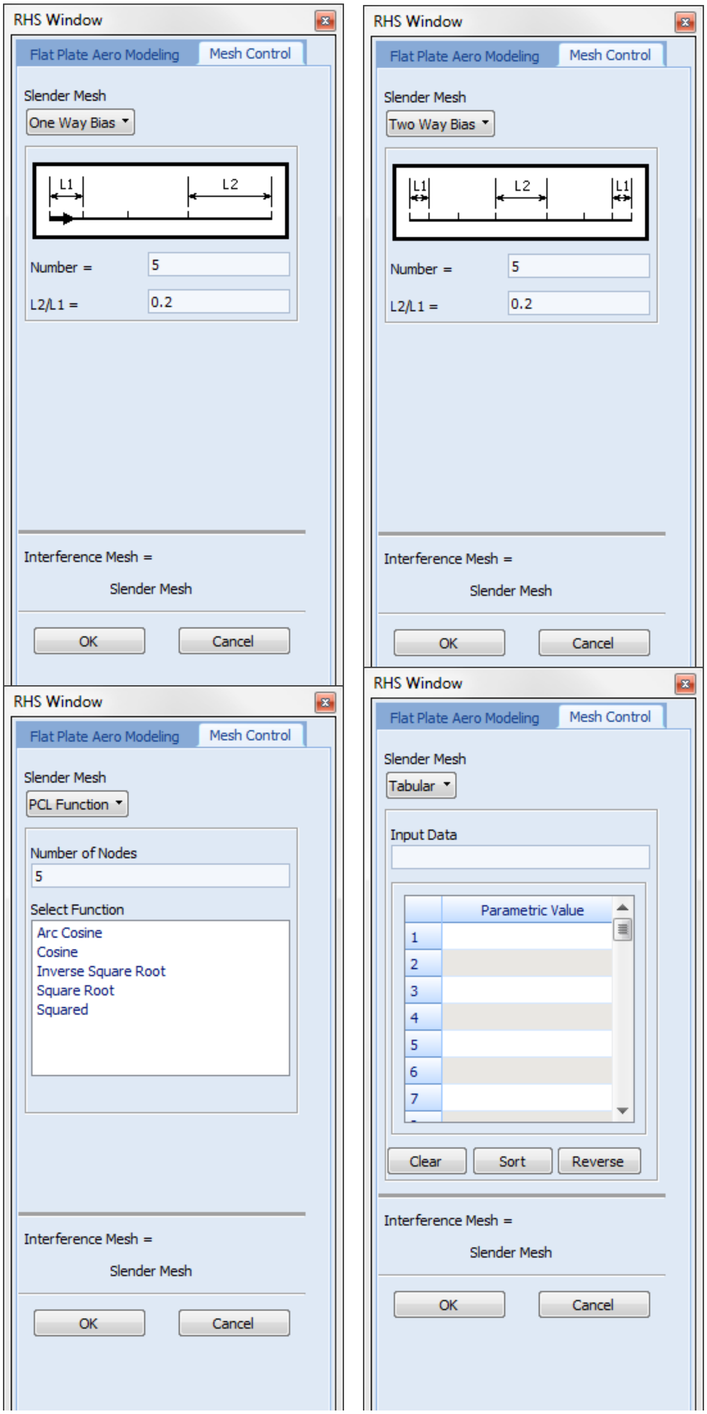
The example form shown here is for the Uniform Mesh Control Type. Following this are illustrations of the form fields that appear for each of the four additional options for Mesh Control Type. The mesh descriptions for the Lifting Surface (see pages page 34 to page 38) apply here as well.



The PCL functions are:

1. ArcCosine
2. Cosine
3. Inverse Square Root
4. Square Root
5. Squared

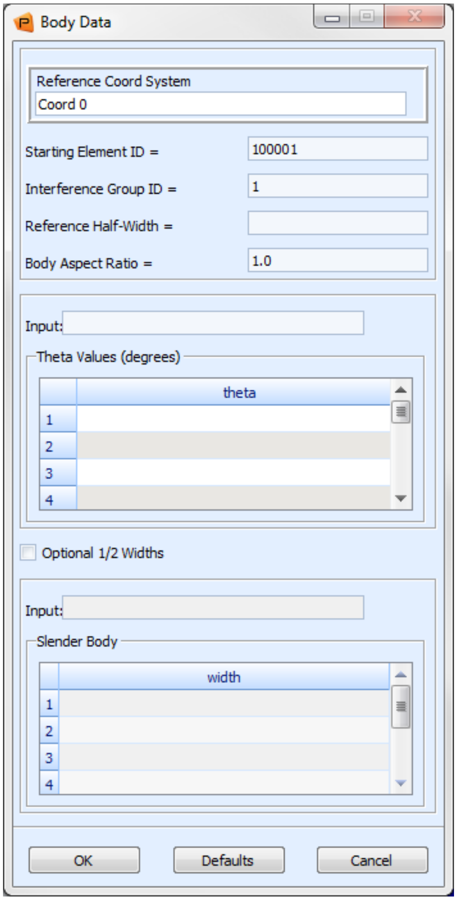
Mesh Control Subform (continued)


RHS Window
Flat Plate Aero Modeling
Mesh Control
Slender Mesh
One Way Bias
L1
L2
Number =
5
L2/L1 =
0.2
Interference Mesh =
Slender Mesh
OK
Cancel
RHS Window
Flat Plate Aero Modeling
Mesh Control
Slender Mesh
Two Way Bias
L1
L2
L1
Number =
5
L2/L1 =
0.2
Interference Mesh =
Slender Mesh
OK
Cancel
RHS Window
Flat Plate Aero Modeling
Mesh Control
Slender Mesh
PCL Function
Number of Nodes
5
Select Function
Arc Cosine
Cosine
Inverse Square Root
Square Root
Squared
Interference Mesh =
Slender Mesh
OK
Cancel
RHS Window
Flat Plate Aero Modeling
Mesh Control
Slender Mesh
Tabular
Input Data
Parametric Value
1
2
3
4
5
6
7
Clear
Sort
Reverse
Interference Mesh =
Slender Mesh
OK
Cancel

Note: The Tabular spreadsheet is initially set to 25 rows. This can be expanded by the user as required, just continue to add data.

Additional Body Data

This subform requests the user to provide Additional Data required for successful Body creation. Optional data on this form is either loaded with default data values or explicitly entered. The optional 1/2 width values should be used to produce body forces (due to a change in width).

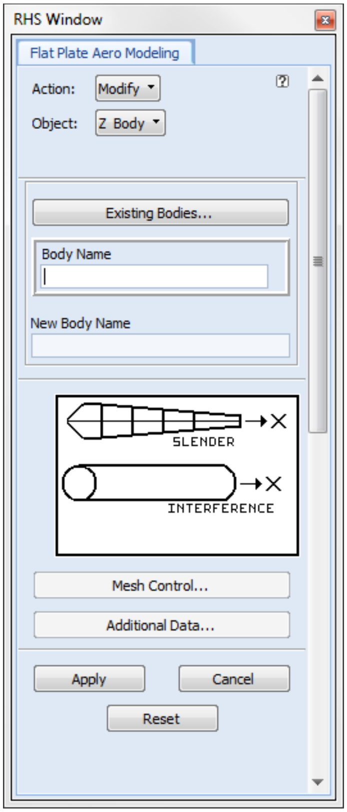


Parameter	Description
Reference Coord System	The Reference Coord System can be used to describe the Aero Body geometry.
Starting Element ID	Body starting node and element ID. This value is obtained from the Aero Modeling Options form.
Interference Group ID	Interference Group ID = current IGID specified on Flat Plate Aero Options form.

Parameter	Description
Reference Half-Width	Body Reference Half-Width is required and initially blank.
Body Aspect Ratio	Body Aspect Ratio (height/width) allows for elliptical definition. The default value is =1.0 (circular).
Theta Values (degrees)	Table is for the Interference Body Theta Values (angles). At least one value must be defined.
Slender Body Width	Table allows the user to optionally define the Slender Body Half Widths. A value must be supplied for each slender body division point. The table is N+1 in size where N is the number of slender bodies created from the mesh operation.

Modify/Body

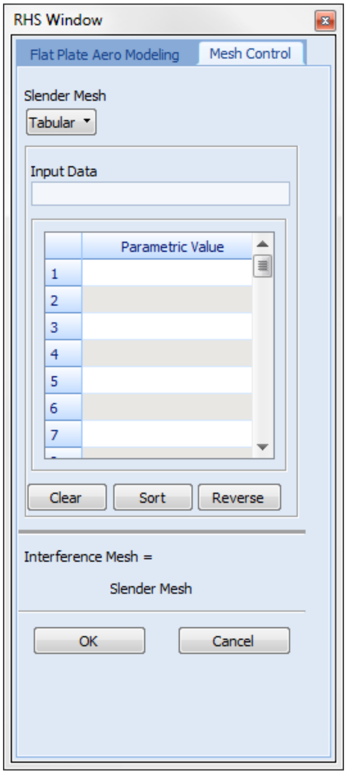
This form allows the user to Modify existing Body definitions and is applicable for all bodies (Z, Y, ZY). The Mesh Control and Additional Data buttons remain disabled until an existing Body is selected. These buttons are then enabled, and the Existing Bodies ellipsis button is disabled. The initial mesh distribution and corresponding half widths may be modified. If the number of Slender Body elements is modified, the original starting ID is used if possible.



Parameter	Description
Mesh Control	Body mesh control is initially set to Tabular; however, any mesh distribution technique may be used.
Additional Data	The Additional Data subform that was completed during Body creation displays when the user selects this button. Any data may be modified.

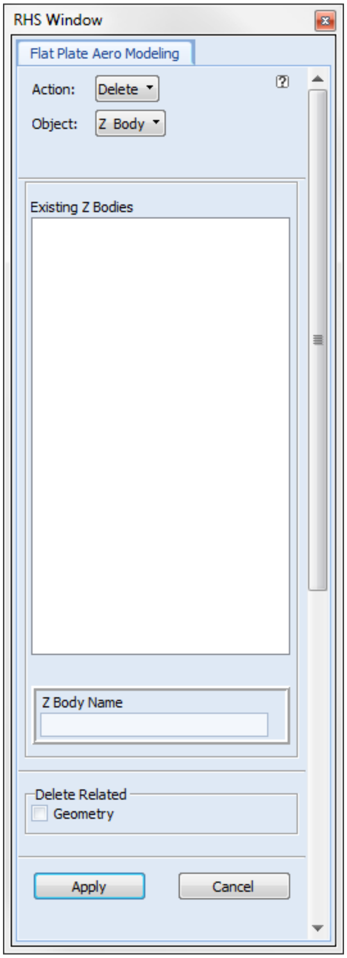
Mesh Control

The initial Modify state is set to allow changes in the Tabular form for the Slender Body. However, all mesh control options are available.



Delete/Body

This form allows the user to select which Bodies are to be deleted. This form is exactly the same for the (Z,Y,ZY) Bodies except that the indicated labels change to Reflect the shown Object type.

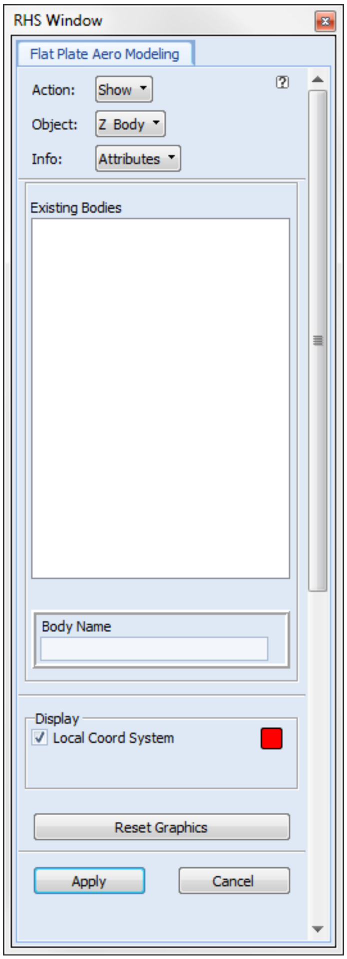


Parameter	Description
Existing Bodies	Select Bodies to delete by selecting from the Existing Bodies list, or (see next)
Body Name	Place the cursor in the Z Body Name databox, then graphically select the body. If multiple bodies are graphically selected, then the cycle pick menu displays.

Note:	
	1. Label changes to reflect the body type. (In this case Body Z)
	2. Delete Related Geometry is set to OFF by Default.

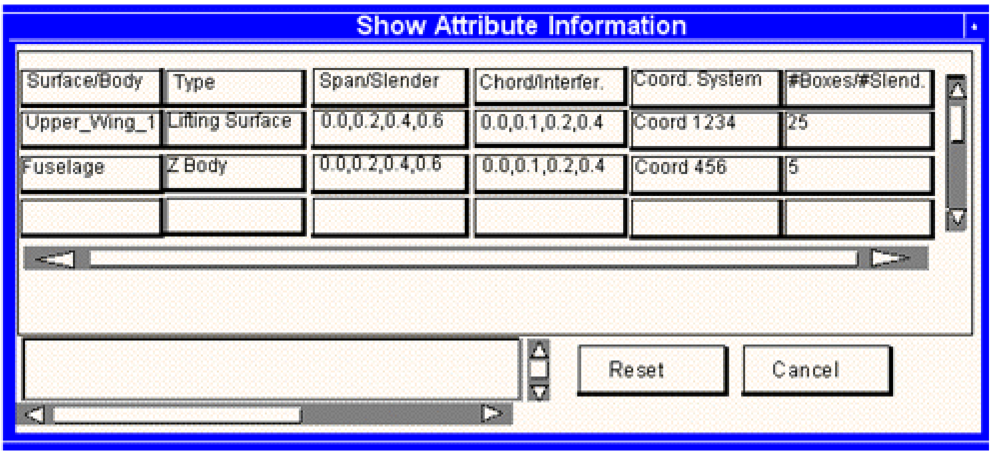
Show/Body/Attributes

This form allows the user to display Body Attributes (type, slender and interference parametric mesh locations, and Reference Coordinate System). The display for Show/Attributes is shown on the following page.



Parameter	Description
Existing Bodies	Identify surfaces to show by selecting from the Existing Bodies list, or (see next)
Body Name	Place the cursor in the Body Name databox, then graphically select the surface.
Local Coord System	Displays the positive direction (C1) for each aero element.
Apply	Take the selected Body Name and display it and its associated data in the appropriate form.

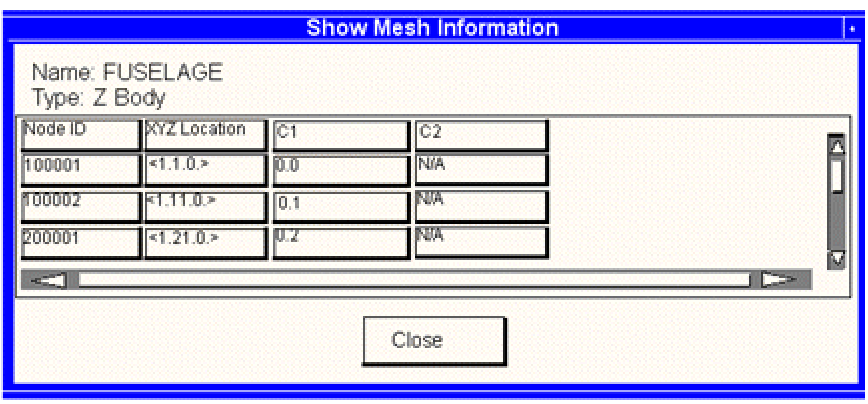
This subform is displayed when the Apply button on the Show/Body/Attributes form is selected. Note that just like Lifting Surfaces the data is appended to the information stored in the form from previous viewings until the Reset button is selected.



Show/Body/Mesh

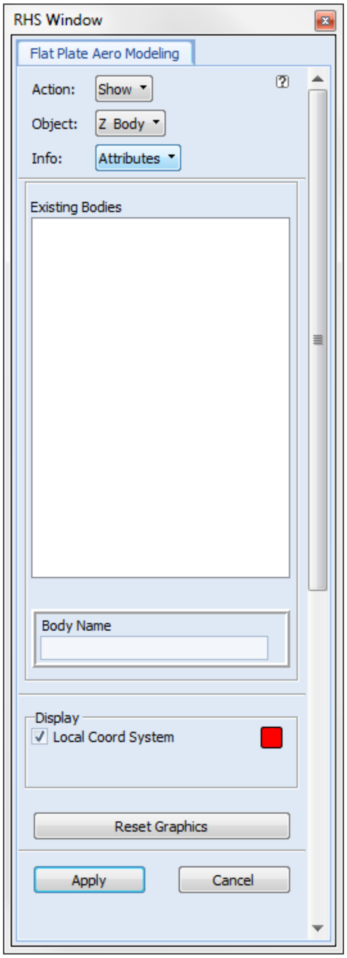
This form (not shown), looks just like the Show/Body/Attributes form. This form is used to display the Mesh Information (node points) for the selected bodies.

The following Subform is displayed when the Apply button on the Show/Body/Mesh form is selected.



Show/Body/Interference

This form allows the user to graphically display the Interference Body as a 3D-entity. The Interface Body is shown with markers that are plotted along the constant theta positions.



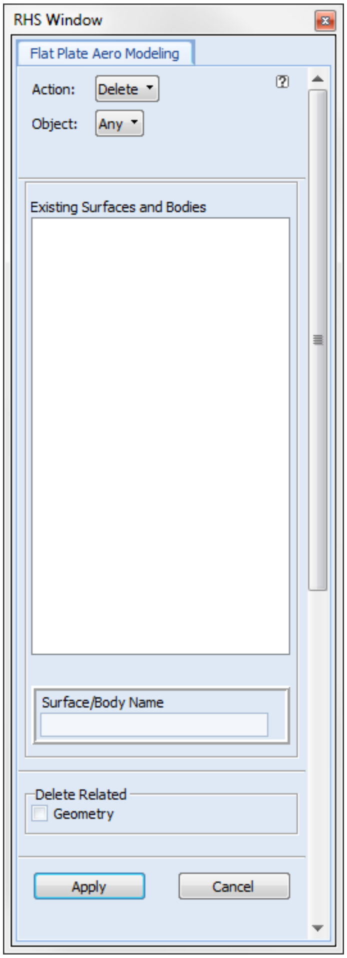
Parameter	Description
Existing Bodies	Select bodies to display by selecting from the Existing Bodies list.

Show/Body/Slender

This form (not shown), is exactly the same as the Show/Body/Interface form and allows the user to graphically display the Slender Body as a 3D-entity.

Delete/Any

This form allows the user to select which Lifting Surfaces and bodies are to be deleted.

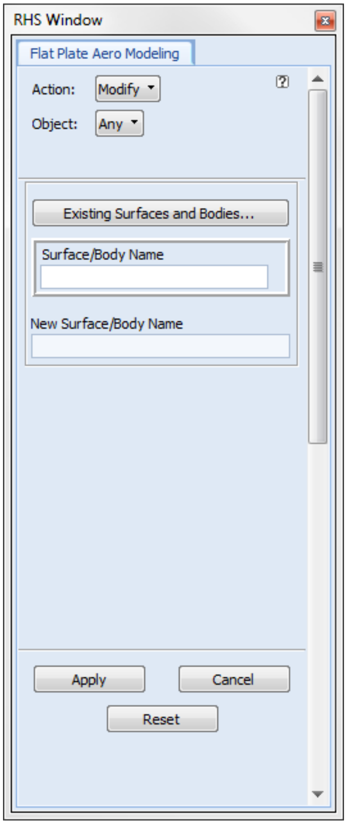


Parameter	Description
Existing Surfaces and Bodies	Identify surfaces to delete by selecting from the Existing list, or (see next)
Surface/Body Name	Place the cursor in the Surface Name databox, then graphically select the surface. If multiple surfaces are graphically selected, then the cycle pick menu displays.

Note: Delete Related Geometry is set to OFF by Default.

Modify/Any

The Modify/Any form is used to modify any existing Lifting Surface or Body.

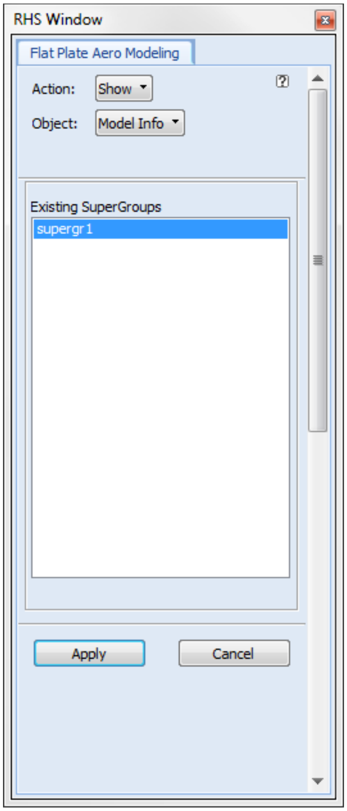


Parameter	Description
Object	Select Object = Any to list all Flat Plate Aero Modeling surfaces and bodies.
Existing Surfaces and Bodies	Identification can be by either selecting from the existing list, or (see next)
Surface/Body Name	Put focus in the New Surface Name select databox and then graphically selecting the surface. If multiple surfaces are graphically selected, then the cycle pick menu displays.

Note: The remainder of the form is completed after selection of a Flat Plate Aero surface or body.

Show/Model Info

This form allows the user to see the detailed aerodynamic model information for the selected SuperGroup.



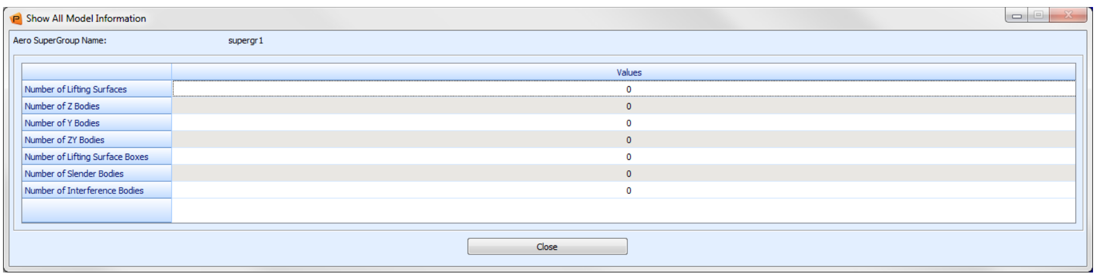
Parameter	Description
Existing SuperGroups	Select the SuperGroup for the Lifting Surface and Body information you wish to display.

Note: After pressing Apply, a spreadsheet is created detailing the aerodynamic model information. This new spreadsheet can be oriented in rows instead of the usual columns. This displays the number of lifting surface and bodies by type, total number of lifting surface boxes (summed over all lifting surfaces) and the total number of slender and interference bodies (summed over all bodies, slender and interference shown separately). This information is directly obtained from the database entries for lifting surfaces and bodies. The current SuperGroup is shown on the form including the AOM, a text description of the current aero SuperGroup and Apply and Cancel buttons.

Show/All Model Information

This spreadsheet displays the following data for the selected SuperGroup and includes:

- Number of Lifting Surfaces
- Number of Z Bodies
- Number of Y Bodies
- Number of ZY Bodies
- Number of Lifting Surface Boxes
- Number of Slender Bodies
- Number of Interface Bodies

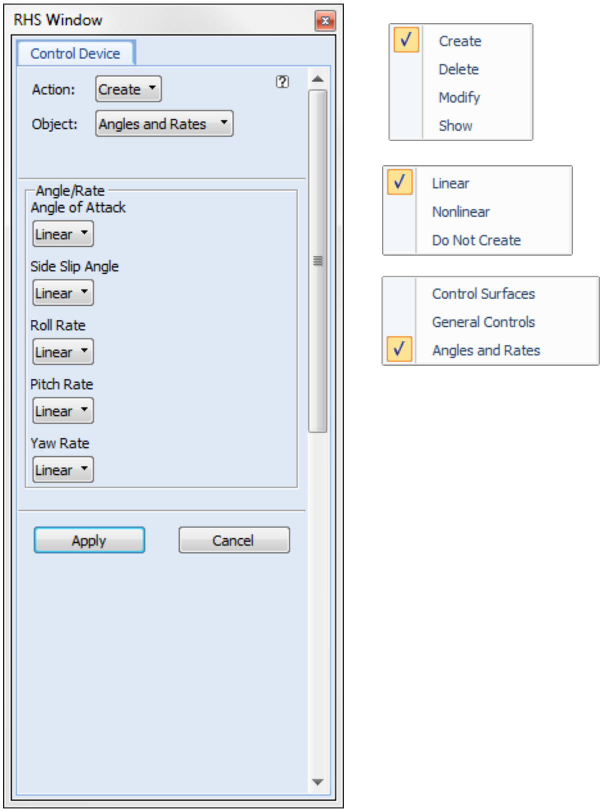


Control Device

Control devices are those components of an air vehicle that can be directly deflected to affect the trajectory of the vehicle. Examples include elevators, rudders, spoilers, and flaps. All of these are aerodynamic control surfaces. However, "Control Device" encompass a second set of parameters that have a more general definition: any parameter whose perturbation causes a change in an applied load. Examples from the more general set are the angle-of-attack and vertical acceleration. Typically these values result from pilot inputs to "control surfaces", but in the simulation we have access to these "control device" to simulate prescribed quasi-static maneuvers. Both kinds of devices are defined through this interface.

Overview

When the user selects Control Device, the first form that displays is Create/Angles and Rates. Shown adjacent to this form are **ALL** the different Actions, Objects, and Methods that can appear as part of the Control Device option:



Most of the Control Device forms are shown and annotated in the following pages, grouped by action as follows:

- Create
- Delete
- Modify
- Show

Definitions for Control Device Objects and Methods

Control Surface is an aerodynamic surface that is made up of some portion of one or more lifting surfaces.

Surface Boxes are elements of the lifting surfaces that were created in the Flat Plate Aero Modeling option.

Control Devices

Following is a list of the Flat Plate Aero Modeling forms and subforms related to Control Devices:

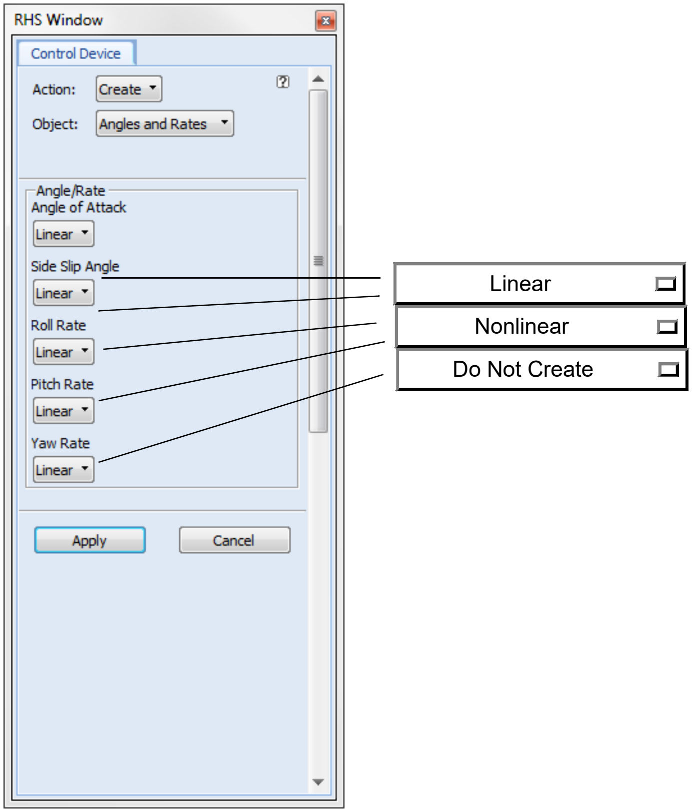
- Create/Angles and Rates

- Create/Control Surfaces/Linear
- Create/Control Surfaces/Nonlinear
- Create/Control Surfaces subforms
    - Existing Control Surfaces
    - Hinge Line, Reference Chord Length, and Reference Area
    - Optional Limits
- Create/General Controls
- Delete/Any
- Delete/Angles and Rates
- Delete/Control Surfaces
- Delete/General Controls
- Modify/Angles and Rates
- Modify/Control Surfaces/Linear
- Modify/Control Surfaces/Nonlinear
- Modify/General Controls
- Show/Control Surfaces/Attributes
    - Show Attribute Information
- Show/Any Controllers
    - Show Controller Information

Control Device Forms

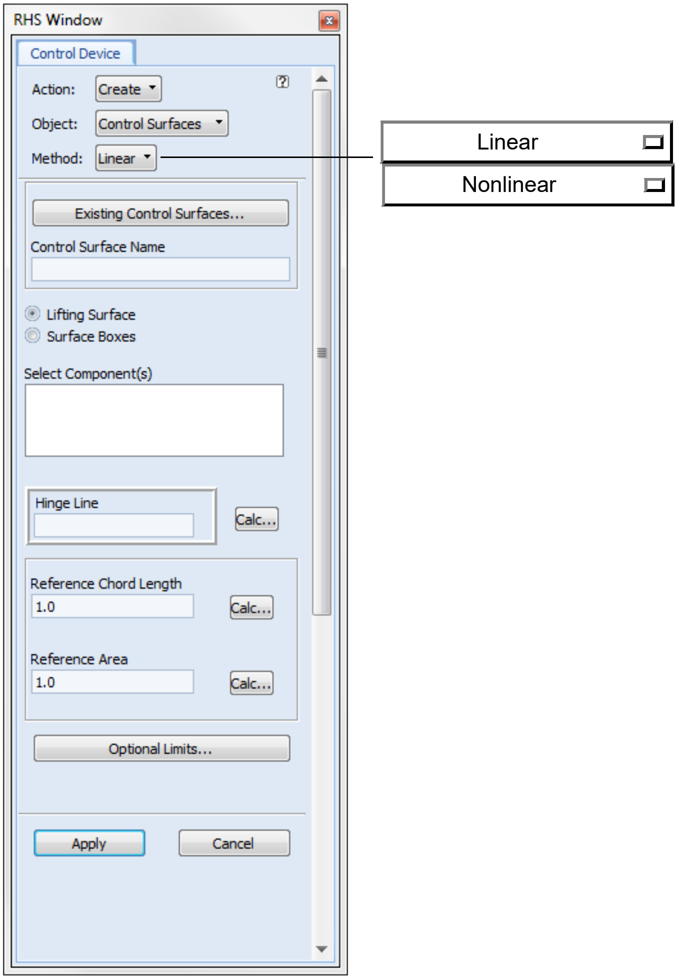
Create/Angle and Rates

An Angle and Rate is defined by setting it to Linear or Nonlinear. If the user does not want to create a particular angle and rate, then that option menu should be set to Do Not Create.



Create/Control Surfaces/Linear

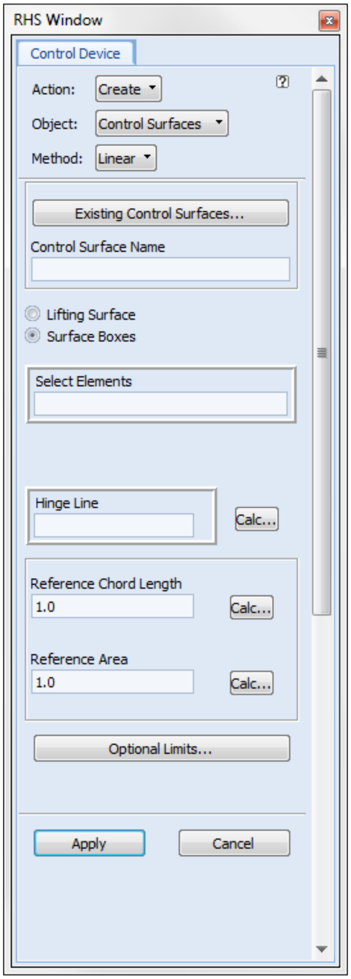
A Linear Control Surface can be defined by selecting all elements associated with one or more lifting surfaces. The user is also asked to provide a hinge line coordinate system (Y-axis must be along positive hinge axis), a Reference Chord Length and a Reference Area. The reference values are only used in the calculation of non-dimensional hinge moment coefficients and do not affect any other trim or loads calculations.



Parameter	Description
Existing Control Surfaces	Displays the Existing Control Surfaces.
Control Surface Name	Name of the new Control Surface.
Lifting Surface/Surface Boxes	Used to determine the method for entering the components/elements.

Parameter	Description
Select Component(s)	Select one or more lifting surfaces to define Control Surface.
Hinge Line	Coordinate system defining the Hinge Line. **Note:** The Y-axis must be along the positive hinge axis.The Calc button provides an easy method to create a hinge coordinate system.
Calc...	The user is required to define the Reference Chord Length and Reference Area. The Calc buttons provide easy length and area calculation tools.
Optional Limits	Optionally define hinge moment and position limits.

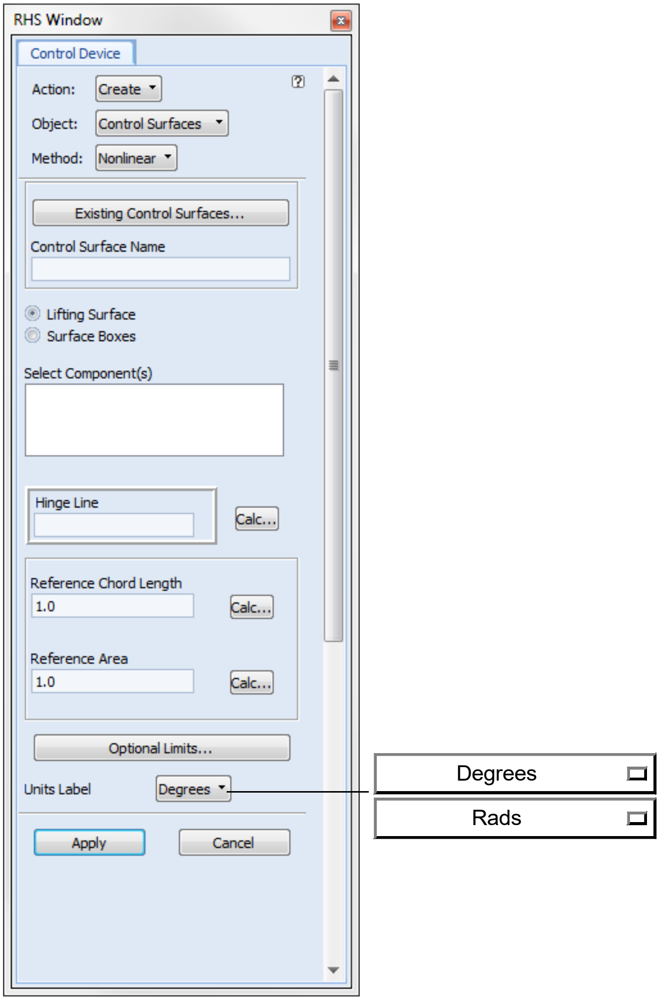
Linear Control Surfaces can also be defined using a set of selected elements which are not required to be connected. This form is similar to the one on the previous page and only those things that are different for element selection are described below.



Parameter	Description
Lifting Surface/Surface Boxes	Used to determine the method for entering the components/elements.
Select Elements	Select one or more lifting surfaces to define the Control Surface.

Create/Control Surfaces /Nonlinear

Nonlinear Control Surfaces are defined similar to the Linear Control Surfaces except that a Units Label must also be defined.

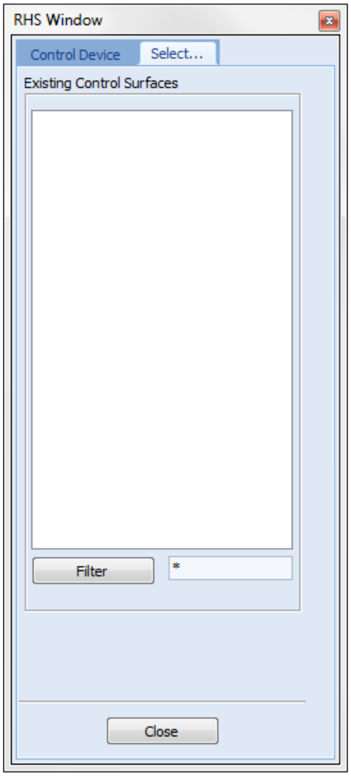

RHS Window
Control Device
Action: Create
Object: Control Surfaces
Method: Nonlinear
Existing Control Surfaces...
Control Surface Name
Lifting Surface
Surface Boxes
Select Component(s)
Hinge Line
Calc...
Reference Chord Length
1.0
Calc...
Reference Area
1.0
Calc...
Optional Limits...
Units Label
Degrees
Apply
Cancel
Degrees
Rads

Parameter	Description
Lifting Surface/Surface Boxes	Used to determine the method for entering the components/elements.
Select Component(s)	Only displayed when "Lifting Surface" is selected. If the "Surface Boxes" switch is selected than this listbox is replaced with a select databox called "Select Elements".
Hinge Line	Coordinate system defining the Hinge Line. **Note:** The Y-axis must be along the positive hinge axis.The Calc button provides an easy method to create a hinge coordinate system.
Calc...	The user is required to define the Reference Chord Length and Reference Area. The Calc buttons provide easy length and area calculation tools.
Optional Limits	Optionally define hinge moment and position limits.
Units Label	Used to determine the units for defining the data.

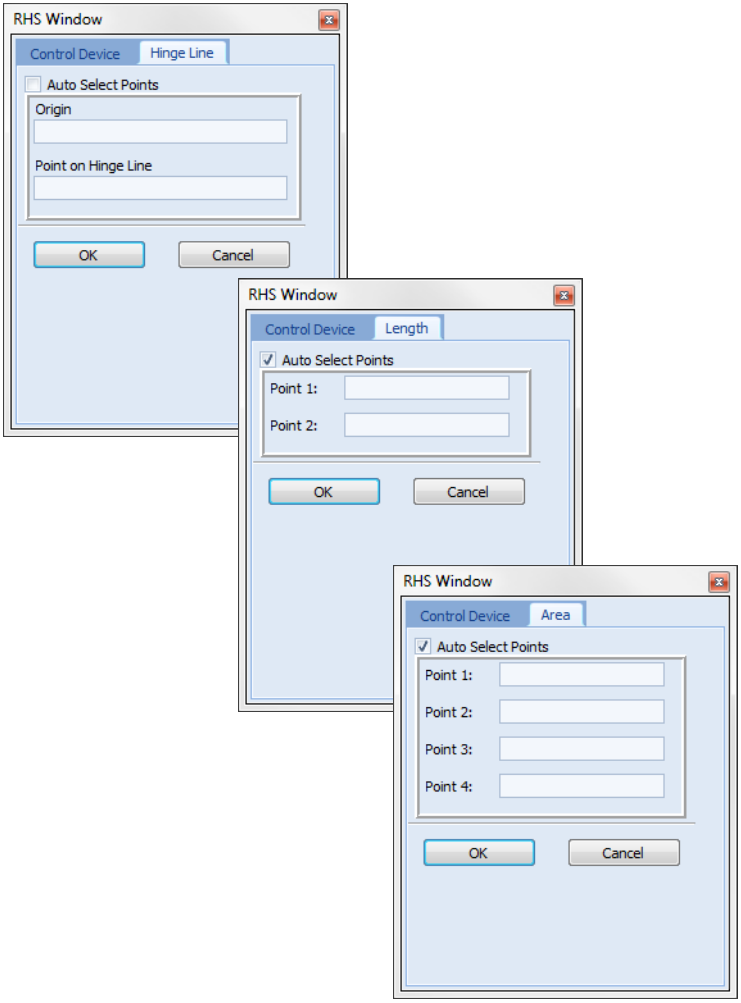
Create/Control Surfaces Subforms

Existing Control Surfaces

This form simply lists all the Existing Control Surfaces. Selection of a surface does *not* result in any information being transferred back to the Create/Control Surfaces form.


RHS Window
Control Device
Select...
Existing Control Surfaces
Filter
*
Close

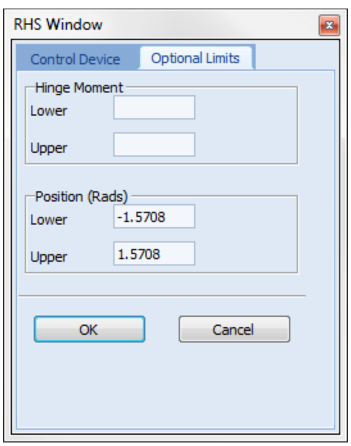
Hinge Line, Reference Chord Length, and Reference Area



Parameter	Description
Hinge Line	A Coordinate System whose Y-axis defines the positive hinge axis is required for Control Surface definition. This form provides a simple tool to create this Coordinate System. The user is only required to identify the hinge origin and positive axis.A Coordinate System whose Y-axis defines the positive hinge axis is required for Control Surface definition. This form provides a simple tool to create this Coordinate System. The user is only required to identify the hinge origin and positive axis.
Length/Area	These tools help the user calculate length and area dimensions for Control Surface definitions.

Limits: Hinge Moment and Position

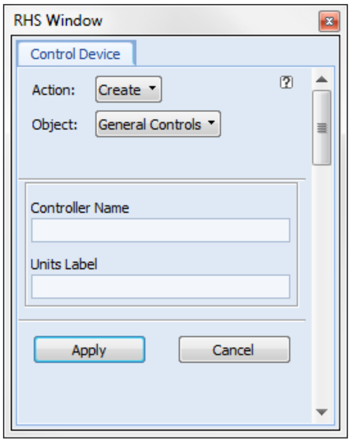
The Optional Limits subform allows the user to provide Control Surface Hinge Moment and Position limits. These limits are used to check for saturation of the Control Surface during an aeroelastic trim calculation.



Parameter	Description
Lower/Upper Hinge Moment	The lower and upper hinge moment limits are used during trim calculations. If left blank, the moments are unlimited.
Lower/Upper Position	This label is either Position (Rads) or Position (Degrees) depending on the state of the Dimensions (Angle and Rates toggle from the options form. Rads is the default.) The position limits are used during trim calculations. Default values of +90 and -90 degrees are provided.

Create/General Controls

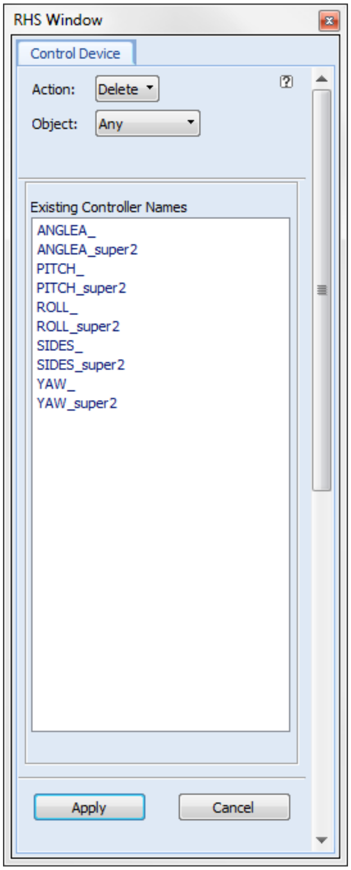
A General Control is defined by specifying a Controller Name and by entering the Units Label.



Note: Both 'Controller Name' and 'Units Label' databoxes allow for a maximum of 8 characters.

Delete/Any

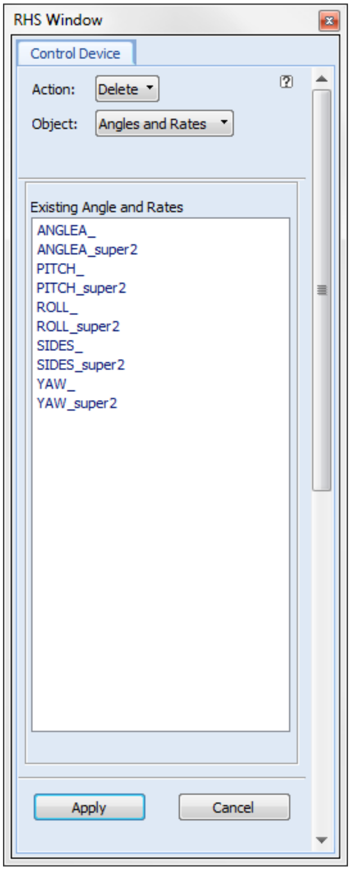
This form allows the user to delete any Control Device from the database.



Parameter	Description
Existing Controller Names	Displays the different types of control devices that were created.

Delete/Angles and Rates

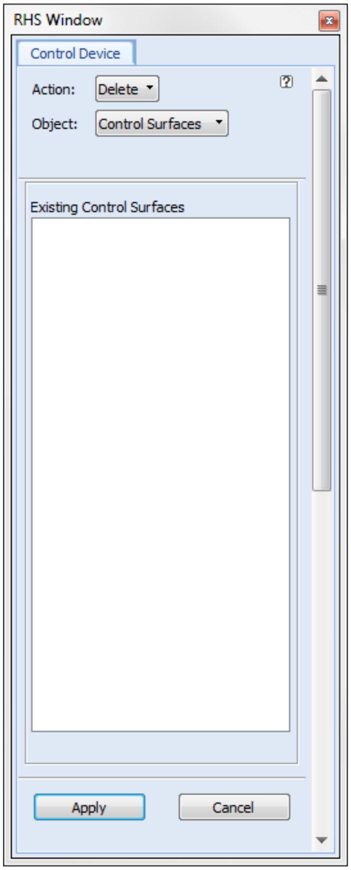
This form allows the user to delete Angles and Rates from the database.



Parameter	Description
Existing Angles and Rates	Displays only those Angles and Rates that were created. Deleting one is the equivalent of selecting "Do Not Create" on the Create form for this Object type.

Delete/Control Surfaces

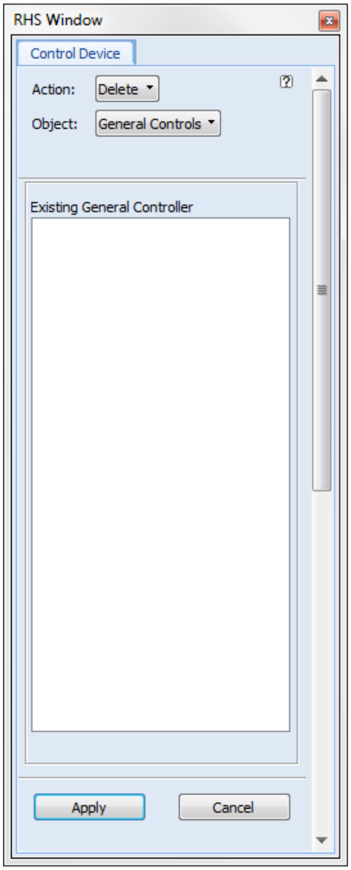
This form allows the user to delete linear and nonlinear control surfaces from the database.



Parameter	Description
Existing Control Surfaces	Displays only the Control Surfaces that were created.

Delete/General Controls

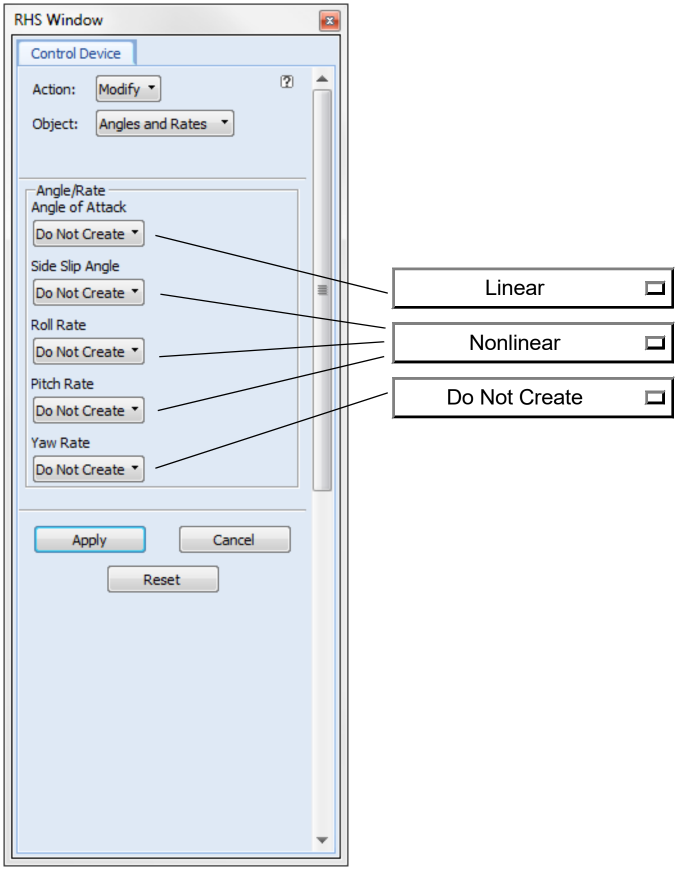
This form allows the user to delete general controls from the database.



Parameter	Description
Existing General Controllers	Displays only those General Controls that were created.

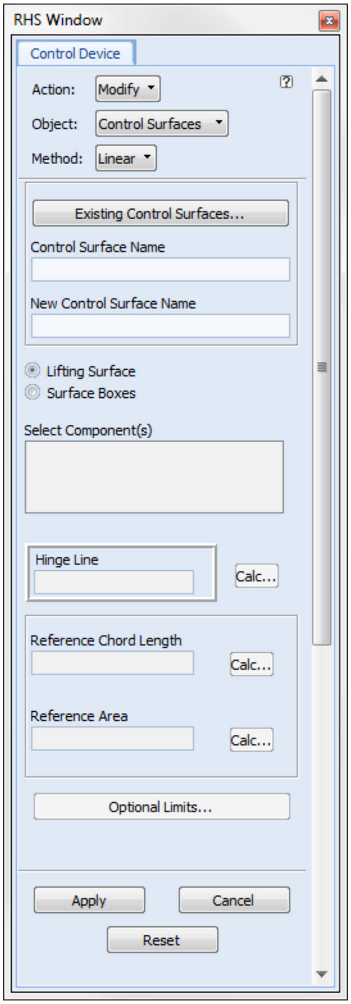
Modify/Angles and Rates

Users can modify existing Angles and Rates by setting them to be Linear, Nonlinear, or Do Not Create.



Modify/Control Surfaces/Linear

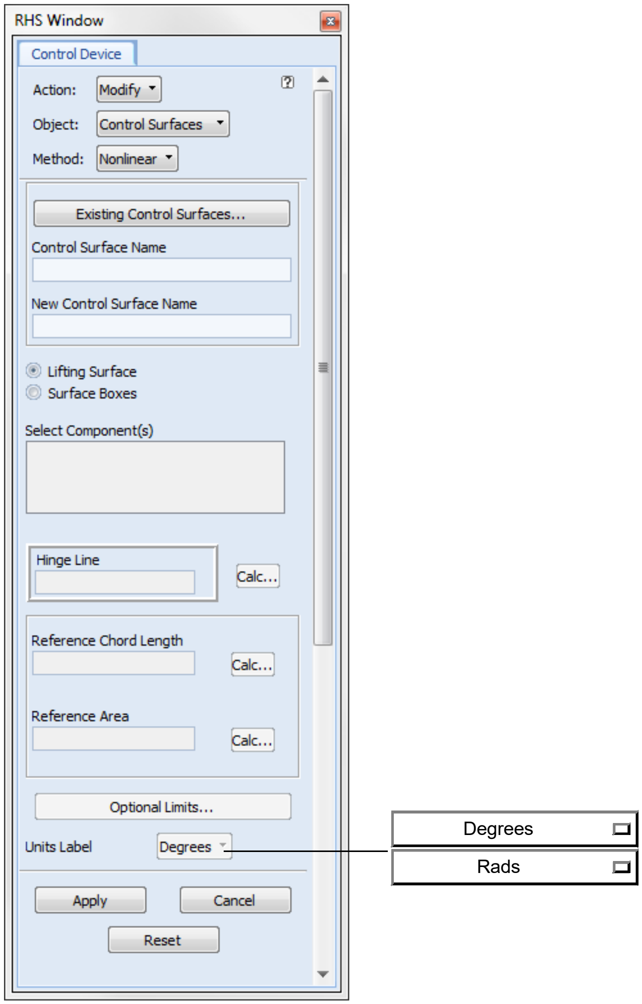
Users can modify existing linear control surfaces. The Linear Control Surface element region can be redefined by selecting one or more existing Lifting Surfaces or selecting individual elements.



Parameter	Description
Existing Control Surfaces	Select a Control Surface for modification.
Control Surface Name	The original Control Surface name is displayed on top and can be modified under **New Control Surface Name**.
Lifting Surface/Surface Boxes	Either way (Lifting Surface or Surface Boxes) can be used to modify the linear control surface region.
Select Elements	This changes similar to Create based on the selected method.

Modify/Control Surfaces/Nonlinear

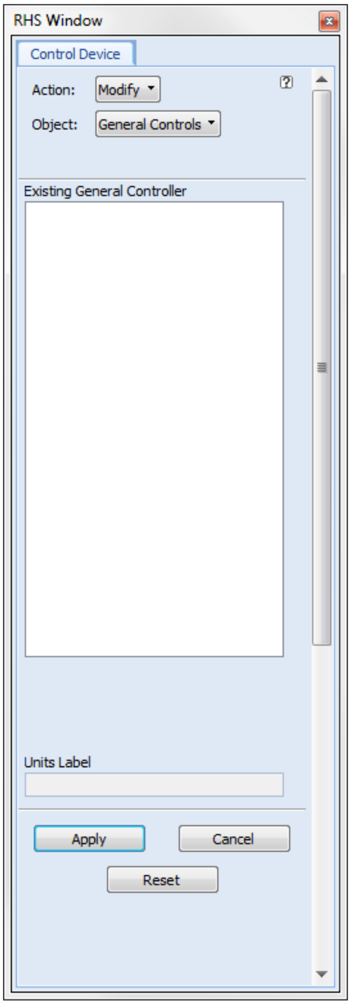
Users can modify existing nonlinear control surfaces the same way they modified linear control surfaces.



Parameter	Description
Lifting Surface/Surface Boxes	Used to determine the method for entering the components/elements.
Select Component(s)	Only displayed when “Lifting Surface” switch is selected. If the “Surface Boxes” switch is selected than this listbox is replaced with a select databox called “Select Elements”. See the Create form for details.
Units Label	Used to determine the units for defining the data.

Modify/General Controls

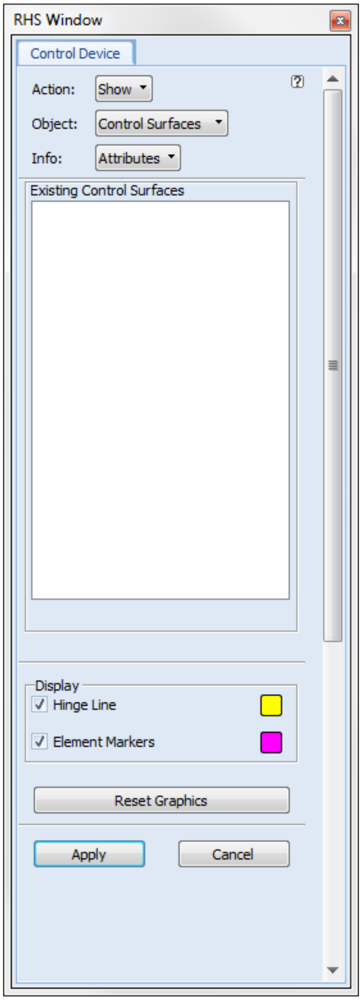
Users can modify the units label assigned to existing General Controls.



Parameter	Description
Existing General Controllers	Displays only those General Controls that were created.
Units Label	Allows for a maximum of 8 characters.

Show/Control Surfaces/Attributes

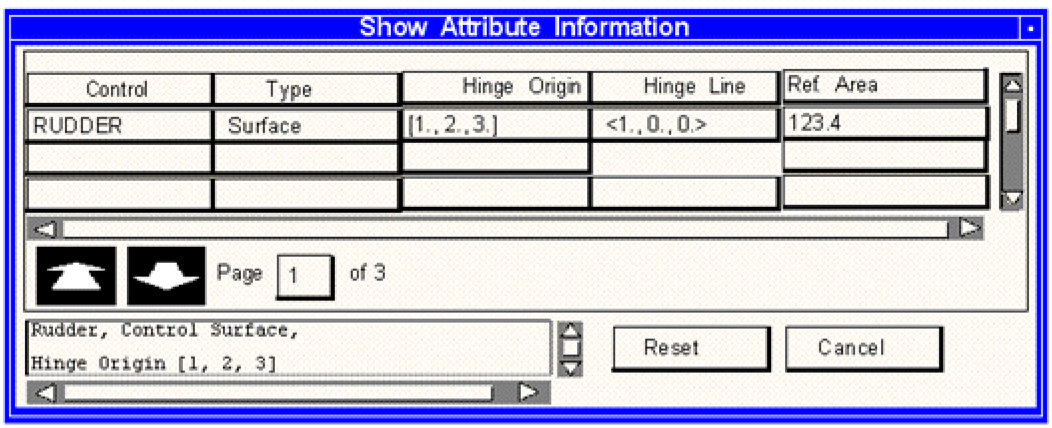
This form allows the user to show attribute information about the existing Control Surface (linear and nonlinear). Select Apply to display the Show Attribute Information subforms as shown on the following page.

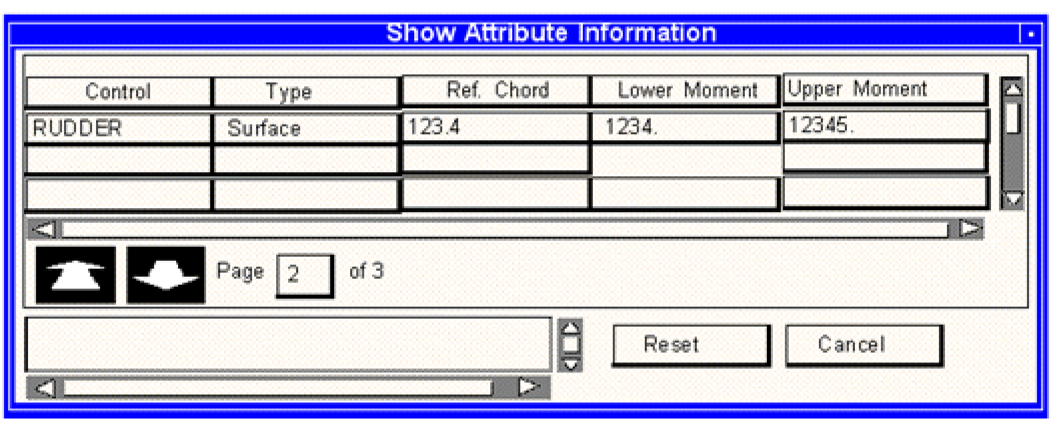


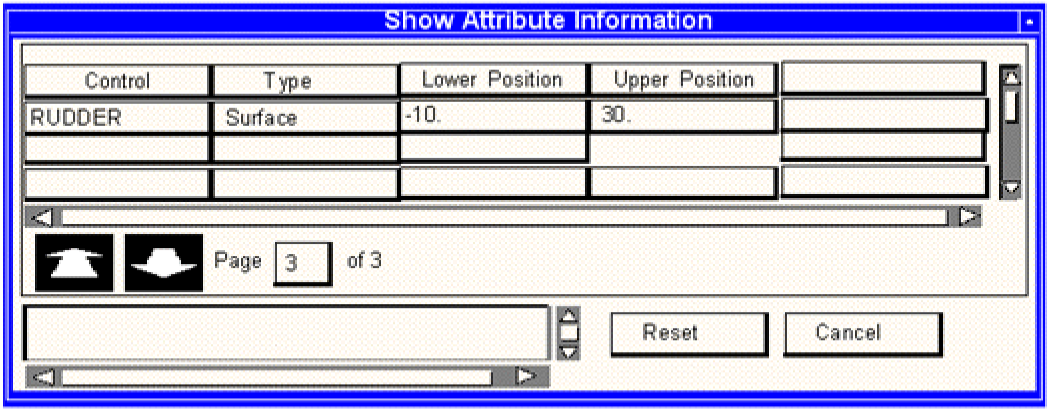
Parameter	Description
Existing Control Surfaces	Control Surfaces to show information. Lists both the linear and nonlinear control surfaces.
Hinge Line	The hinge line can be optionally displayed for each Control Surface as a vector drawn at the hinge origin.
Element Markers	The names of each Control Surface can be optionally drawn on each associated element.

Show Attribute Information

Three tables of attribute information appear for the selected Control Surface(s). Selecting any cell in the spreadsheet displays Control Surface information in the databox.

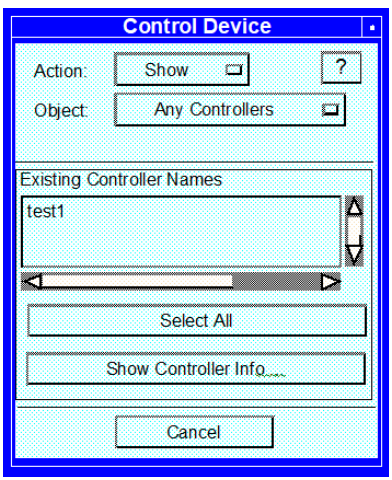






Show/Any Controllers

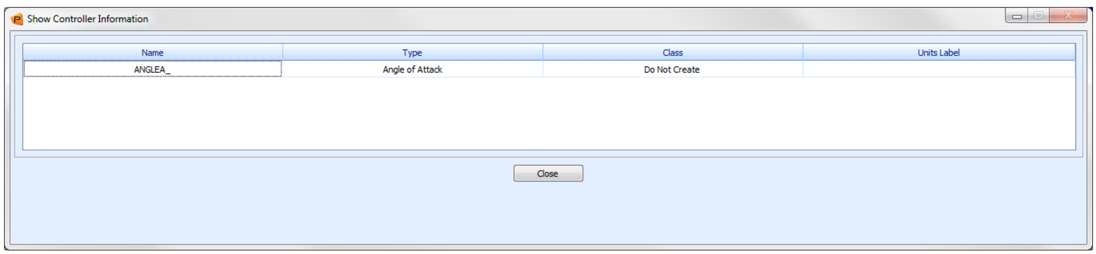
This form allows users to show Controller Information about the existing control devices. Select Apply to display the Show Controllers Info subform as shown on the following page.



Parameter	Description
Existing Controller Names	Contains all existing "Angles and Rates", "Control Surfaces" and "General Controls".
Select All	Selects all items in the listbox.
Show Controller Info	Displays the Show Controller Info subform.

Show/Any Controllers/Show Controller Info

This spreadsheet shows the name, type, class, and unit label for all the selected control devices.



Parameter	Description
Name	Contains the names of the selected Controllers.
Type	Contains the type "Angles and Rates", "Control Surfaces", or "General Controls".
Class	Contains either "Linear", or "Nonlinear".
Units Label	Contains either the input value or the table value.

Aero Monitor Points

Aero Monitor Points are used to represent summations of forces over certain regions of the Aerodynamic finite element (FE) mesh about a selected point. These Monitor points can then be combined to represent the integrated trimmed loads appropriate for critical loads survey.

Overview

When the user selects Aero Monitor Points, the first form that displays is Create/Monitor Points. Shown adjacent to this form are **ALL** the different Actions that are available when using this option.

Definitions for Aero Monitor Points

Aero Monitor Points are made of aero elements or components.

Most of the Aero Monitor Points forms are shown and annotated in the following pages, grouped by Action as follows:

- Create
- Delete
- Modify
- Show

Aero Monitor Points

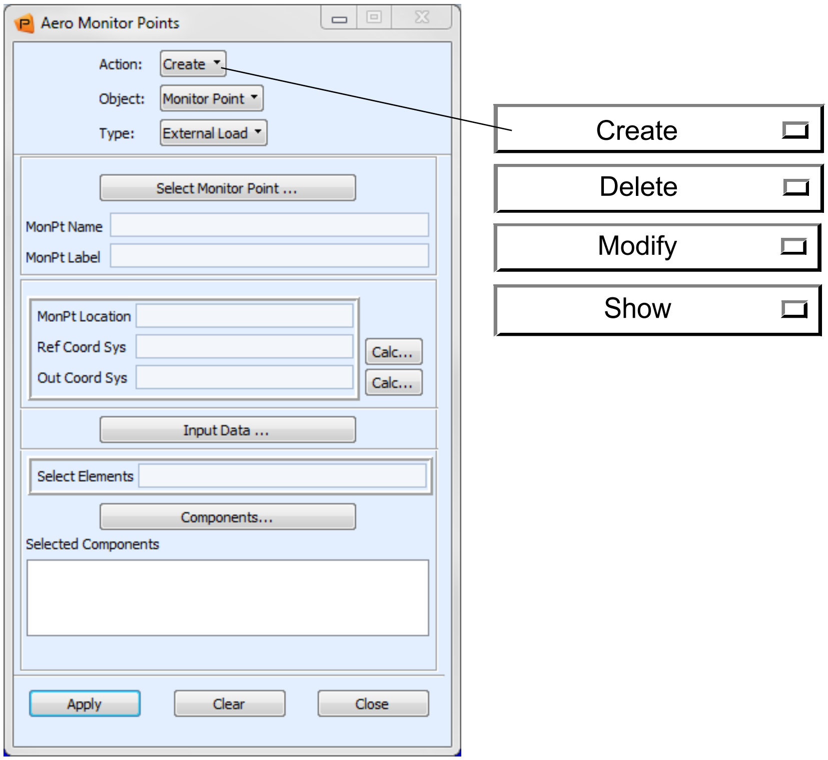
Following is a list of the Aero Monitor Points forms and subforms.

- Create/Monitor Points
  - Aero Monitor Point Definition
- Delete/Monitor Points
- Modify/Monitor Points
- Show/Monitor Points
  - Aero Monitor Point Info

To complete the forms, the user would start by selecting values at the top of the form, choosing an action and then an object. The fields on the remainder of the form display depending on the selections.

Create/Monitor Points

An Aero Monitor Point is defined by selecting aeroelements. The user is also asked to provide a monitor point label, a reference coordinate system and select the monitor components.

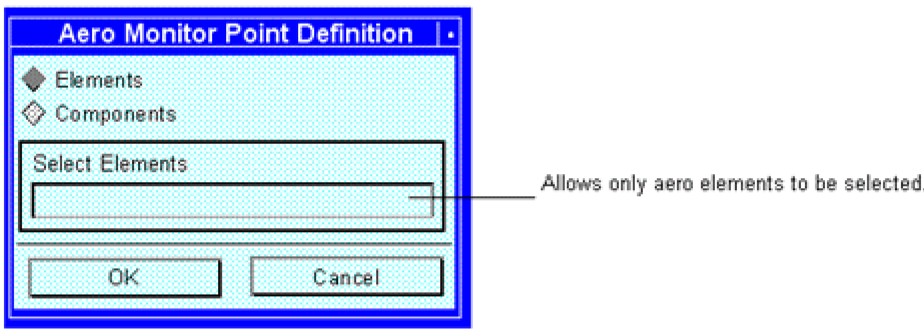


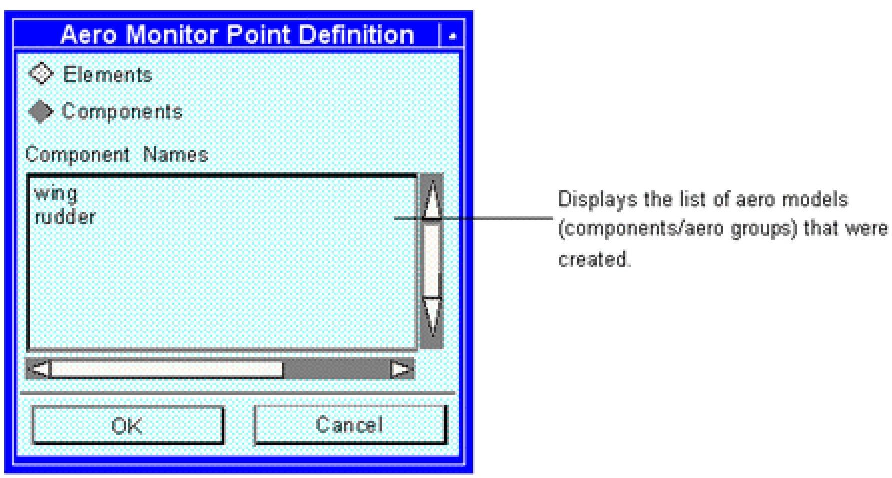
Parameter	Description
Existing Monitor Points	Displays the list of existing aero "Monitor Point Labels" and is here for information only.
Monitor Point Names	Maximum 8 characters.
Monitor Point Labels	Maximum 48 characters.
Monitor Point Definition	Displays the Monitor Point Definition subform
Calc...	Displays the Create - Coord form under Geometry and allows the user to create a new coordinate frame.
Ref Coord Frame	Allows selecting the Reference Coordinate frame value.
Monitor Components	By default all the toggles are selected. The user can select anywhere from 1 to 6 components.

Aero Monitor Point Definition

The Aero Monitor Point Definition subform allows the user to select either Aero elements or whole aero models.

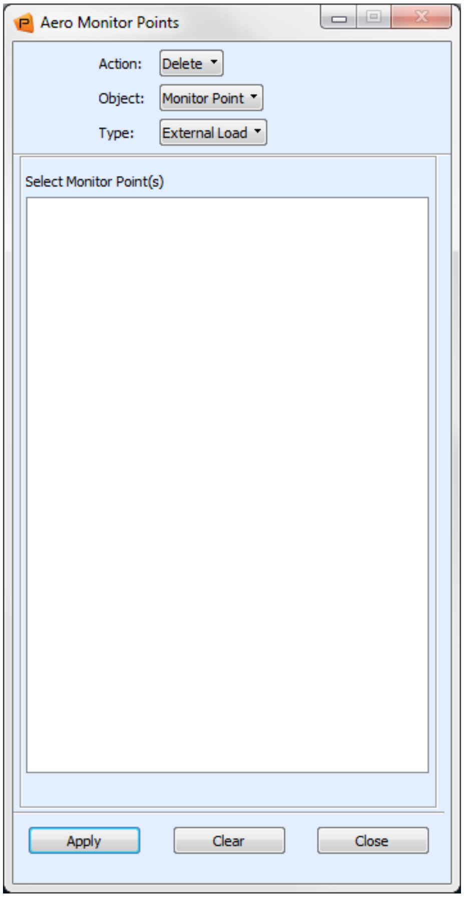
Note: The following two forms are also valid for the Modify action.





Delete/Monitor Points

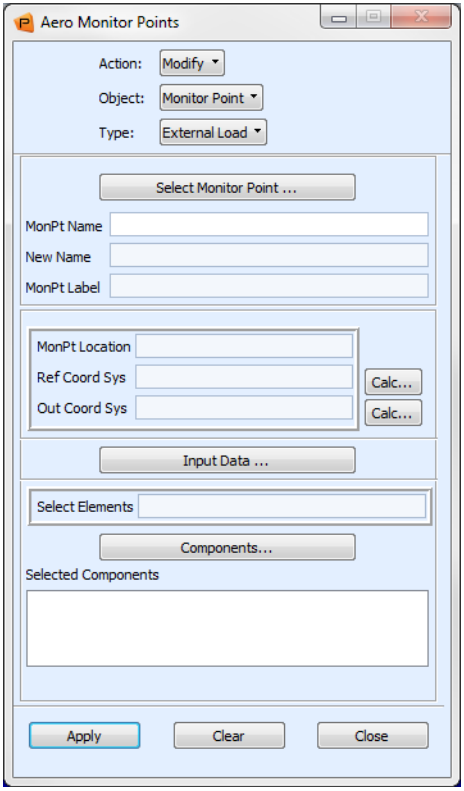
This form allows the user to delete Aero Monitor Points from the database.



Parameter	Description
Existing Monitor Points	Displays the list of existing aero "Monitor Point Labels" choices.

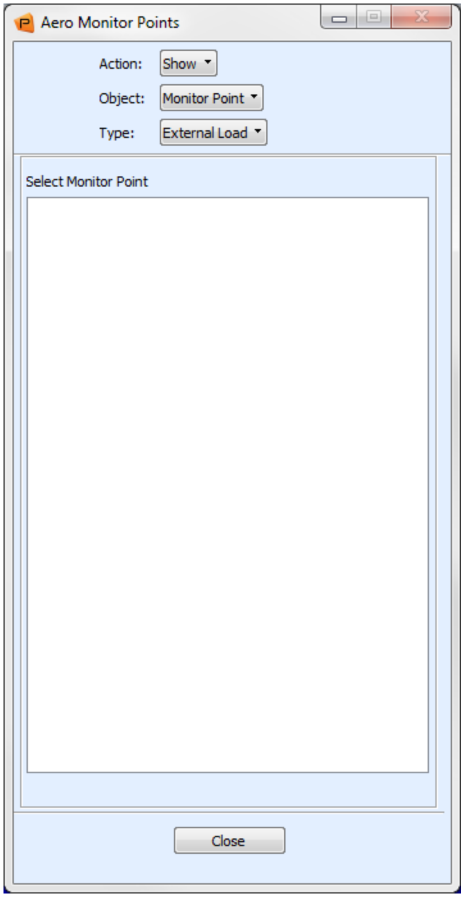
Modify/Monitor Points

Users can modify existing Monitor Points. The label, element region, coordinate frame, and Monitor Components can all be redefined.



Show/Monitor Points

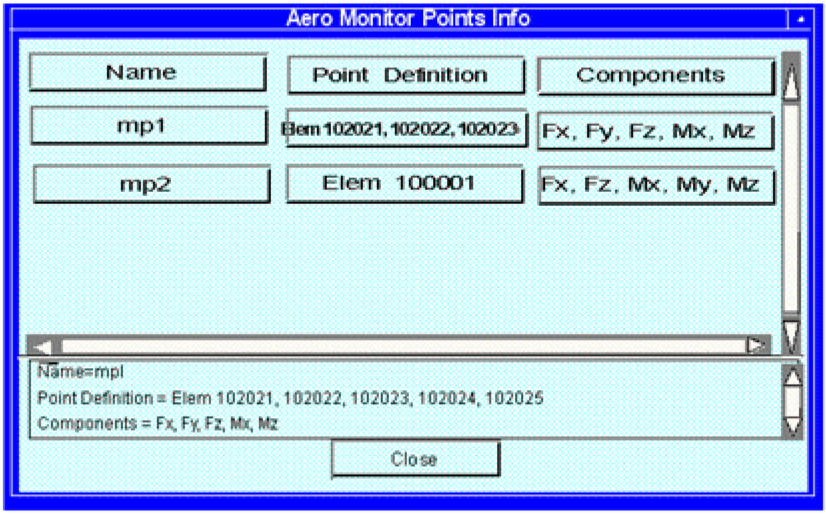
This form allows the user to look at the data used to define the selected Aero Monitor Points.



Parameter	Description
Existing Monitor Points	Displays the list of existing aero "Monitor Point Labels" choices.

Aero Monitor Point Info

This spreadsheet show the Name, Point Definition, and the Components for all the selected Aero Monitor Points.


Aero Monitor Points Info
Name
Point Definition
Components
mp1
Elem 102021, 102022, 102023
Fx, Fy, Fz, Mx, Mz
mp2
Elem 100001
Fx, Fz, Mx, My, Mz
Name=mpl
Point Definition = Elem 102021, 102022, 102023, 102024, 102025
Components = Fx, Fy, Fz, Mx, Mz
Close

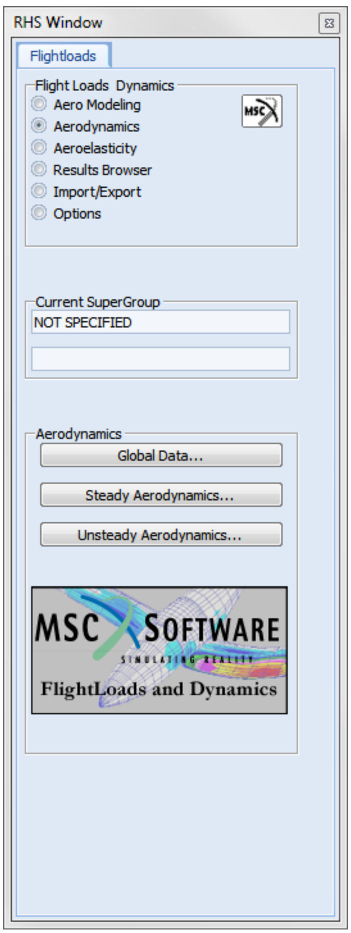
4 Aerodynamics

Introduction

The Aerodynamics module of MSC FlightLoads is divided into the following three application sections:

- Global Data
- Steady Aerodynamics
- Unsteady Aerodynamics

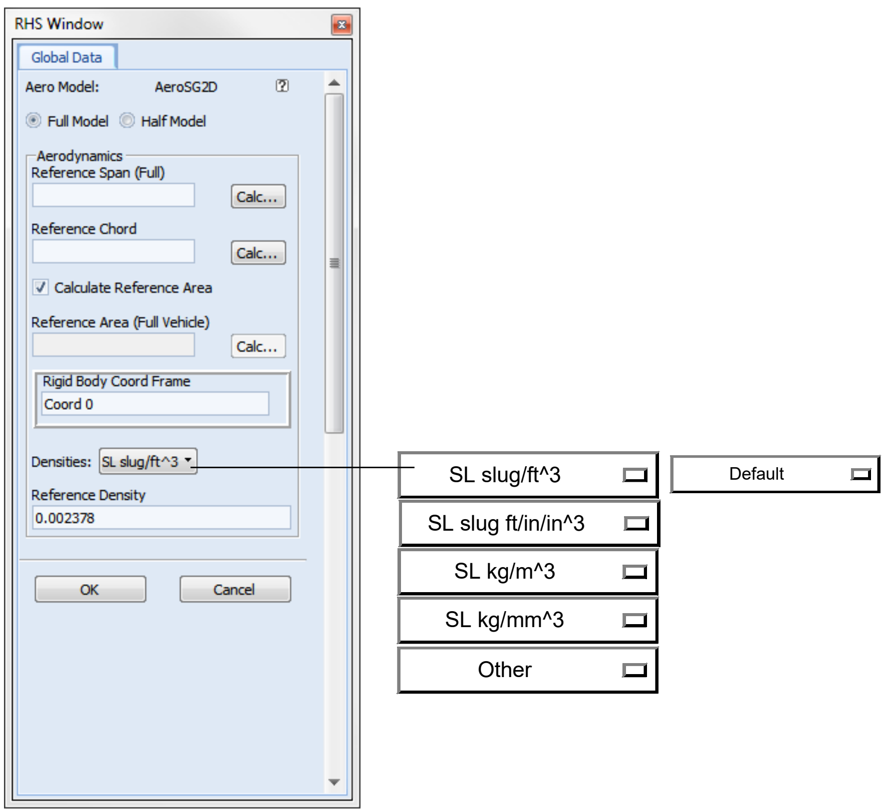
The following is an example of the initial form that is displayed when you choose the Aerodynamics option from the main MSC FlightLoads menu.



Parameter	Description
Aerodynamics	Three Aerodynamics application sections: ■ Global Data ■ Steady Aerodynamics ■ Unsteady Aerodynamics

Global Data

The following is the form that is displayed when you choose the Global Data option from the main MSC FlightLoads Aerodynamics menu.



Parameter	Description
Aero Model	Name of the current SuperGroup.
Reference Span (Full)	Wing Reference Span of full aircraft. The full span value should be input even with half-span models.
Reference Chord	Wing Reference Chord
Calculate Reference Area	If this toggle is depressed, the reference area is automatically computed as the product of the reference span and chord.
Reference Area (Full Vehicle)	Wing Reference Area For half span models a half-area value should be used.
Rigid Body Coord Frame	Aerodynamic Reference Coordinate System for Rigid body motions and for the calculation of stability derivative coefficients. The default is either the "Aerodynamic Flow Coordinate System", or the "Global Coordinate System".
Reference Density	The default Reference Density value changes based on the value chosen from the "Densities" option menu. (Zero is invalid, but any positive number is OK.) SL slug/ft^3 = 0.002378 SL slug ft/in/in^3 = 1.1468E-7 SL kg/m^3 = 1.226 SL kg/mm^3 = 1.226E-9

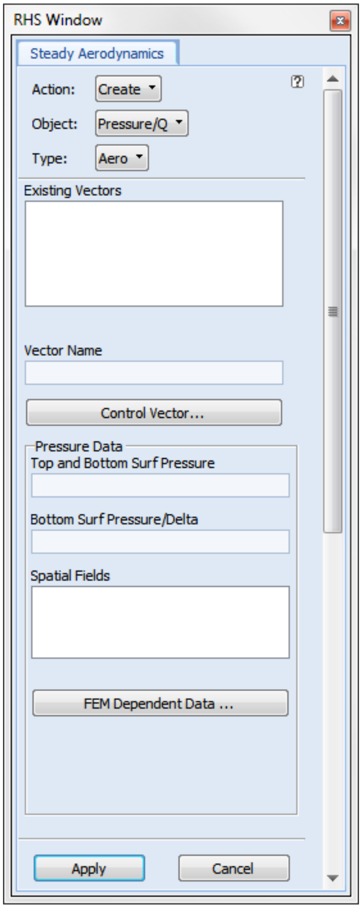
Steady Aerodynamics

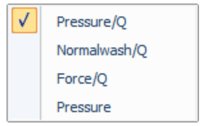
Steady Aerodynamics is used to define a Downwash (Normalwash, a Pressure, or an Aero Structural Force that is associated with a set of values of the vehicle's control vector. These "parametric" loads (that is, functions of the control setting) will then be elasticized and used to trim the airplane. These loads should represent total distributed load for their associated control positions - not incremental loads due to perturbations in the controller!

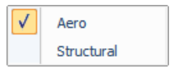
Overview

When the user selects Steady Aerodynamics, the first form that displays is Create/Pressure/Q. Shown adjacent to this form are all the different Actions and Objects that can appear as part of the Steady aerodynamics option:

The following is a portion of the initial form that is displayed when you choose the Steady Aerodynamics option from the main MSC FlightLoads Aerodynamics menu.







Note: When the Object is set to Force/Q an additional option menu called Type is displayed.The two values for this option menu are “structural” and “aero”.

Steady Aerodynamics

Most of the Steady Aerodynamic forms are shown and annotated in the following pages, grouped by Action as follows:

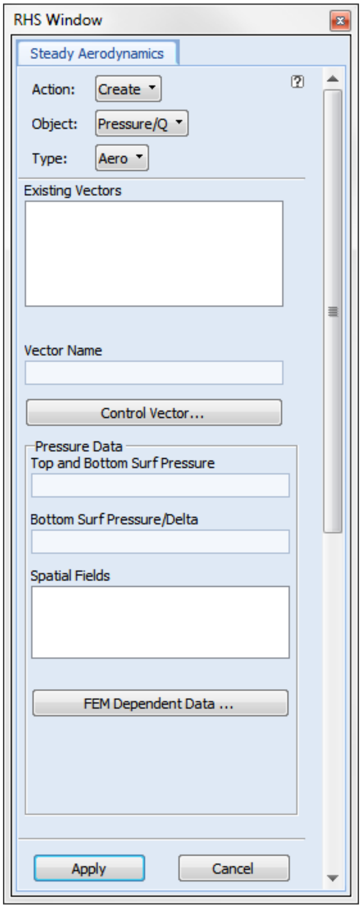
The following is a list of the forms included in the Steady Aerodynamics section:

- Create/Pressure/Q
- Create/Normalwash/Q
- Create/Force/Q

- Create Subforms
    - Create/Any/Control Vector
    - Create/Any/Fem Dependent Data
- Delete/Any
- Modify/Pressure/Q
- Modify/Normalwash/Q
- Modify/Force/Q
- Show/Any/Fringe
- Show Subforms
    - Show/Any/Fringe/Fringe Attributes

Create/Pressure/Q

Pressure/Q can be defined by setting the Control Vector Data, Load/BC Set Scale Factor, and by specifying a field for the top and bottom pressure or just one for the Delta Pressure. For 3D SuperGroups the user is asked to supply a Nodal Pressure instead.



Parameter	Description
Existing Vectors	Sorts the list of existing vector names by object type. Only those of the current object type should be displayed in the listbox. **Note:** The user is able to select a single vector name from the listbox and modify but they must work through the message dialog before the change is applied or they need to change the vector name.
Control Vector	**Note:** Subform displayed by this button is valid for all the objects (Pressure/Q, Normalwash/Q, and Force/Q).
Load/BC Set Scale Factor	Initial Scale factor to be used in defining the Spatial Fields.

Parameter	Description
Top and Bottom Surf Pressure	Name of a field defining the Top Pressure. This value can be left blank when defining a Delta
Bottom Surf Pressure/Delta	The name of a field defining the Bottom Pressure or the Delta. **Note:** Previous two widgets are replaced with the Nodal Pressure databox for 3D SuperGroups.
Spatial Fields	Lists of all the created Spatial Fields.
FEM Dependent Data	Becomes active when the user puts the cursor in the "Top and Bottom Surf Pressure" or the "Bottom Surf Pressure/Delta" databox. This subform allows the users to define a new field.

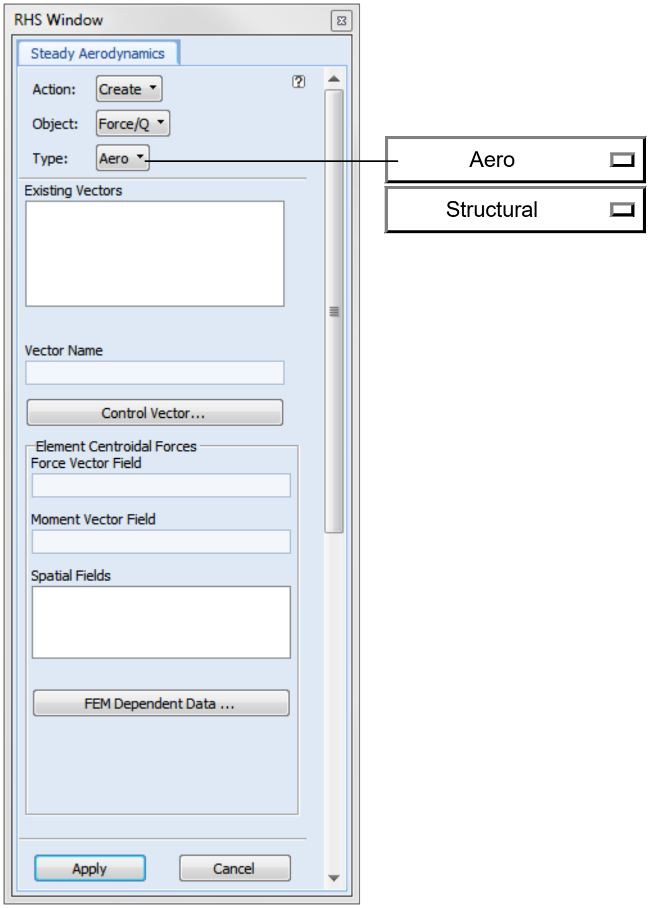
Create/Normalwash/Q

Normalwash/Q can be defined by setting the Control Vector Data, Load/BC Set Scale Factor, and by specifying a field for the normalwash.

Note: This Object is not available when the current SuperGroup is of Type 3D.

Create/Force/Q

Both the structural and the Aero Force/Q are defined by setting the Control Vector Data, the Load/BC Set Scale Factor, and by specifying a field for the Force Vector and one for the Moment Vector.

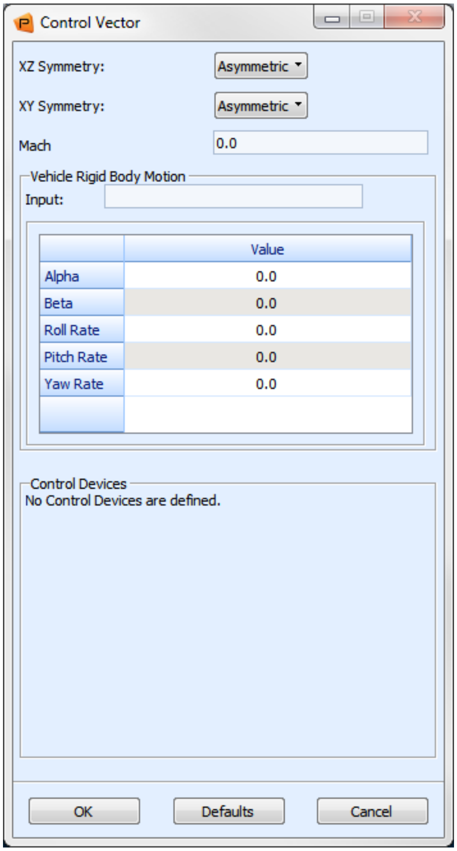

RHS Window
Steady Aerodynamics
Action: Create
Object: Force/Q
Type: Aero
Existing Vectors
Vector Name
Control Vector...
Element Centroidal Forces
Force Vector Field
Moment Vector Field
Spatial Fields
FEM Dependent Data ...
Apply
Cancel
Aero
Structural

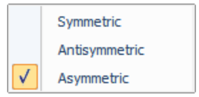
Parameter	Description
Force Vector Field	Name of a field defining the Force Vector.
Moment Vector Field	Name of a field defining the Moment Vector.
Spatial Fields	List of all the created Spatial Fields.
FEM Dependent Data	Becomes active when the user puts the cursor in the "Force Vector Field" or the "Moment Vector Field" databox. This subform allows the users to define a new field.
Analysis Coordinate Frame	Analysis Coordinate Frame that is to be associated to the Force/Q.

Create/Any Subforms

Control Vector

This form is used to enter the information on the symmetry, mach, rigid body motions, and the Control Devices.



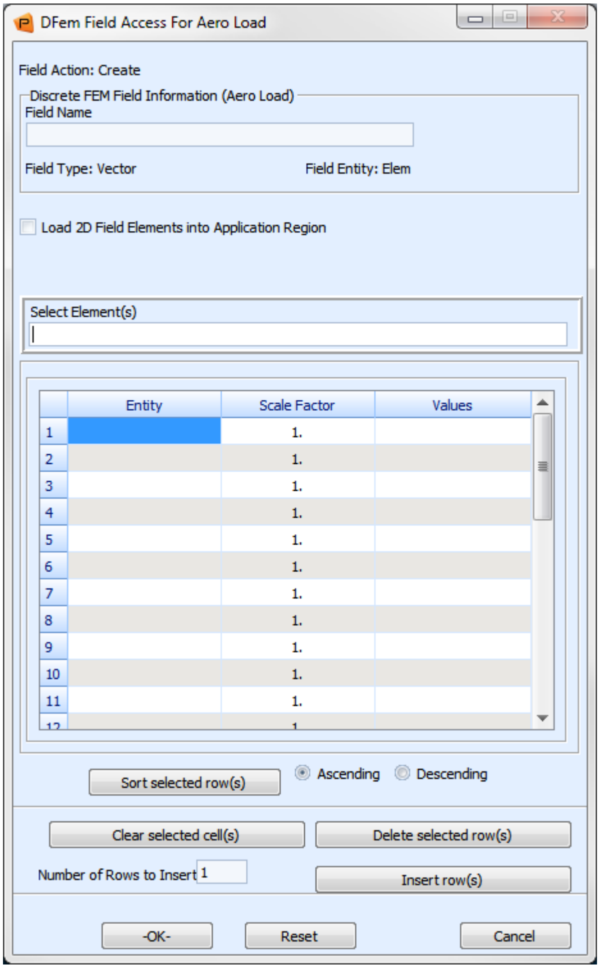


Parameter	Description
Vehicle Rigid Body Motion	**Note:** None of the Acceleration Rigid Body Motions are listed. Only those that are defined under Control Devices are listed.
Control Devices for AeroSG2D	The name of the current SuperGroup is displayed here in the title. **Note:** The default value for both the spreadsheets is 0.0.

Note: This form is valid for the Create version of the Control Vector form and for the Modify version of the Control Vector form.

FEM Dependent Data

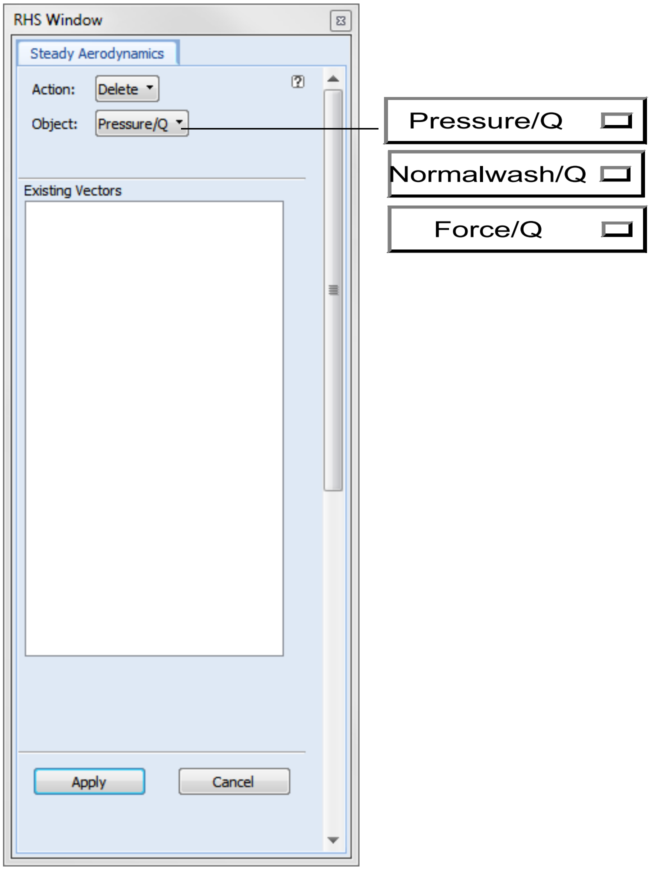
This form is used to define the Nodes/Elements and their values that make up the field that gets used for the pressure, normalwash, force and moments.



Note: This form changes based on which databox activated the button on the Steady Aerodynamics form. It can be used to enter either scalar or vector data and to select either nodes or elements.

Delete/Any

This form allows the user to delete existing Pressure/Q, Normalwash/Q, or Force/Q from the database.



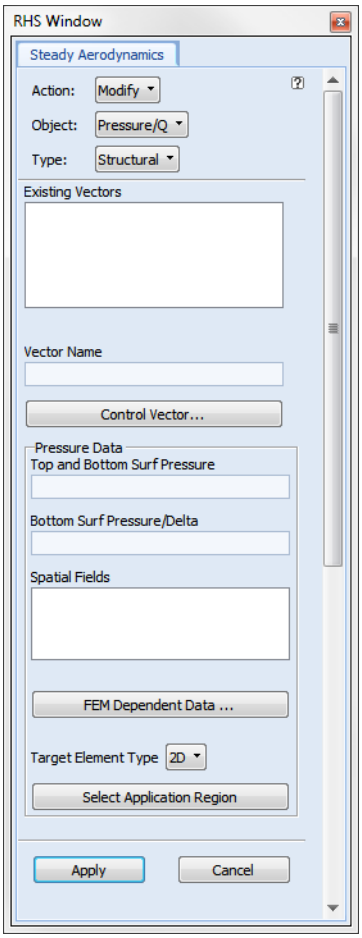
Parameter	Description
Existing Vectors	Sorts the list of existing vector names by object type. Only the current object type is displayed in the listbox.

Note: The Object Normalwash/Q is not available when the current SuperGroup is of type 3D.

Note: This form is valid for all the objects (Pressure/Q, Normalwash/Q, and Force/Q).

Modify/Pressure/Q

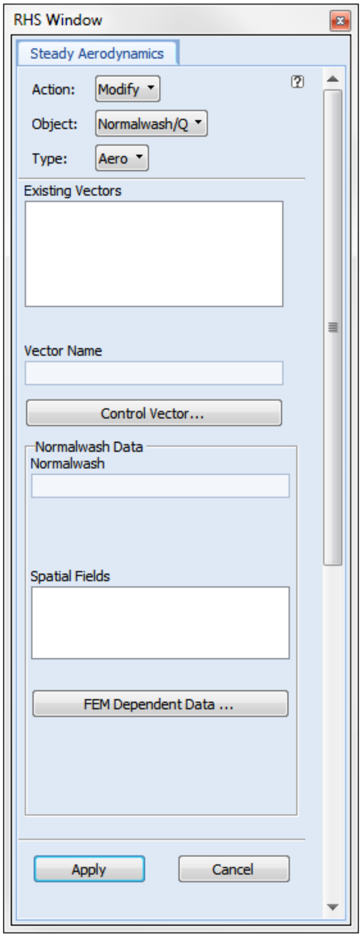
Users can modify existing Pressure/Q. All data associated with Pressure/Q can be redefined.



Parameter	Description
Existing Vectors	Sorts the list of existing vector names by object type. Only the current object type is displayed in the list.
Modify Control Vector	**Note:** The subform displayed by this button is valid for all the objects (Pressure/Q, Normalwash/Q, and Force/Q).
Load/BC Set Scale Factor	The initial Scale factor to be used in defining the Spatial Fields.
Top and Bottom Surf Pressure	Name of a field defining the Top Pressure. This value can be left blank when defining a Delta.
Bottom Surf Pressure/Delta	Name of a field defining the Bottom Pressure or the Delta. **Note:** Above two widgets are replaced with the Nodal Pressure databox for 3D SuperGroups.
Spatial Fields	List of all the created Spatial Fields.
FEM Dependent Data	Becomes active when the user puts the cursor in the “Top and Bottom Surf Pressure” or the “Bottom Surf Pressure/Delta” databox. This subform allows the users to define a new field.

Modify/Normalwash/Q

Users can modify existing Normalwash/Q. All data associated with Normalwash/Q can be redefined.

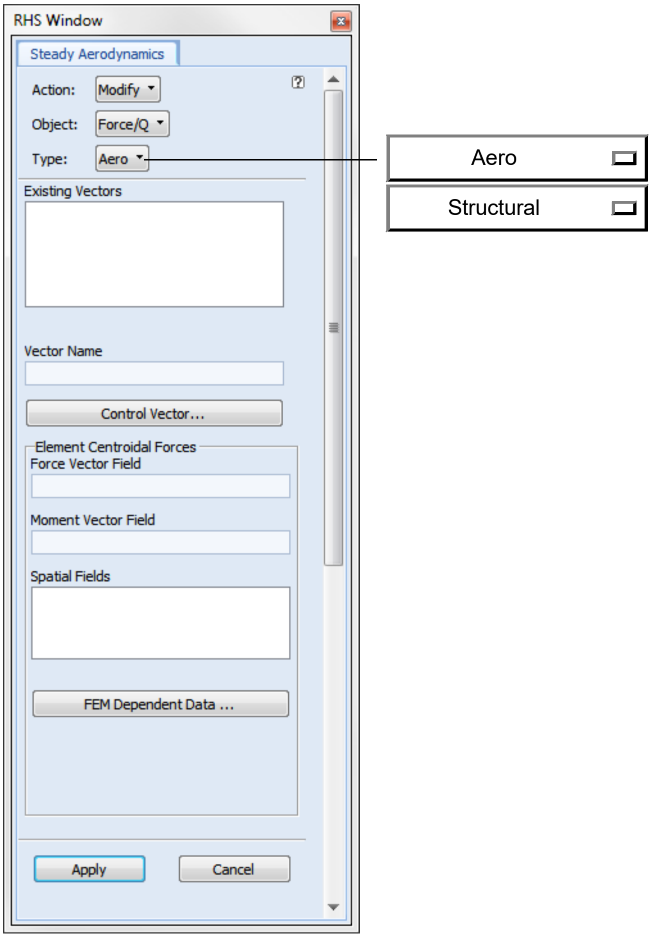


Parameter	Description
Normalwash	Name of a field defining the Normalwash.
Spatial Fields	List of all the created Spatial Fields.
FEM Dependent Data	Becomes active when the user puts the cursor in the "Normalwash" databox. This subform allows the users to define a new field.

Note: The Object Normalwash/Q is not available when the current SuperGroup is of type 3D.

Modify/Force/Q

Users can modify existing Force/Q. All data associated with Force/Q can be redefined.

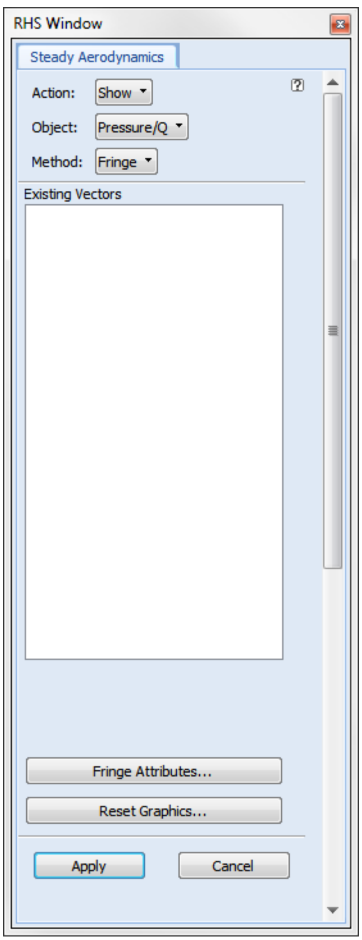


Parameter	Description
Force Vector Field	Name of a field defining the Force Vector.

Parameter	Description
Moment Vector Field	Name of a field defining the Moment Vector.
Spatial Fields	List of all the created Spatial Fields.
FEM Dependent Data	Becomes active when the user puts the cursor in the "Force Vector Field" or the "Moment Vector Field" databox. This subform allows the users to define a new field.
Analysis Coordinate Frame	Analysis Coordinate Frame that is to be associated to the Force/Q.

Show/Any/Fringe

Users can show fringe plots of the existing Pressure/Q or Normalwash/Q.

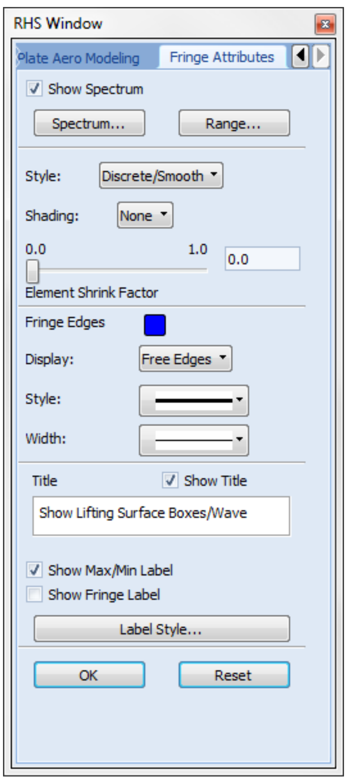


Parameter	Description
Existing Vectors	Sorts the list of existing vector names by object type. Only the current object type is displayed in the list.
Reset Graphics	Resets the graphics to the way they were before the Fringe was applied.

Note: This form is valid for the objects (Pressure/Q and Normalwash/Q).

Show/Any/Fringe /Fringe Attributes

This form allows the users to set the attributes to be used in determining the Fringe plot.



Note: This form is valid for the objects (Pressure/Q and Normalwash/Q).

Unsteady Aerodynamics

In order to perform dynamic aeroelastic stability (Flutter) and response analysis, first define the aerodynamics at specific Mach numbers, M (ratio of speed to the speed of sound) and Reduced Frequencies, k (nondimensional frequency). In MSC FLD, these defined data are used as the basis for interpolation to user-defined analysis points. The interpolation is performed on the union of all Mach-Frequency Pair Sets that are used in any given run. Under user control, these data may be interpolated within a Mach number (e.g., frequencies only) or across all Mach-k pairs in the union of sets. Typically only 6 to 10 frequencies per Mach number are required for an adequate interpolation. Too many frequencies can actually degrade the quality of the interpolation!

In this form you are able to define named sets of Mach Frequency pairs. You will then select these sets on a subcase by subcase basis in Flutter Subcase Creation.

The basic approach is to define ranges of values and select Uniform, Biased or Tabular methods to fill interstitial values. The Mach numbers are typically a single value per set, so it is possible to merely specify a Minimum value if only one is desired. The Reduced Frequency may be defined nondimensionally (that is, by k value) or dimesionally by combinations of cyclic frequency (in Hertz) and dimensional speed. The k value is related to these dimensional parameters by the equation:

$$K = (2F \times b)/2 \times V$$

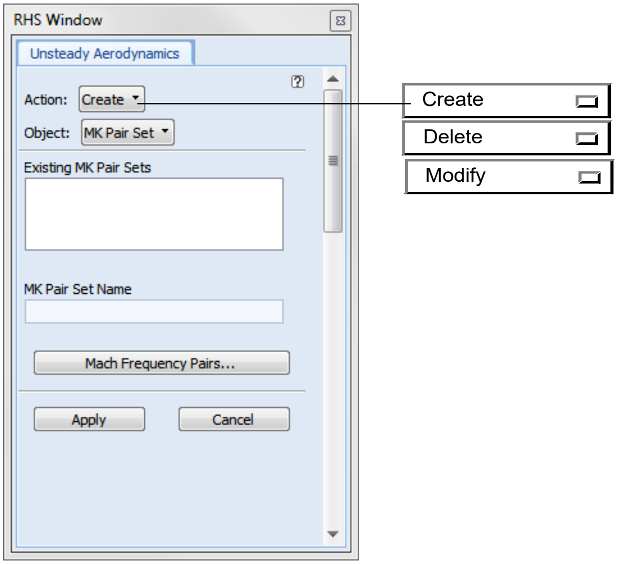
where F is the cyclic frequency, b is the reference chord length from the Global Data and V is the dimensional Velocity.

You may choose dimensional input as a convenience if you understand the frequency range of your analysis (typically coming from the normal mode frequency range of your vehicle) and the speed range (typically coming from the flight condition and flight envelope).

In either case, the data will be converted into reduced frequency values for computational purposes.

Overview

When the user selects Unsteady Aerodynamics, the first form that displays is Create/MK Pair Set. Shown adjacent to this form are all the different actions that can appear as part of the Unsteady Aerodyanmic option:


RHS Window
Unsteady Aerodynamics
Action: Create
Object: MK Pair Set
Existing MK Pair Sets
MK Pair Set Name
Mach Frequency Pairs...
Apply
Cancel
Create
Delete
Modify

Most of the Unsteady Aerodynamic forms are shown and annotated in the following pages, grouped by Action as follows:

- Create
- Delete
- Modify

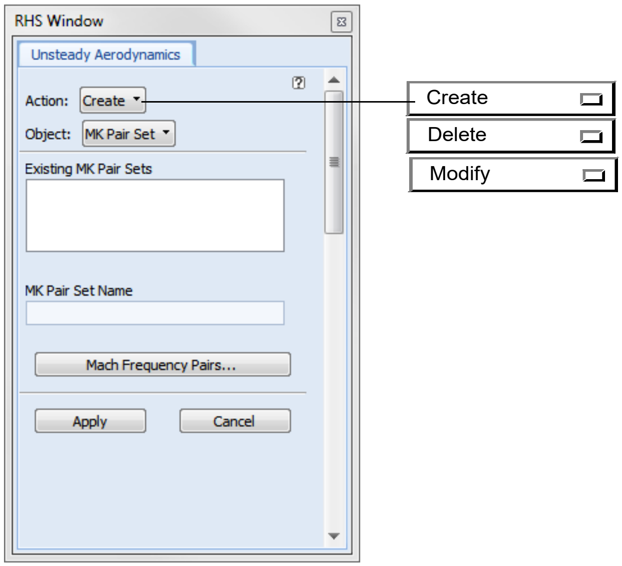
Unsteady Aerodynamics

The following is a list of the forms included in the Unsteady Aerodynamics section:

- Create/MK Pair Set
- Delete/MK Pair Set
- Modify/MK Pair Set
- Mach-Frequency Pairs Subforms
  - Mach-Frequency Pairs/Uniform/NonDimensional
  - Mach-Frequency Pairs/Uniform/Dimensional
  - Mach-Frequency Pairs/One Way Bias/NonDimensional
  - Mach-Frequency Pairs/One Way Bias/Dimensional
  - Mach-Frequency Pairs/Tabular

Create/MK Pair Set

MK Pairs are defined by creating a set name and then defining a series of Mach-Frequency Pairs.



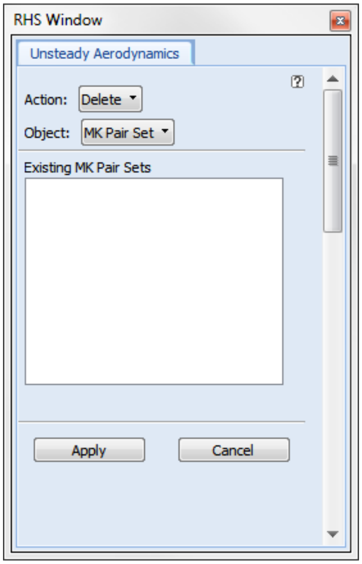
Parameter	Description
Existing MK Pair Sets	This listbox contains the names of all the created MK Pair Sets that are associated with the current Supergroup so that the user can enter a unique name.
MK Pair Set Name	This is where the unique MK Pair Set name is entered.
Mach Frequency Pairs	This button displays the Mach - Frequency Pairs subform.

Note:

1. The user is able to create duplicates of already existing Mach - Frequency Pair Sets. This can be done by selecting an existing name from the "Existing MK Pair Sets" listbox. This name is entered into the "MK Pair Set Name" databox and the data for this selection is displayed on the "Mach - Frequency Pairs" subform. The name needs to be changed before the "Apply" button is selected otherwise an error message is generated.
2. Selecting "Apply" associates the "Mach - Frequency Pairs" with the "MK Pair Set Name" and stores the data in the database in association with the current Supergroup.

Delete/MK Pair Set

This form allows the user to delete existing MK Pairs from the database.



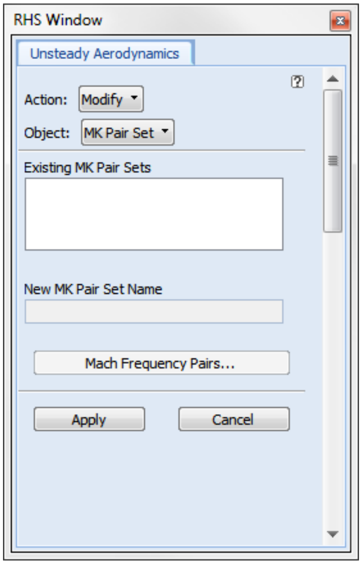
Parameter	Description
Existing MK Pair Sets	Contains the names of all the created MK Pair Sets associated with the current Supergroup. The user selects the Mach - Frequency Pair Sets that are to be deleted.

Note:

1. The user can select any number of Mach - Frequency Pair Sets to be deleted.
2. Selecting "Apply" deletes the "Mach - Frequency Pairs" data and the "MK Pair Set Name" from the database and disassociates it from the current Supergroup.

Modify/MK Pair Set

This form allows the user to modify existing MK Pairs by allowing the set name and the actual Mach-Frequency pairs to be redefined.



Parameter	Description
Existing MK Pair Sets	Contains the names of all the created MK Pair Sets that are associated with the current Supergroup. The user is able to select one name from the list at a time for modification.
New MK Pair Set Name	Name of the MK Pair Set selected for modification.
Mach Frequency Pairs	This button displays the Mach - Frequency Pairs subform.

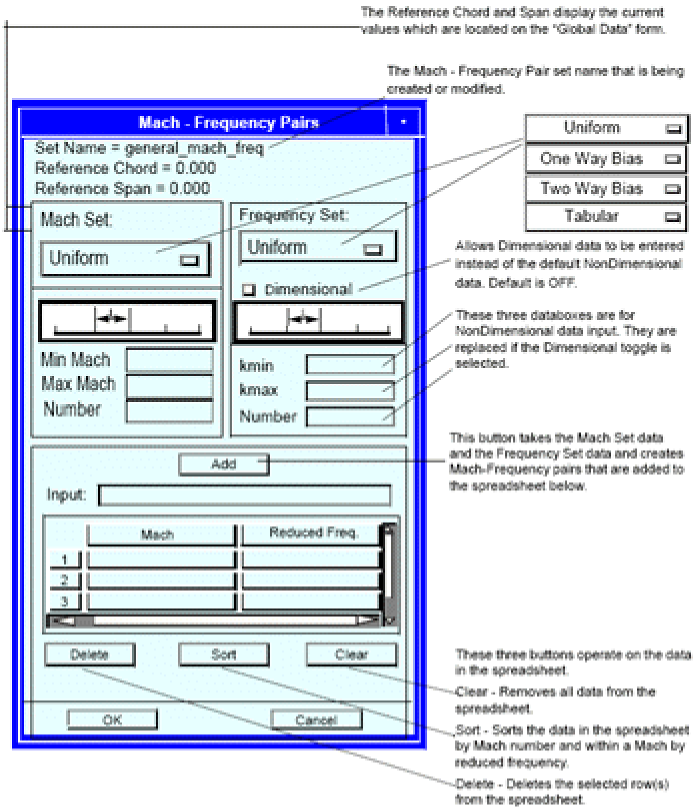
Note:

1. The user selects an existing name from the "Existing MK Pair Sets" listbox. This name is entered into the "New MK Pair Set Name" databox and the data for this selection is displayed on the "Mach - Frequency Pairs" subform.
2. This form can also be used to change the name of the "MK Pair Set". If the name in the "New MK Pair Set Name" databox is different than the selected name in the "Existing MK Pair Sets" listbox, then the new name is associated with the new "Mach - Frequency Pairs" data in the database and the old name/data are deleted.
3. Selecting "Apply" modifies the "Mach - Frequency Pairs" data that is associated with the selected "MK Pair Set Name" and restores the data in the database in association with the current Supergroup.

Mach-Frequency Pairs/Subforms

Mach-Frequency Pairs/Uniform/NonDimensional

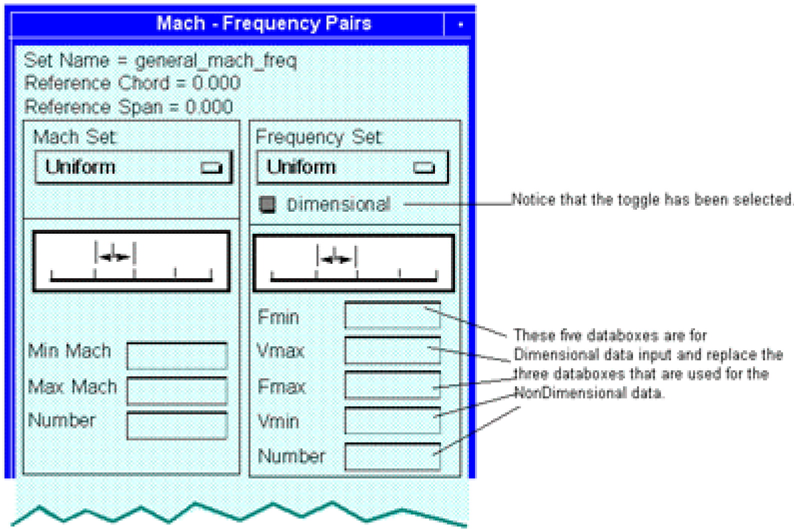
This form is used to define the Mach-Frequency Pairs using uniform nondimensional data.


The Reference Chord and Span display the current values which are located on the "Global Data" form.
The Mach - Frequency Pair set name that is being created or modified.
Mach - Frequency Pairs
Set Name = general_mach_freq
Reference Chord = 0.000
Reference Span = 0.000
Mach Set:
Uniform
Frequency Set:
Uniform
Dimensional
Uniform
One Way Bias
Two Way Bias
Tabular
Allows Dimensional data to be entered instead of the default NonDimensional data. Default is OFF.
Min Mach
Max Mach
Number
kmin
kmax
Number
These three databoxes are for NonDimensional data input. They are replaced if the Dimensional toggle is selected.
Add
This button takes the Mach Set data and the Frequency Set data and creates Mach-Frequency pairs that are added to the spreadsheet below.
Input:
Mach
Reduced Freq.
1
2
3
Delete
Sort
Clear
These three buttons operate on the data in the spreadsheet.
Clear - Removes all data from the spreadsheet.
Sort - Sorts the data in the spreadsheet by Mach number and within a Mach by reduced frequency.
Delete - Deletes the selected row(s) from the spreadsheet.
OK
Cancel

Note:	
	1. This form is valid for both the Create action and the Modify action. If an existing MK Pair Set is selected, then the data associated with that Mach - Frequency Pair is displayed in the spreadsheet. If this is a modify action, these data can be modified. 2. This form should not be displayed if the "Reference Chord" and the "Reference Span" widgets on the "Global Data" form have not already been defined. A message to direct the user should be issued. 3. M = Mach, k = Reduced Frequency, F = Cyclic Frequency, and V = Velocity. 4. The "Min Mach", "Max Mach", "kmin", and "kmax" databoxes are all of type Real and blank by default. 5. The two "Number" databoxes are both of type Integer and are blank by default. 6. The spreadsheet has 20 rows to start. When the "Add" button is selected, the data entered above is transferred into the spreadsheet below any rows that are already filled in. If there are not enough rows in the spreadsheet to hold the new data the "Add" button causes new rows to be created. 7. When the "OK" button is selected, all the information that is currently in the spreadsheet is stored in the database. This data is to be stored in the same way that the "Trim Parameter" data for the "Static Aeroelasticity" solution type is currently being stored. 8. The "Mach Set" option menu and the "Frequency Set" option menu work independent of one another so they do not have to be set to the same values. 9. If only 1M or 1k is desired Min or Max Uniform with 1 should be allowed. 10. Must have at least 1 Mk pair at the time the OK button is selected. 11. The data from this form is used with the MKAERO1 card. 12. The user should be able to cycle the Mach set and Freq. Set and press the "Add" button multiple times - thereby accumulating pairs.

TMach-Frequency Pairs/Uniform/Dimensional

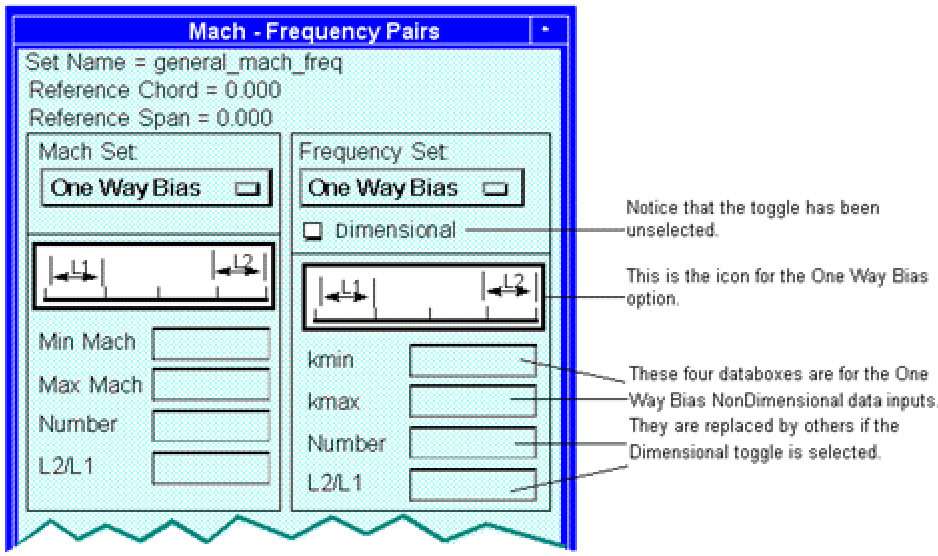
This form is similar to the main Mach-Frequency Pairs form defined on the previous page, but this version uses Uniform Dimensional data.



Note:	
	1. F = Cyclic Frequency and V = Velocity. 2. The “Min Mach”, “Max Mach”, “Fmin”, “Vmax”, “Fmax”, and “Vmin” databoxes are all of type Real and blank by default. 3. The two “Number” databoxes are both of type Integer and are blank by default.

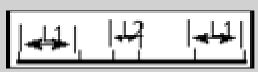
Mach-Frequency Pairs/One Way Bias/NonDimensional

This form is similar to the two previous forms that define the Mach-Frequency pairs but this one uses one way bias nondimensional data.



Note:

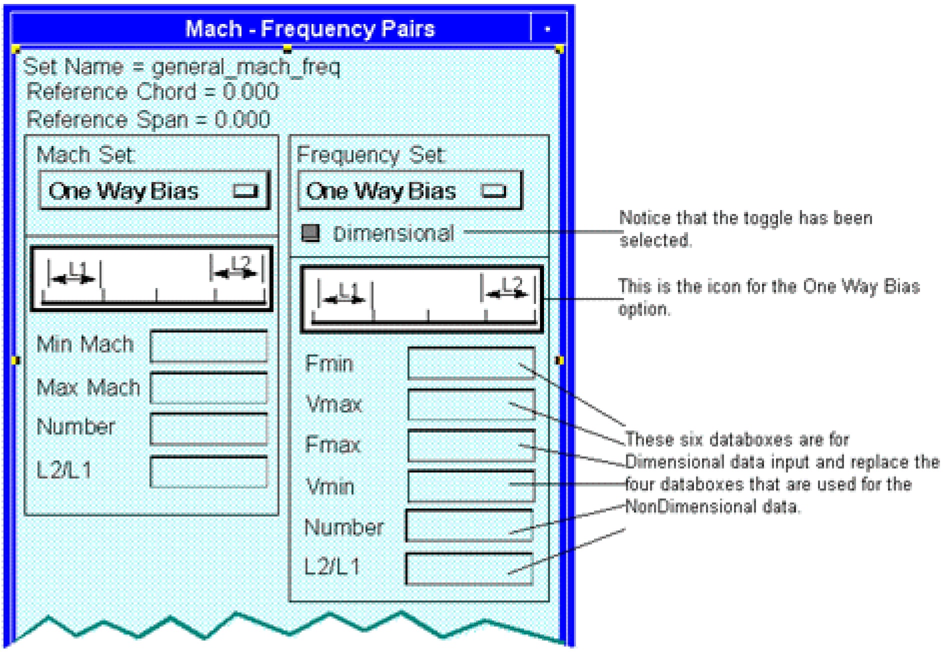
1. This form is valid for both the "One Way Bias" and the "Two Way Bias" option. The only difference between the two forms is the icon. For the "Two Way Bias" option the icon looks like this:

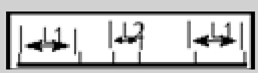
   

2. k = Reduced Frequency.
3. The "Min Mach", "Max Mach", "kmin", "kmax", and the two "L2/L1" databoxes are all of type Real and blank by default.
4. The two "Number" databoxes are both of type Integer and are blank by default.
5. These icons are the same ones that are used on the "Mesh Control" forms.

Mach-Frequency Pairs/One Way Bias/Dimensional

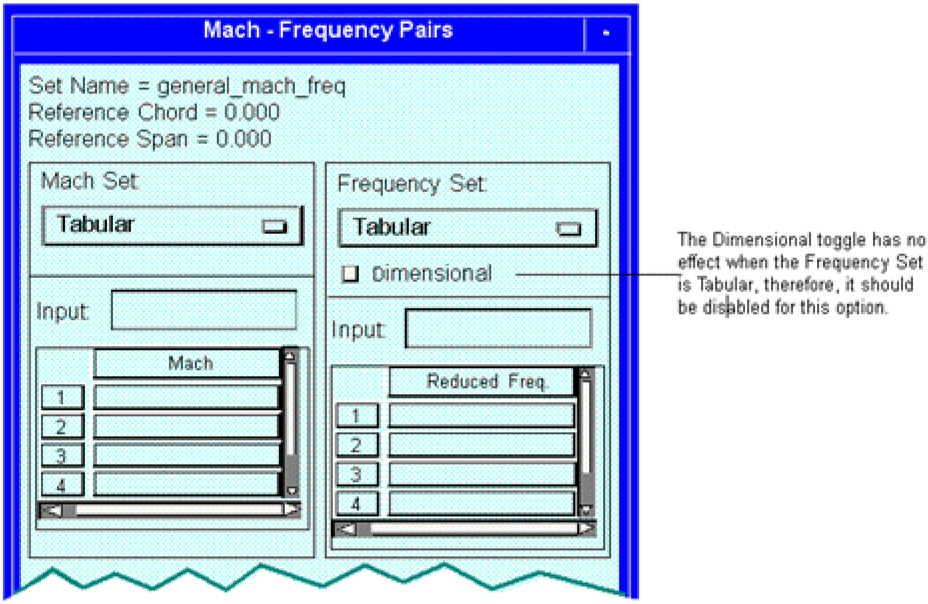
This form is similar to all the previous forms that define the Mach-Frequency pairs but this one uses one way bias dimensional data.



Note:	1. This form is valid for both the "One Way Bias" and the "Two Way Bias" option. The only difference between the two forms is the icon. For the "Two Way Bias" option the icon looks like this:  2. F = Cyclic Frequency and V = Velocity. 3. The "Min Mach", "Max Mach", "Fmin", "Vmax", "Fmax", "Vmin", and the two "L2/L1" databoxes are all of type Real and blank by default. 4. The two "Number" databoxes are both of type Integer and are blank by default. 5. These icons are the same ones that are used on the "Mesh Control" forms.

Mach-Frequency Pairs/Tabular

This form is similar to all the previous forms that define the Mach-Frequency pairs but this one uses tabular data.



Note:

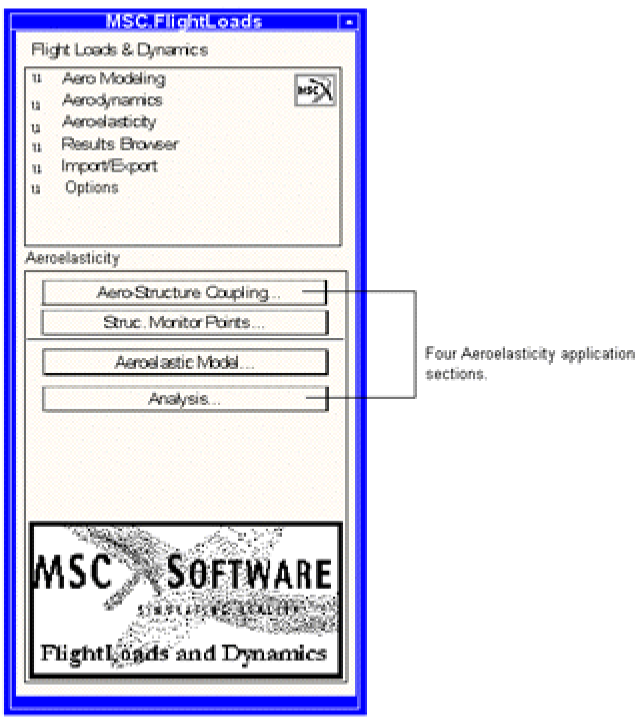
1. There are 25 rows in each spreadsheet.
2. The user enters the data (a real number) in the "Input" databox and presses the Return/Enter key to store the data in the "Mach" or "Reduced Freq." spreadsheet.
3. The user deletes the data in the "Input" databox and presses the Return/Enter key to remove data from the selected cell.

5 Aeroelasticity

Introduction

The Aeroelastic Module is used first to couple the aerodynamic and structural models and subsequently to perform aeroelastic analyses. The module is separated into four areas that are selectable by clicking on one of the ellipsis buttons at the bottom of the form:

- Aero-Structure Coupling - Couples the aerodynamic and structural models. In this module you can create, modify and verify your coupling model that relates the aerodynamic and structural meshes. A number of splining (coupling) methods are available (See Appendix B).
- Struc. Monitor Points - Used to represent summations of forces. In this module you can create and modify your monitor points.
- Aeroelastic Model - User selects the structural and aerodynamic models for use in the current analysis, as well as the associated splines. Structural and aerodynamic parameters that are associated with the model (as opposed to subcase specific parameters) are also defined here.
- Analysis - User selects the analysis type, defines one or more conditions (subcases) and submits the job.



Aero-Structure Coupling

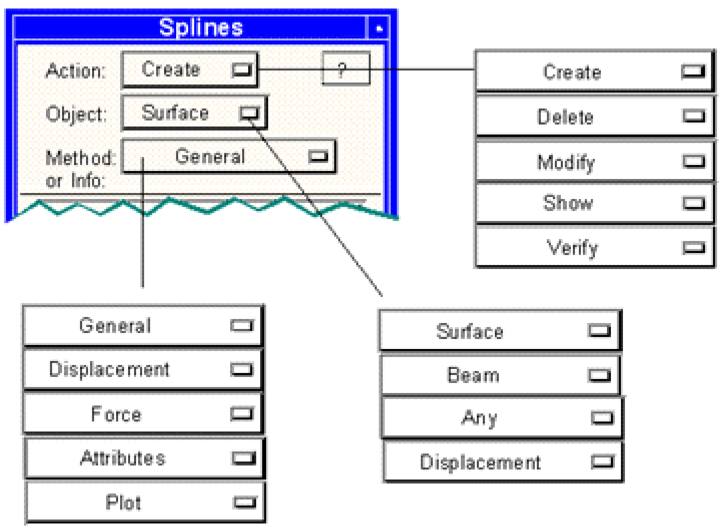
The aerodynamic and structural models are created and exist as completely separate entities. In the MSC.Patran database, any number of structural models and aerodynamics models (SuperGroups) may exist. To perform an analysis, a pair of these models must be "connected" to each other.

In aeroelasticity, what is required is that you relate the model in such a way that the aerodynamic forces can be mapped to the structural model (with equilibrium preservation) and that the structural deformation be mapped to the aerodynamic model to allow aeroelastic forces to be computed.

It is the aero-structure coupling that brings these two models together using splining concepts. Splines (App. B) of this User's Guide provides theoretical information on the variety of splines available in MSC.FlightLoads. Accurate splining is a key task in an aeroelastic analysis and Splines (App. B) provides guidelines on the creation and validation of the splines. This module provides extensive tools for both aspects of coupling.

Overview

When Aero-Structural Coupling is selected, the first form displayed is Create/Surface/General. Shown adjacent to this form are all the different possible Actions, Objects, Methods, and Infos for Splines.



Not all combinations of Action-Object have a Method menu. In some instances it is substituted with an Info menu and in others there is nothing.

There are two basic methods for splines: beam splines and surface splines. In general, beam splines work well for high aspect ratio wings, bodies and for beam structural models. Surface splines work well for low aspect ratio wings and all built-up structures. Note that, in general, it is the nature of the structural model that determines the best spline choice.

Note:	Beam Splines are not available if the current SuperGroup is of type 3D.

Most of the Aero-Structural Coupling forms are shown and annotated in the following pages, grouped by Action as follows:

- Create
- Delete
- Modify
- Show
- Verify

Definitions for Spline Methods

The possible Methods are: "Displacement" which is used to transfer structural displacements to the aero model, "Force" which is used to transfer aerodynamic forces to the structural model and "General" where in the same structure and aero points are used to construct both the displacement and force transfer mechanism. In this latter case, which is the most commonly used method in MSC.Nastran, the matrix used to spline forces is simply the transpose of the matrix used to spline displacements. Users may, however, want to separate the spline usage because of the desire to map aerodynamic forces only to structural load pick up points (i.e., aircraft wing spar/rib intersections) while a larger set of structural points would more accurately define a displacement shape to be mapped to the aerodynamic model. Using this separate approach may actually increase solution efficiency due to the (potentially) reduced spline matrix densities.

Aero-Structure Coupling

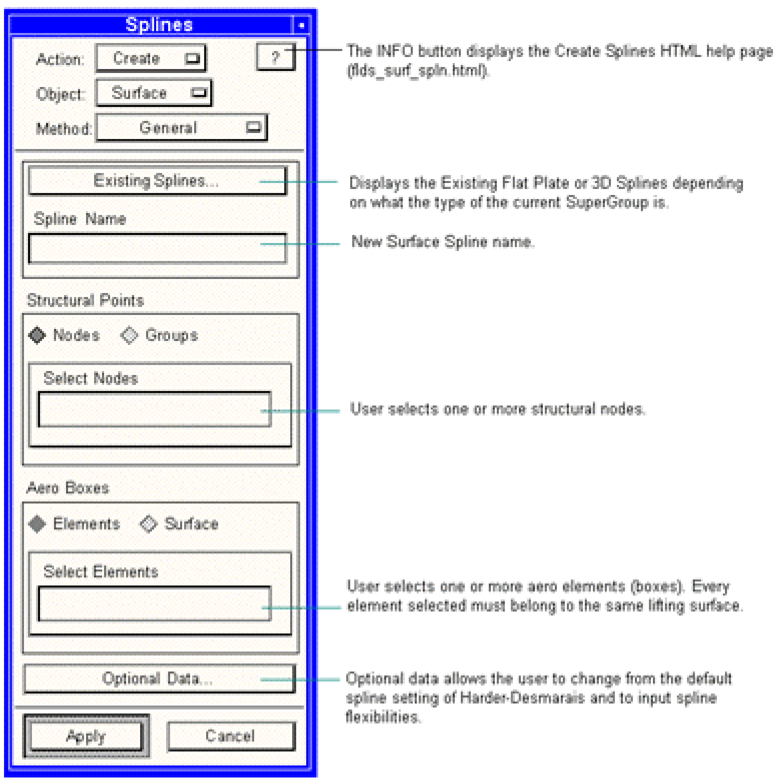
Following is a list of the Aeroelastic forms and subforms related to Aero-Structure Coupling (Splines):

- Create/Surface (General, Displacement, or Force)
- Create/Surface Subforms
  - Existing Splines
  - Select Groups
  - Select Surface
  - Optional Data
- Create/Beam (General, Displacement, or Force) Aero Body
- Create/Beam (General, Displacement, or Force) Aero Surface
- Create/Beam Subforms
  - Select Body
  - Select Surface
  - Optional Data
- Delete (Any, Surface, or Beam)

- Modify/Surface (General, Displacement, or Force)
- Modify/Beam (General, Displacement, or Force)
- Show/Surface or Beam/Attributes
- Show Attribute Information
- Verify/Displacement/Plot

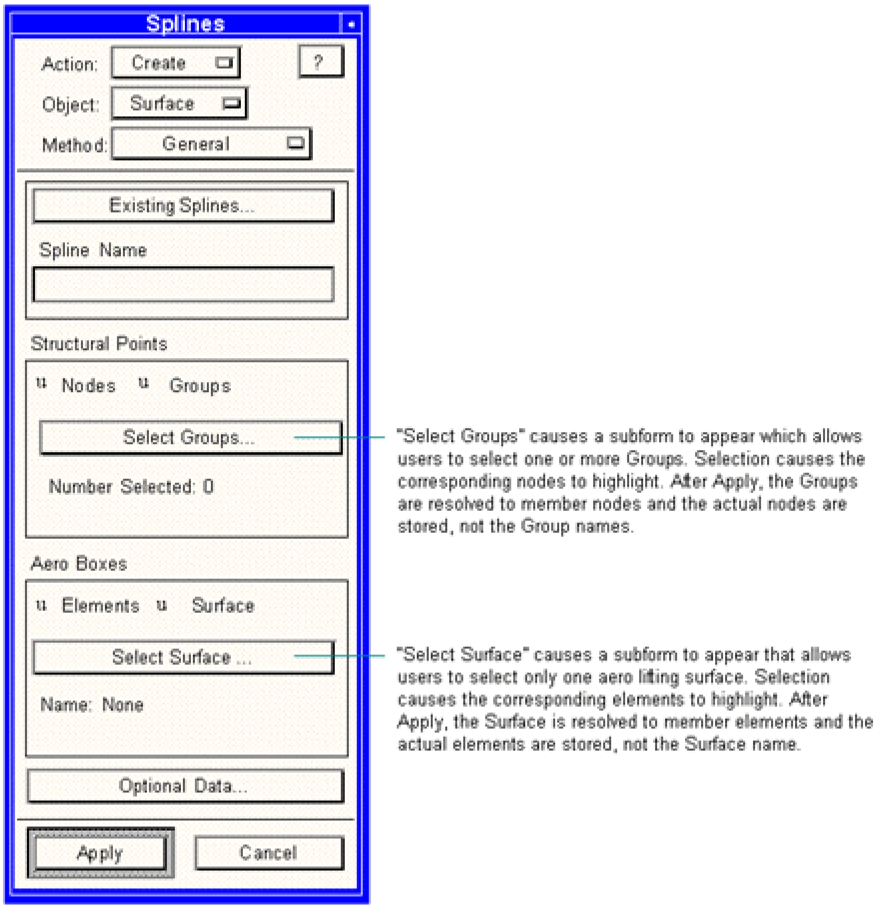
Create/Surface (General, Displacement, or Force)

Surface splines are typically used to couple aerodynamic lifting surfaces to structural plate models representing upper and/or lower lifting surfaces. The Create/Surface form is used to create an intermediate surface that serves to develop the spline matrices. FlightLoads supports three variations of surface splines. The first is the Harder-Desmarais spline where the intermediate surface is a flat plate that is infinite in extent. The second is the Thin Plate spline where the intermediate "surface" is the swarm of structural points (which must not be coplanar). The third type is a Finite Plate spline wherein the intermediate surface is finite in extent.



Note: Only thin plate surface splines are allowed when the current SuperGroup is of Type 3D.

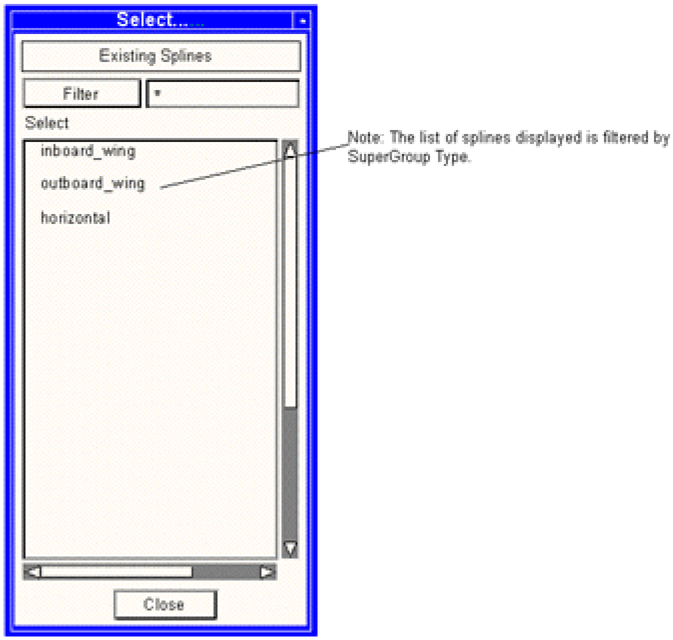
The Create/Surface form defines a name for the spline and identifies the structural points and aerodynamic boxes that are connected to the spline. Subforms allow the user to display the names of the previously Existing Splines and to set Optional Data. The structural node selection can be done by selecting or typing in the node ID's or by selecting one or more structural groups. In a similar manner, the Aero Boxes can be selected by typing in or picking the element ID's or the Surface option can be used to include all of the elements from only one Lifting Surface.



Create/Surface Subforms

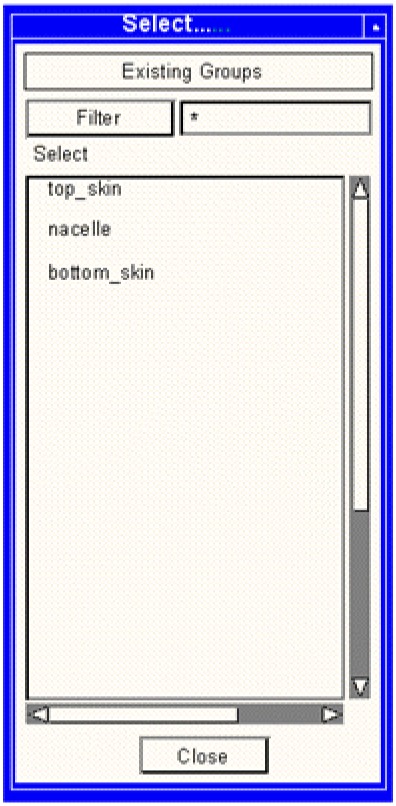
Existing Splines

This subform allows users to see the names of previously created splines. Selection of a spline does not result in any information being transferred back to the Create form.



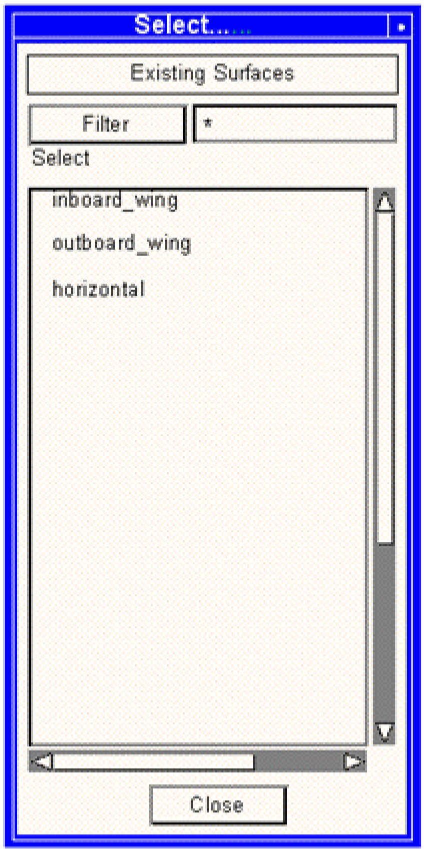
Select Groups

This subform allows users to conveniently select a set of structural nodes to use in the spline simply by their membership in one or more Groups. After spline creation, the structural nodes are stored, not the Group names.



Select Surface

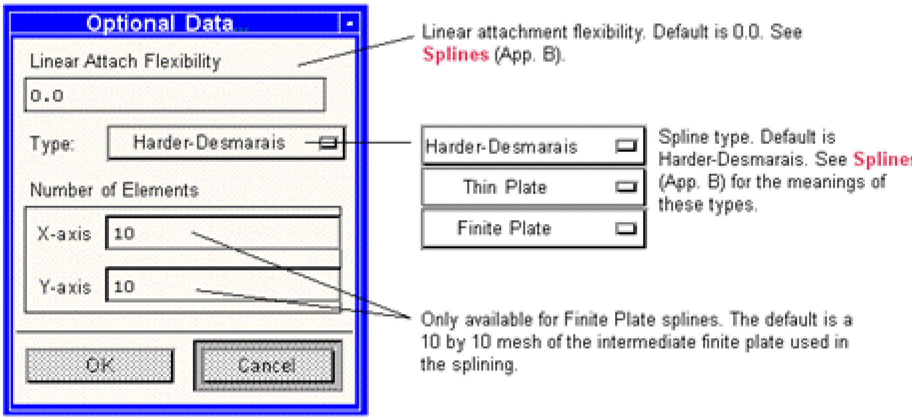
This subform allows users to conveniently select a set of aerodynamic elements to use in the spline simply by their membership in a Lifting Surface or a 3D Aero surface. After spline creation, the aerodynamic elements are stored, not the Lifting Surface name.



Optional Data

This subform allows users to change the default spline type from Harder-Desmarais to either Thin Plate or Finite Plate. Depending on the spline type selected, the allowable user input varies (see below).

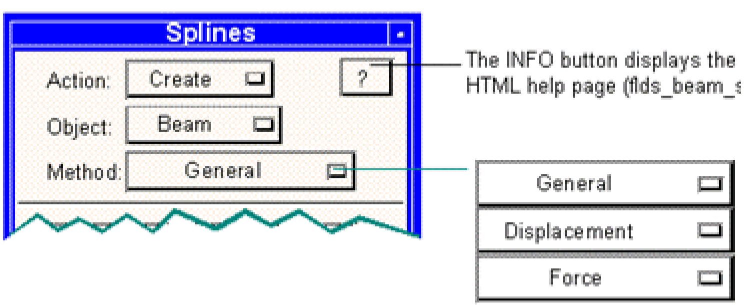
Note: If the current SuperGroup is of type 3D then the only type available is thin plate.



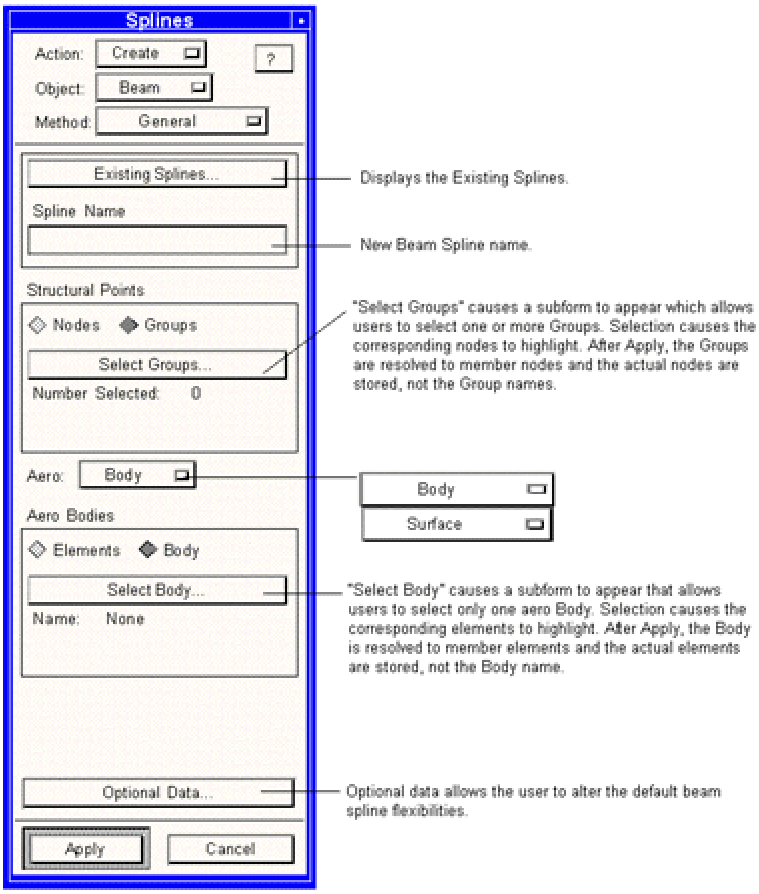
Create/Beam (General, Displacement, or Force) Aero Body

Beam splines are typically used to couple either aerodynamic lifting surfaces or bodies to structural beam models whose stiffness represent the bending and torsion characteristics of an actual vehicle component (i.e., wing, fuselage, etc.). The Create/Beam form is used to generate a spline that is conceptually based on an intermediate structure which is a one-dimensional beam of infinite extent. The form itself is similar to the Create/Surface form with the modification that the associated Aero elements can come from either a lifting surface or a body. If a lifting surface is being splined, it is necessary to define a coordinate system whose Y-axis extends along the beam. (Aerodynamic bodies do not require a coordinate system; the body's X-axis defines the spline axis.)

Note: This object is not available when the current SuperGroup is of type 3D.

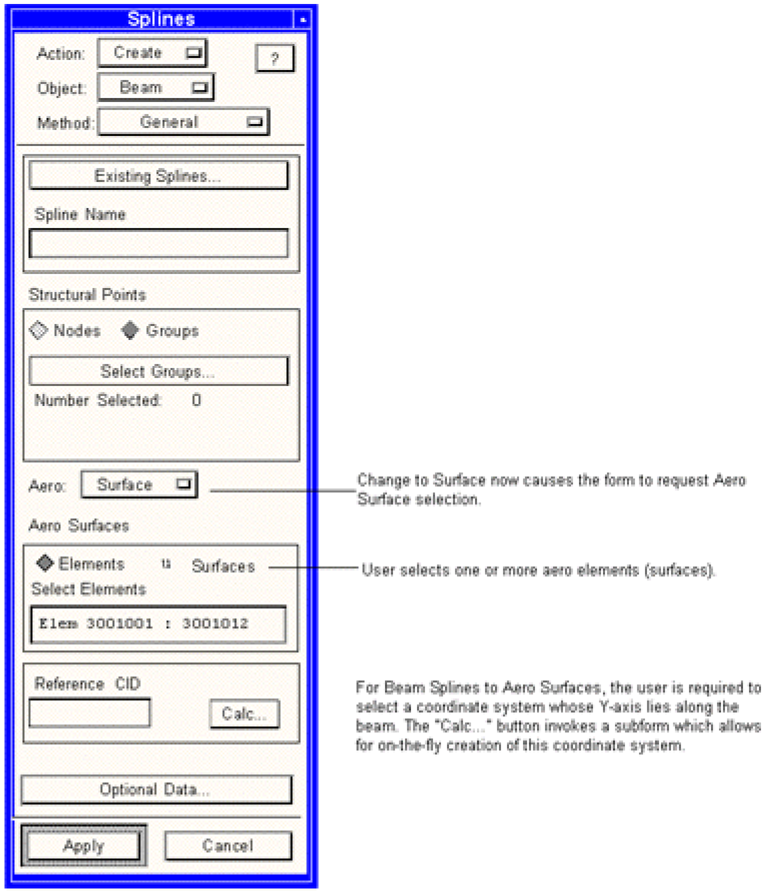


This form is used to spline structural beams to aerodynamic bodies. The Create/Beam form defines a name for the spline and identifies the structural points and aerodynamic bodies that are connected to the spline. Subforms allow the user to display the names of the previously Existing Splines and to set Optional Data. The structural node selection can be done by selecting or typing in the node ID's or by selecting one or more structural groups. In a similar manner, the Aero Bodies can be selected by typing in or picking the element ID's or the Body option can be used to include all of the elements from only one Body.



Create/Beam (General, Displacement, or Force) Aero Surface

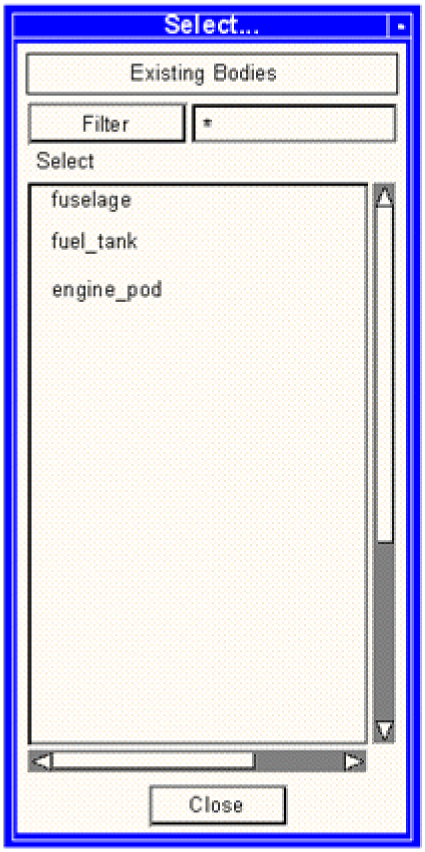
This form is used to spline structural beams to aerodynamic lifting surfaces.



Create/Beam Subforms

Select Body

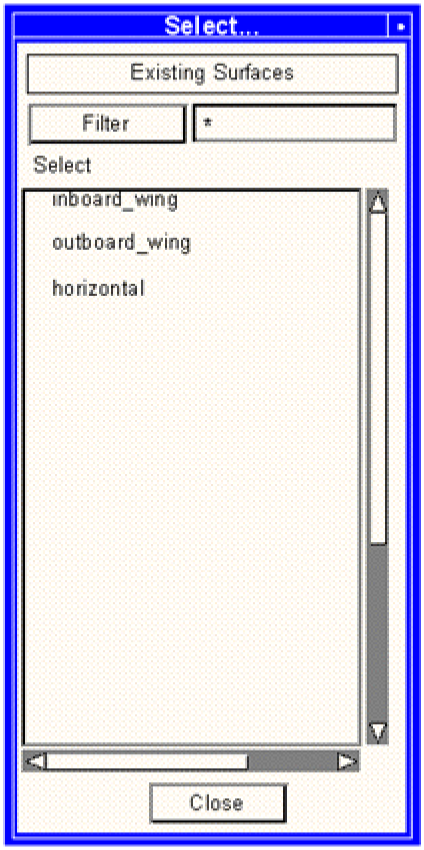
This subform allows users to conveniently select a set of aerodynamic elements to be used in the spline simply by their membership in a Body. After spline creation, the aerodynamic elements are stored, not the Body name.



Note: The user selects one Surface.

Select Surface

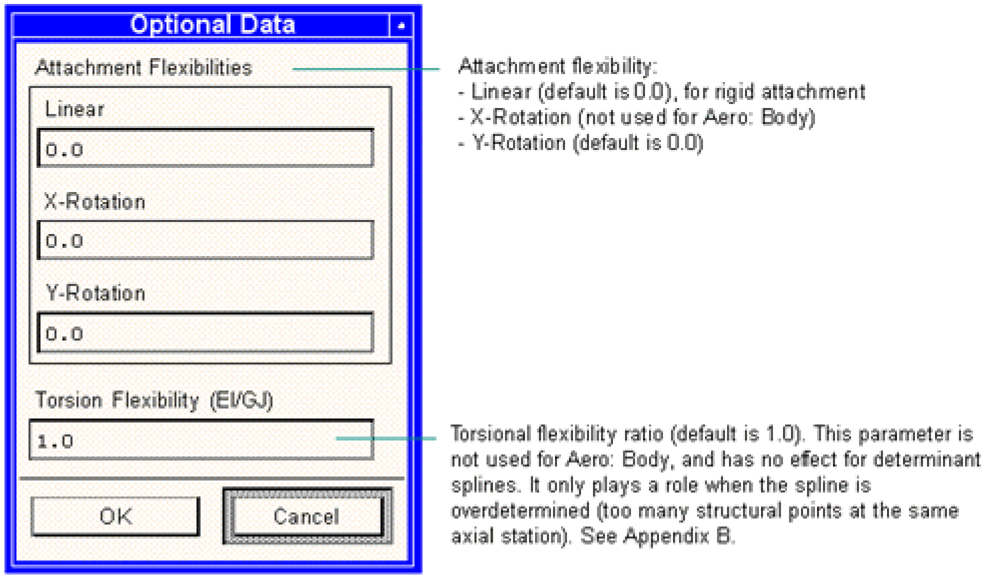
This subform allows users to conveniently select a set of aerodynamic elements to be used in the spline simply by their membership in a Lifting Surface. After spline creation, the aerodynamic elements are stored, not the Lifting Surface name.



Note: The user selects one Surface.

Optional Data

This subform allows users to change the default beam spline attachment flexibility. A negative value for the rotation attachments means the spline is not attached to these rotations; that is, the structural rotations are not transferred to the aero model (types DISPLACEMENT, BOTH) and/or the aerodynamic forces are not put a moment on the associated structural points (types FORCE, BOTH).

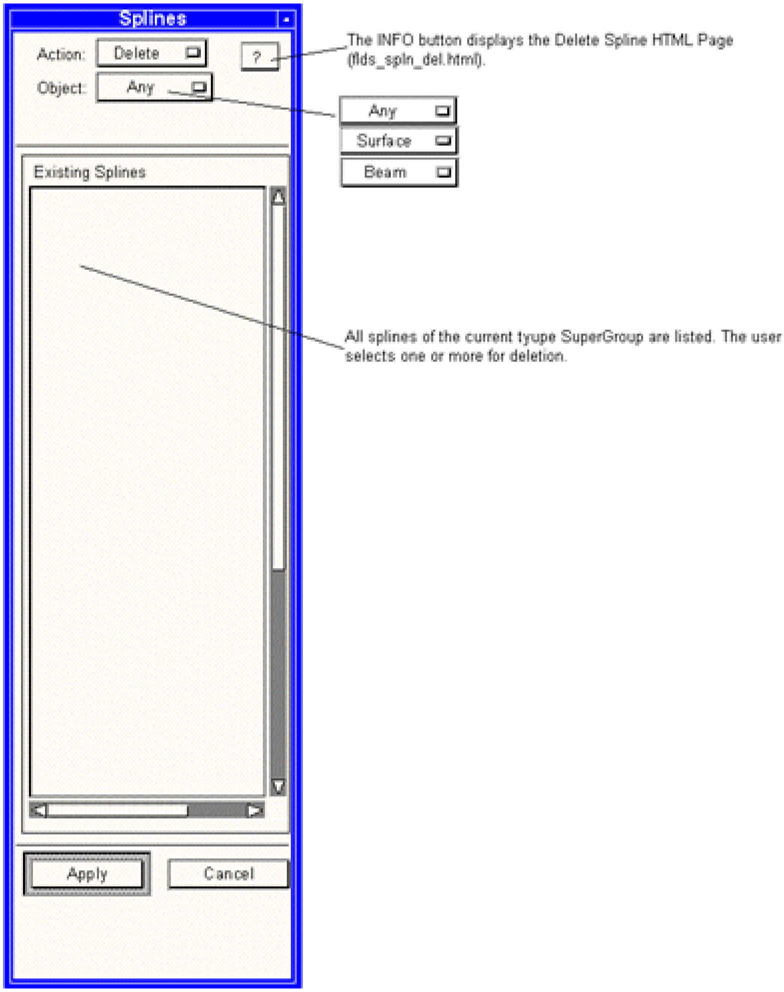


Delete Any, Surface or Beam

Delete allows users to remove previously created Surface and Beam Splines from the database. For Object = Any, users may select Surface and/or Beam Splines for subsequent deletion.

The Delete/Any version of the form is shown below.

Note:	If the current SuperGroup is of type 3D, then only the Any and Surface Object will be available on this form.

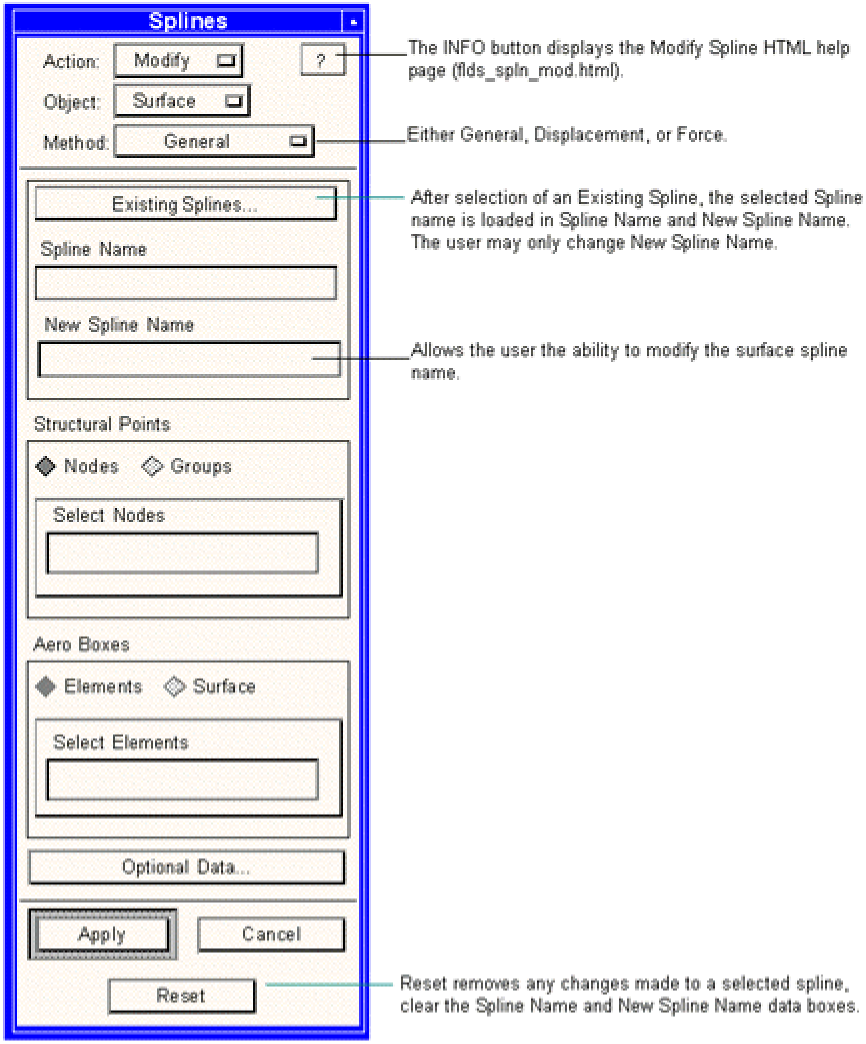


Note: If Object is set to a specific object (i.e., Surface or Beam), then the "Existing..." label reflects the current object. The listbox is also filtered.

Modify/Surface (General, Displacement, or Force)

This form is identical to Create/Surface Spline form, with the added ability to modify the spline name. It allows users to alter previously created surface splines.

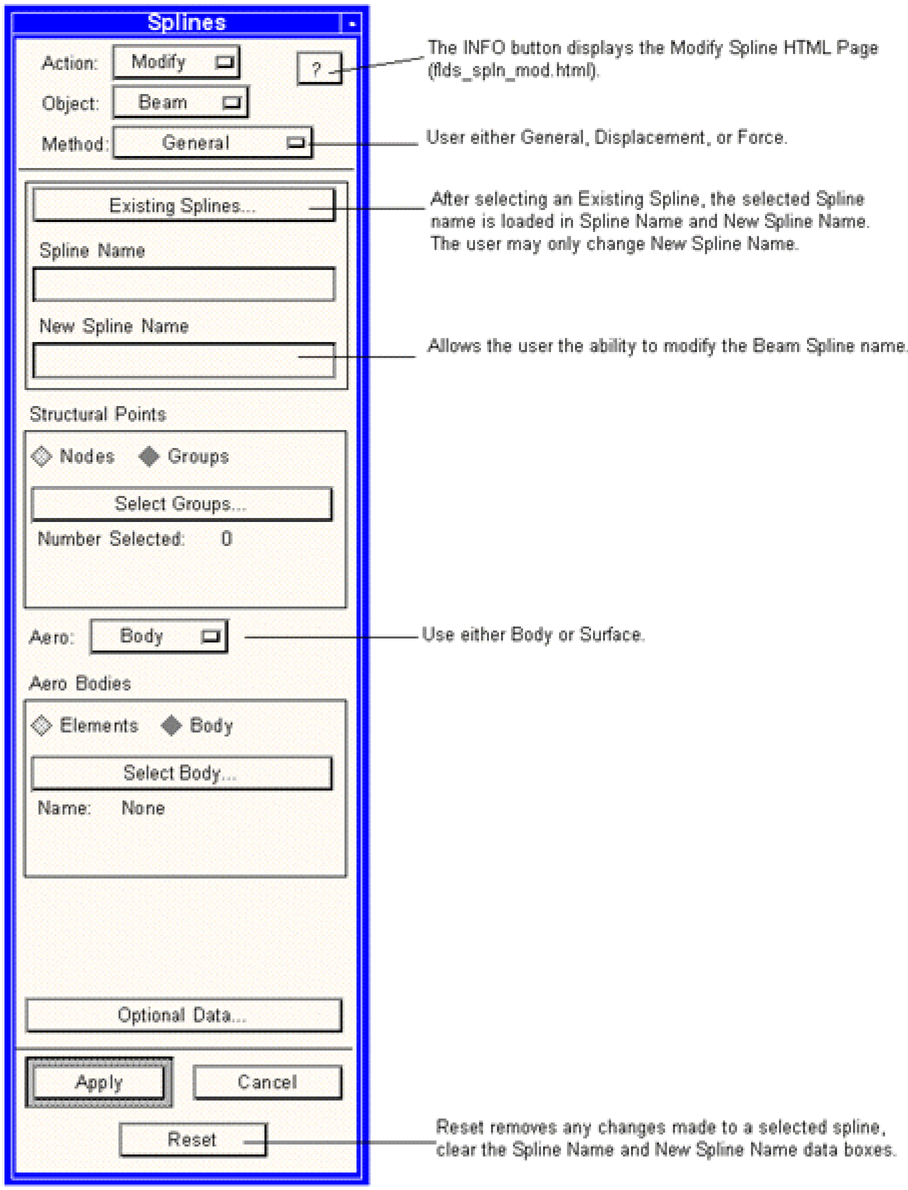
Note: Only Thin Plate Surface splines can be modified when the current SuperGroup is of type 3D.



Modify/Beam (General, Displacement, or Force)

This form is identical to Create/Beam Spline form for both Aero Body and Aero Surface (Aero Body is shown below), with the added ability to modify the spline name.

Note: This Object is not available when the current SuperGroup is of type 3D.

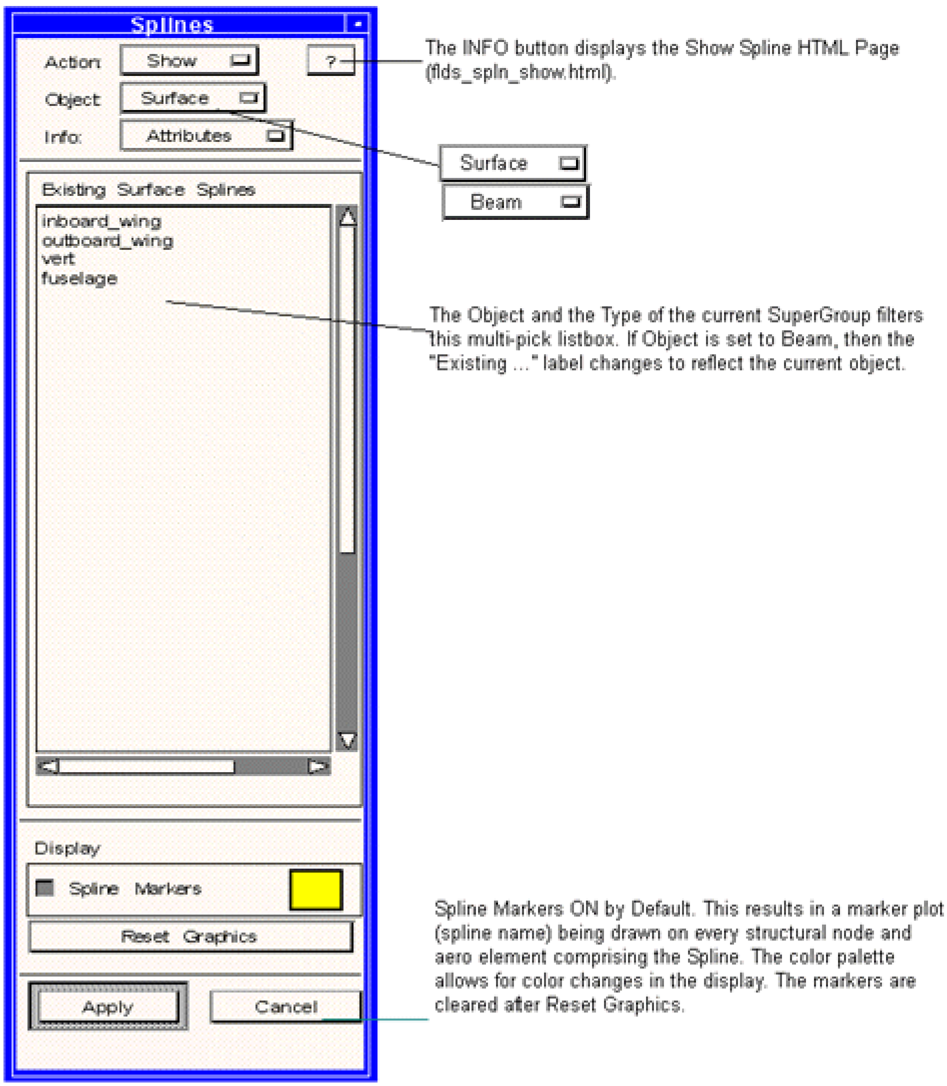


Show Surface or Beam/Attributes

Show allows users to review information on previously created Surface and Beam Splines. This information is displayed in both tabular and graphical formats. Selecting Apply on this form results in the display of a spreadsheet of information (see next page) and optionally markers on the selected splines. The

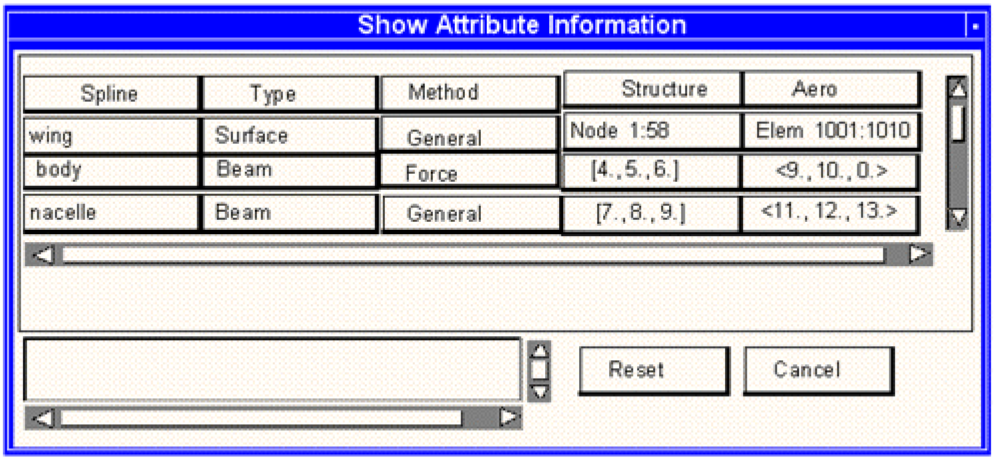
Show/Surface/Attributes version of the form is shown here. Showing Spline attributes results in the display of a spreadsheet of information (see next page) and, optionally, markers on the selected Splines.

Note: This Object Beam is not available when the current SuperGroup is of type 3D.



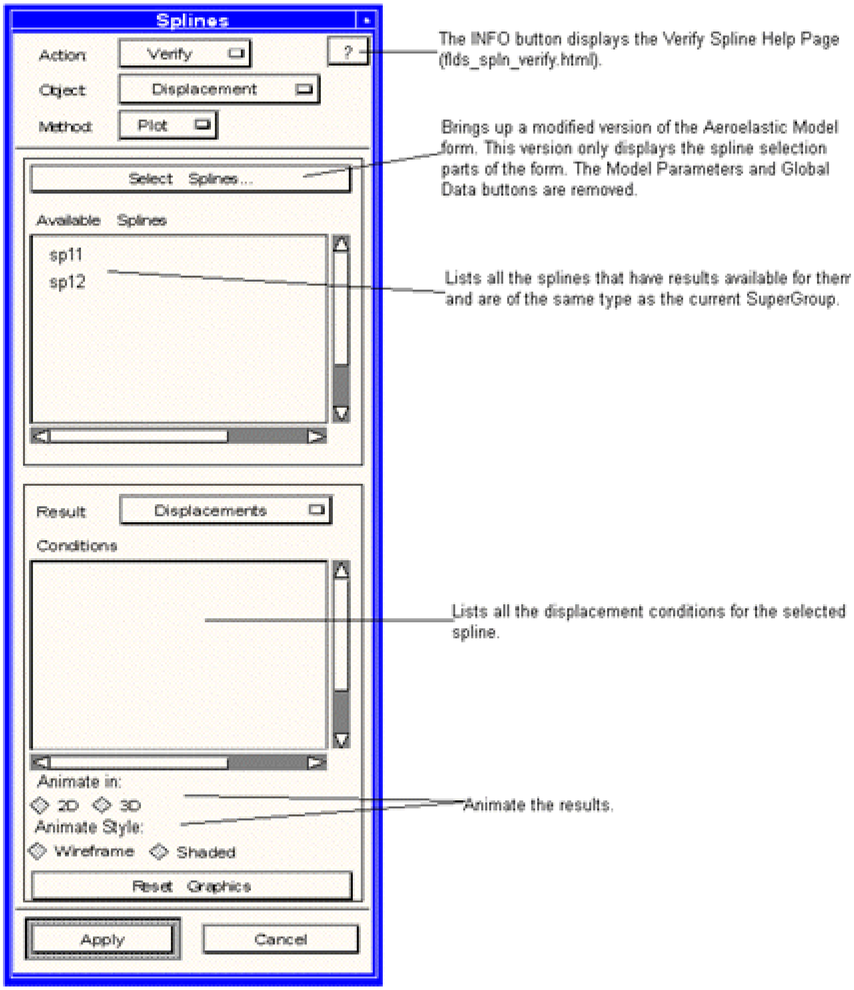
Show Attribute Information

Selection of any cell in the spreadsheet causes that row's information to be displayed in more detail in the information box.



Verify/Displacement/Plot

Verify allows the users to verify previously created Surface and Beam Splines using displacements and to, optionally, animate the results.

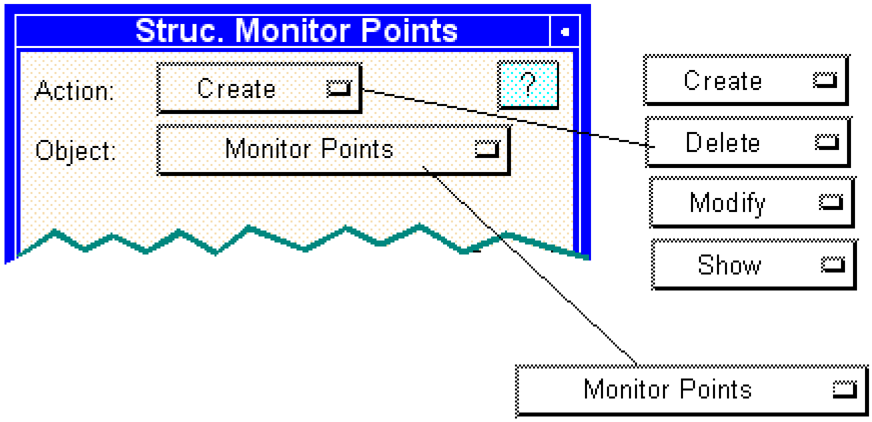


Structural Monitor Points

Structural Monitor Points are used to represent summations of forces over certain regions of the Structural finite element (FE) mesh about a selected point. These Monitor Points can then be combined to represent the integrated trimmed loads appropriate for critical load survey. For Structural Monitor Points, only Structural FE nodes may be selected.

Overview

When the user selects Struc. Monitor Points, the first form that displays is Create/Monitor Points form. Shown adjacent to this form are ALL the different Actions and Objects that can appear as part of the Struc. Monitor Points option.



To complete the forms, the user would start by selecting the Action and Object and then completing the remainder of the form.

Most of the Struc. Monitor Point forms are shown and annotated in the following pages, group by action as follows:

- Create
- Delete
- Modify
- Show

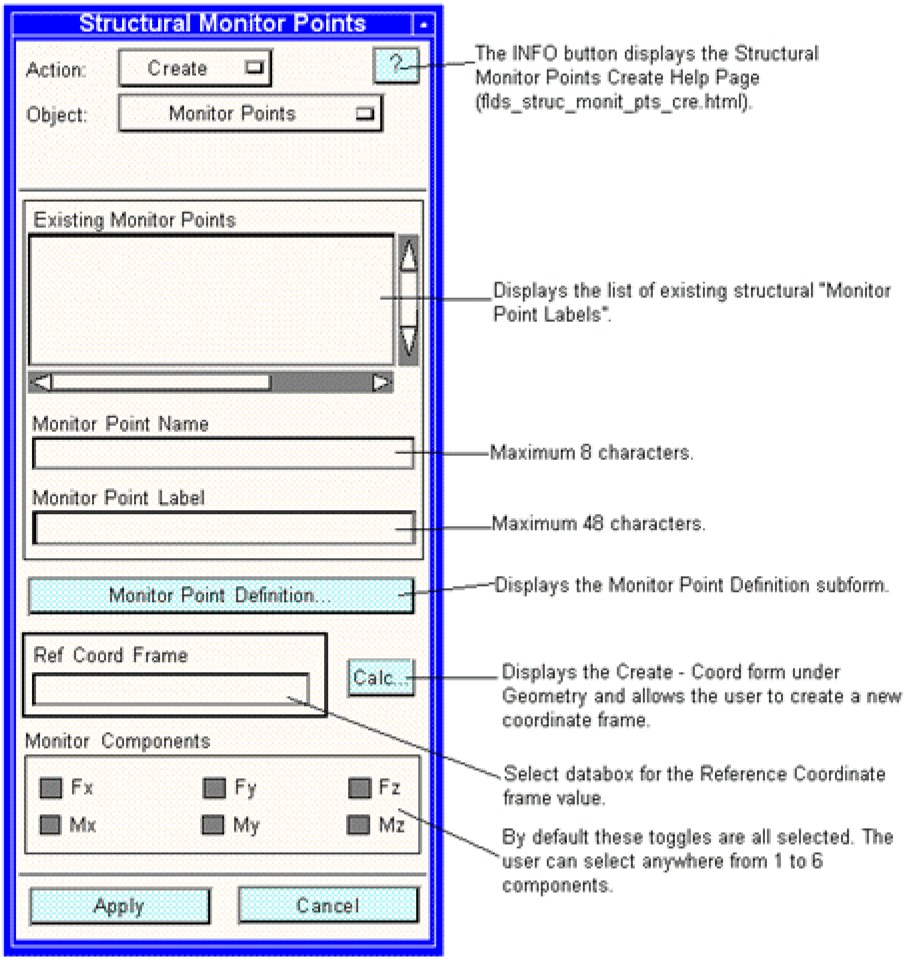
Structural Monitor Points

Following is a list of the Structural Monitor Points forms and subforms:

- Create/Monitor Points
    - Monitor Point Definition
- Delete/Monitor Points
- Modify/Monitor Points
- Show/Monitor Points
    - Structural Monitor Point Info

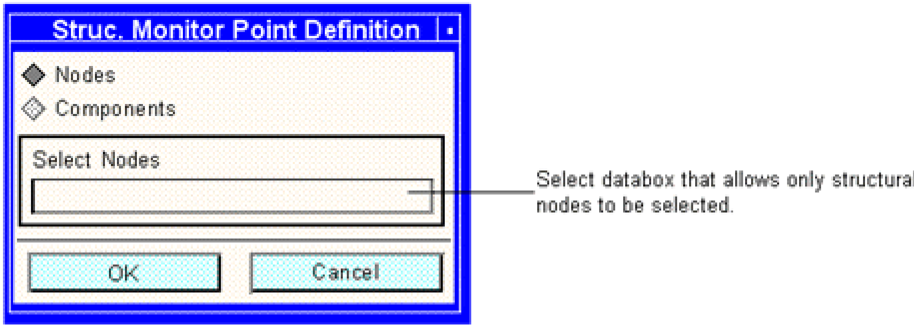
Create/Monitor Points

Structural Monitor Points can be defined by selecting a structural node or group and by providing a Label, the Reference Coordinate Frame, and the Monitor Components.

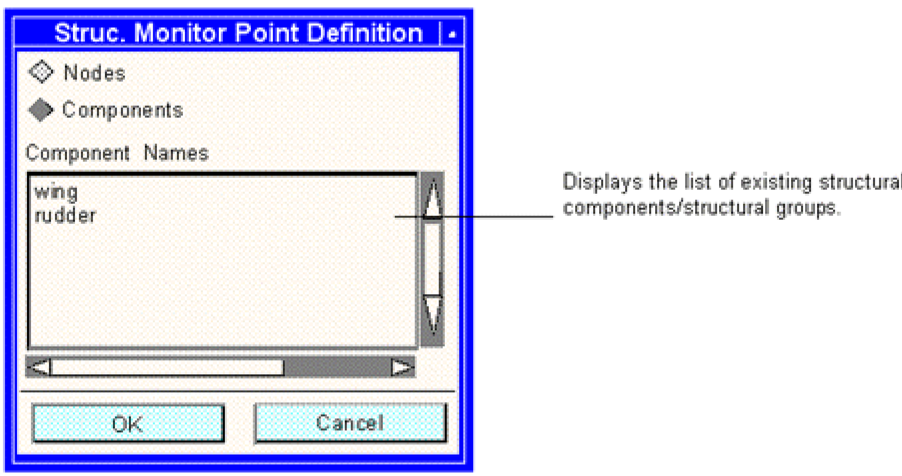


Monitor Point Definition

This form allows the user to select either structural nodes or groups for defining the structural monitor point.



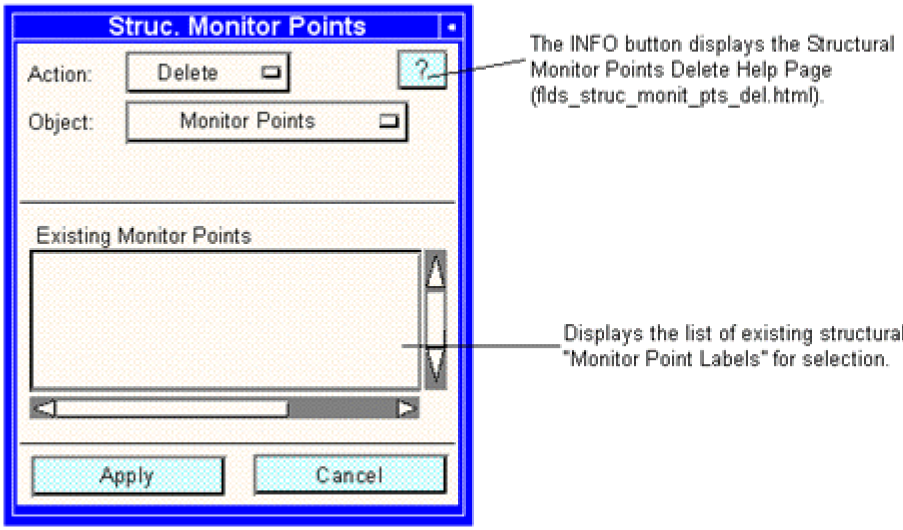
Note: This form is also valid for the modify action.



Note: This form is also valid for the modify action.

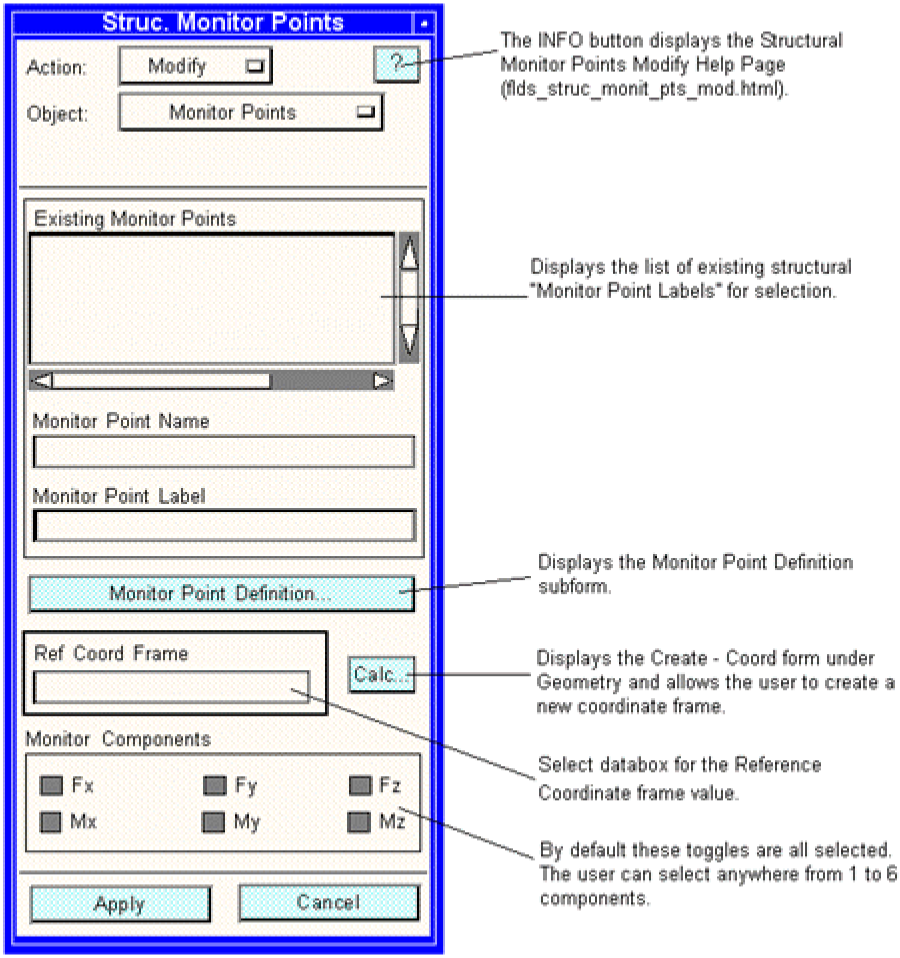
Delete Monitor Points

This form allows the user to delete structural Monitor Points from the database.



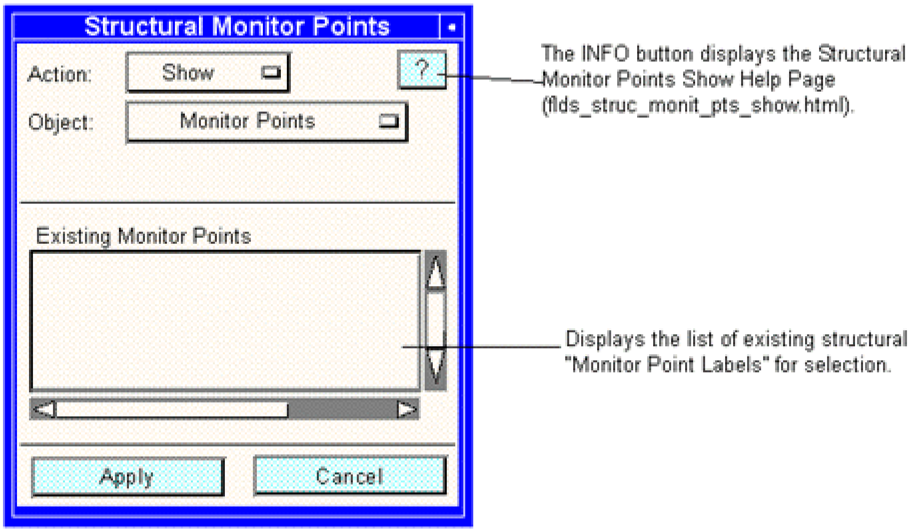
Modify Monitor Points

Users can modify existing structural Monitor Points. All the data associated to the Structural Monitor Point can be redefined.



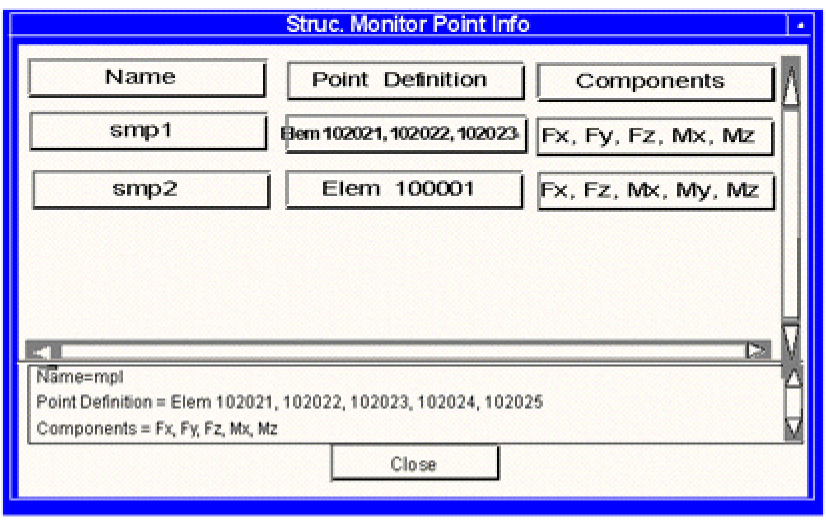
Show/Structural Monitor Points

This form allows the user to look at the data used to define the selected Structural Monitor Points.



Structural Monitor Point Info

This spreadsheet show the Name, Point Definition, and the Components for all the selected Structural Monitor Points.



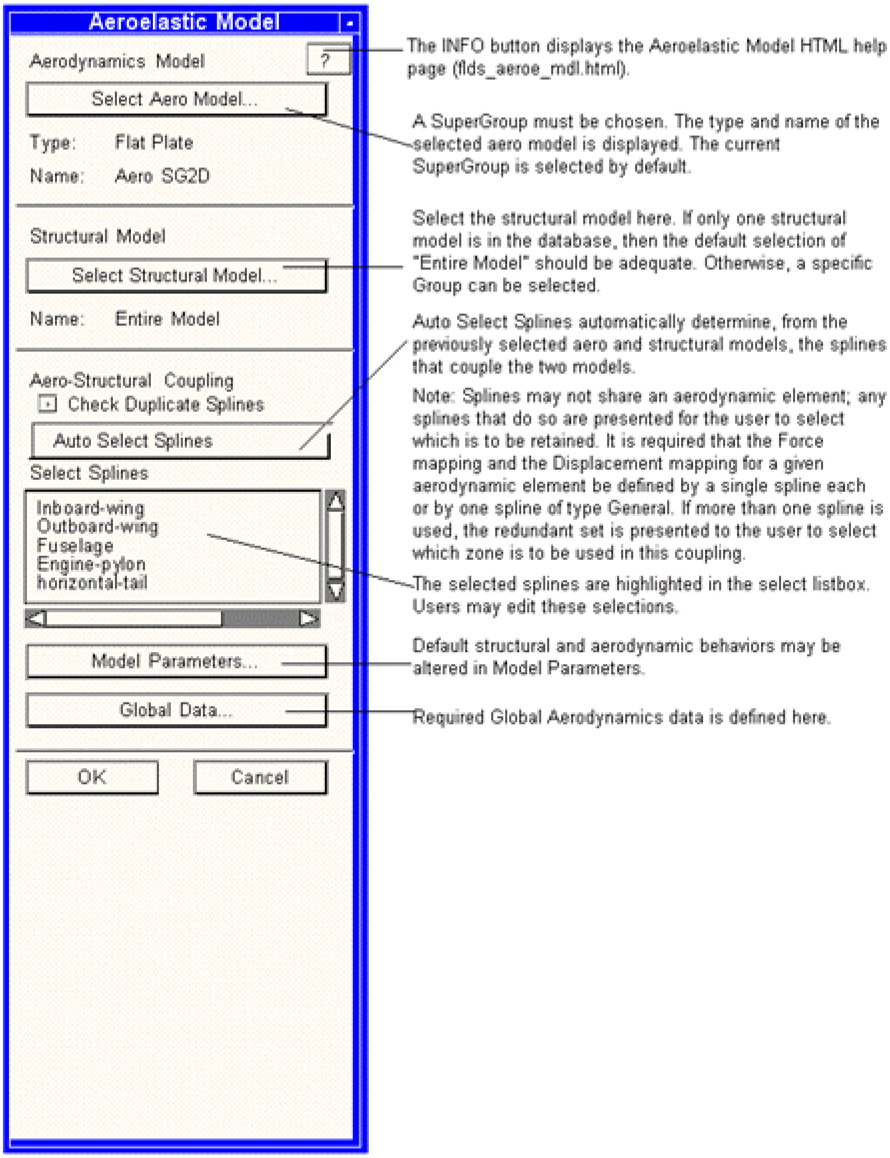
Aeroelastic Model

Aeroelasticity, as the name implies, is the coupling of aerodynamics and structures. Prior to performing an aeroelastic analysis with MSC.FlightLoads and Dynamics, an aeroelastic model must be identified. This model is comprised of a structures model (of necessity, one that is appropriate for dynamics analysis -- it should include appropriate inertias and produce good normal modes), an aerodynamic model, and the splines that couple the two models. FlightLoads was designed to support the evolution (maturation) of aero and structural models. Multiple representations of the structure and the aerodynamic meshes may appear in a single database for subsequent use in aeroelastic analyses. It is also appropriate at this point in the simulation process to define the model-dependent parameter data pertaining to structural and aerodynamic model behaviors.

Following is a list of the Aeroelastic Model form and its subforms:

- Aeroelastic Model
- Aeroelastic Model Subforms
    - Select Aero Model
    - Select Structural Model
    - Model Parameters
    - Global Data

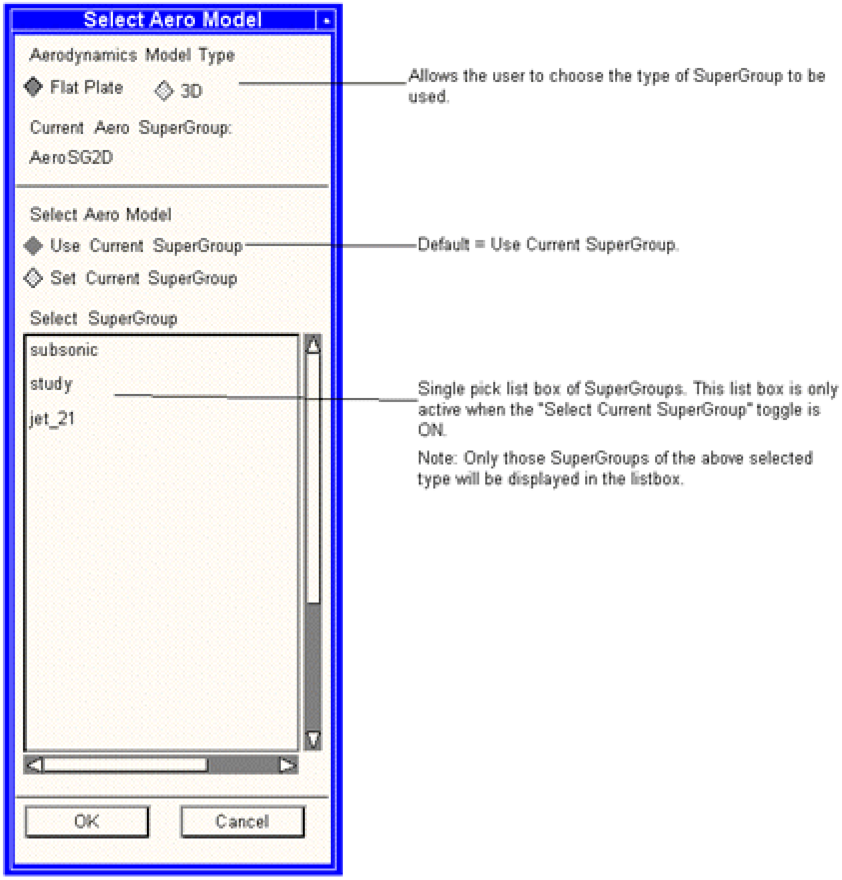
Aeroelastic Model Form



Aeroelastic Model Subforms

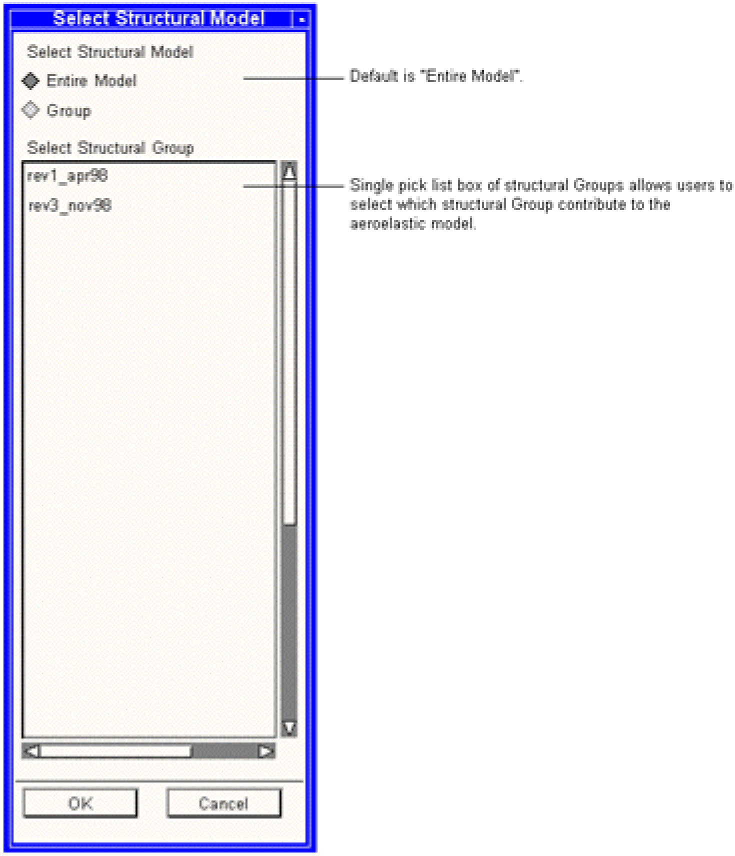
Select Aero Model

This subform is used to identify the aerodynamic component of the aeroelastic model. The selected aero model must be a SuperGroup. The current SuperGroup is the default selection. An alternative SuperGroup may also be selected.



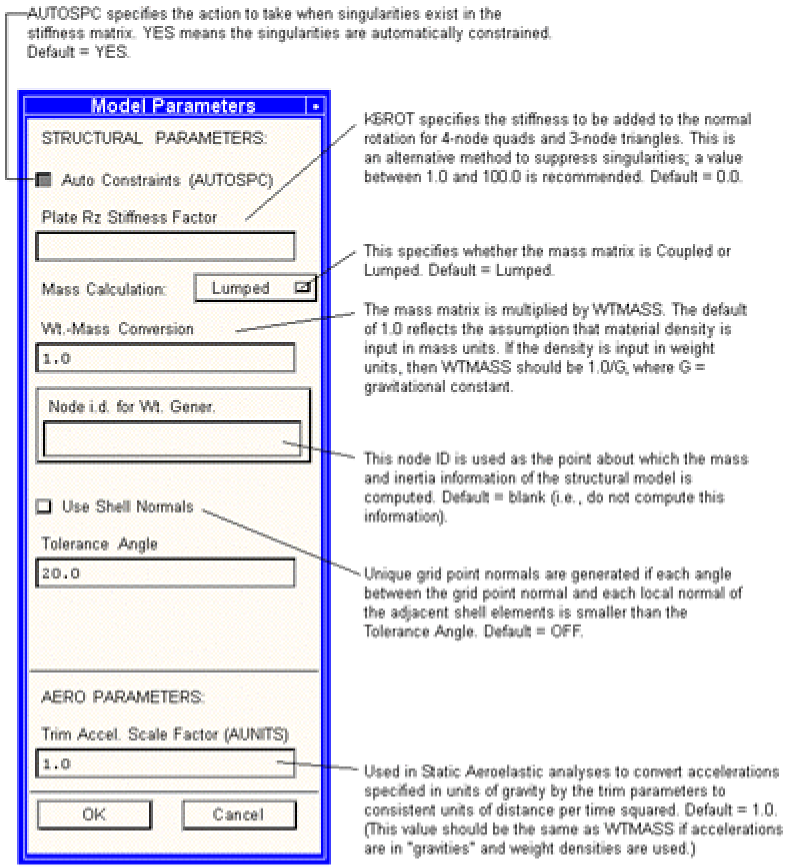
Select Structural Model

This subform is used to select the structural component of the aeroelastic model. As in standard structural analyses from MSC.Patran, users can select all structural elements in the database or a specific Group. The default selection is "Entire Model".



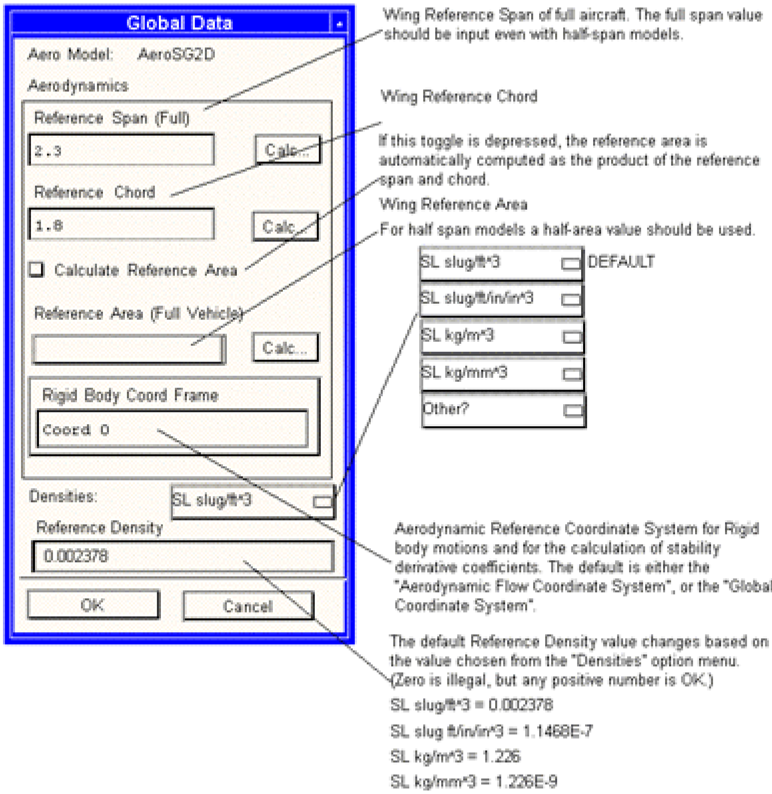
Model Parameters

Model Parameters allows the user to change the default model behavior and representation in MSC.Nastran.

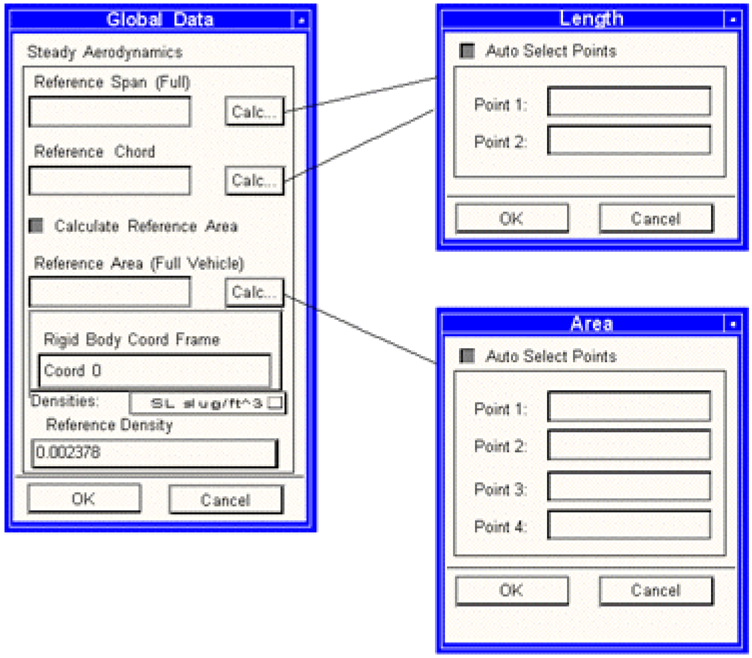


Global Data

This form is used to define the basic properties for static aeroelasticity.



These forms are subforms for Global Data provided as a convenience to users. Length and area calculations are performed based on selection of points from the graphics viewport.



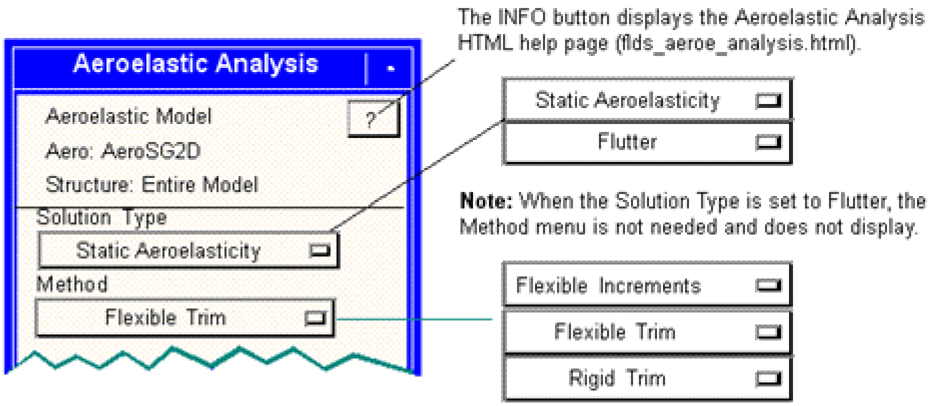
Analysis

The third step on the Aeroelasticity main form, after coupling the aero and structural models and defining the aeroelastic model, is the actual aeroelastic analysis. The following pages describe the procedure required to perform an aeroelastic analysis choosing a solution type of either Static Aeroelasticity or Flutter Analysis.

Note: This button is disabled if the current SuperGroup is a 3D SuperGroup.

Overview

When the user selects Analysis, the form is displayed in its default state with the Solution Type set to Static Aeroelasticity. Shown adjacent to this form are ALL the different Solution Types and Methods that are available when using this option.



Analysis Forms

Most of the Analysis forms and its subforms are shown and annotated in the following pages, grouped as follows:

- Aeroelastic Analysis
- Analysis Subforms
  - Target Databases
  - Subcase Create (for Static Aeroelasticity)
  - Trim Parameters
  - FEM Rigid Body DOFs
  - Output Requests
  - Direct Text Input
- Subcase Create (for flutter)
  - Mach-Frequency Pairs
  - Flutter Parameters
  - Output Requests
  - Direct Text Input
  - Select Superelements
  - Real Eigenvalue
  - Complex Eigenvalue
- Subcase Select
- Existing Jobs
- Job Parameters
- Analysis Manager

Aeroelastic Analysis

The basic Aeroelastic Analysis form is the same for any of the selected Methods associated with Static Aeroelasticity. Please note that the Flexible Increment Method is not allowed to be mixed with the Rigid or Flexible Trim Method in a single analysis job. If either a Flexible Increment or Rigid/Flexible Trim Subcase is selected for analysis, the other option(s) will be disabled.

Three Methods are available with Static Aeroelasticity: Flexible Increments, Flexible Trim, and Rigid Trim:

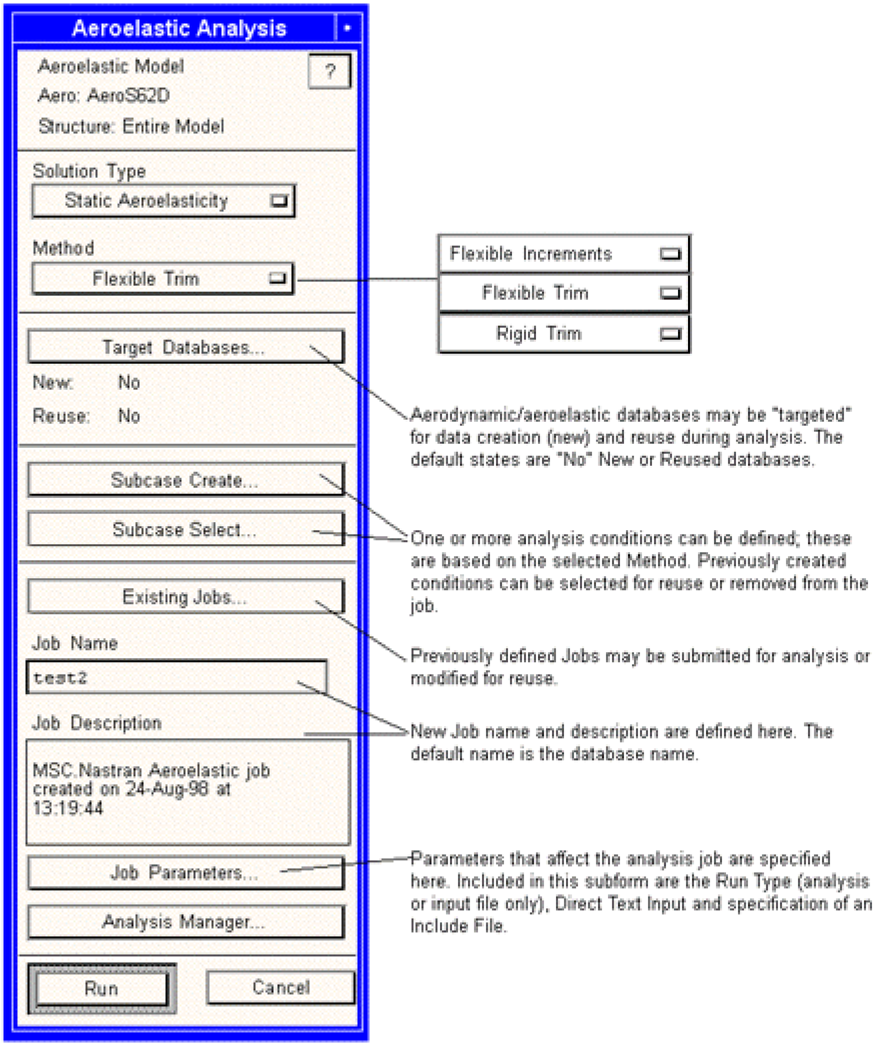
Flexible Trim is the most common Static Aeroelastic analysis Method performed in MSC.Nastran. This is the calculation of the trim parameters (vehicle rigid body motion and control device) settings required to maintain the desired aircraft attitude, rate and/or acceleration. The result of this calculation is not only the trim parameters but also the resulting external loads on the air vehicle. These loads include the components of aerodynamics (rigid), inertia (structural mass), flexible (structural flexibility) and trim (use of control devices or rigid body motion).

Rigid Trim is identical to Flexible Trim, with the exception that structural flexibility effects are ignored. The vehicle inertial properties are obtained from the structural model, identified as part of the aeroelastic model. Note that the structural model is still elastic (it deforms); "rigid" implies that the increment in aerodynamic load induced by those deformations is ignored (no aeroelastic feedback).

Flexible Increments is the calculation of flexible load increments due to unit perturbations of each trim parameter, one at a time. These are always calculated as part of a Flexible Trim analysis. By requesting a Flexible Increment Method, the trim solver will not be invoked and the Flexible Increments may be optionally stored on the aeroelastic database. If stored, these may then be reused for rapid trim solutions as the calculations up the point of calculation of Flexible Increments is no longer required.

The Basic Flutter Analysis capability allows you to define MSC.Nastran Flutter Analysis: PK, PKNL, K, and KE.

In addition to the Flutter subcase there are forms to allow the user to select the defined hardpoint aerodynamics, Mach Number and Reduced Frequency pair, for aerodynamic database population. Note that the creation of the data and the use of the data are separate.



Note: When the Solution Type is set to Flutter, the Method menu is not needed and does not display.

Target Database

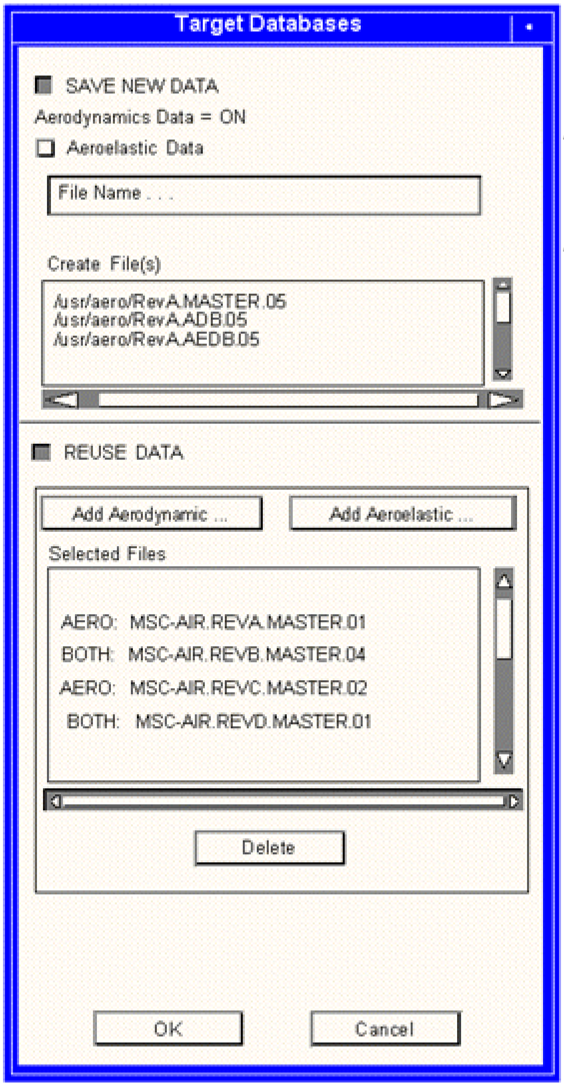
MSC.FlightLoads has been designed to simplify the creation and reuse of archival collections and aerodynamic and static aeroelastic dates. This capability has been in MSC.Nastran aeroelasticity but required a very knowledgeable user to ensure the correct data were saved and significant DMAP work was needed to reuse. Now these features are automated. Appendix E and F contain a more detailed description of the contents of the two collections. The aerodynamic database contains no structural information and all the aerodynamic model geometry mesh and rigid aerodynamics that are needed for any subsequent aeroelastic analysis.

The aeroelastic database requires the aerodynamic database, and includes the aerodynamic data after it has been coupled to a particular structure. Of necessity, these data are a function of the selected dynamic pressures in addition to all the structural boundary conditions.

The aerodynamic database is useful to perform studies of a given configuration with more than one structural or spline representation. The aeroelastic database is associated with a single pair of math models, but can simulate any trimmed, quasi-static maneuver as a simple linear combination of the archival collection. It is useful to rapidly generate trimmed distributed loads for a large number of maneuvers within a given Mach and dynamic pressure (i.e., altitude and speed). Appendix C discusses the "unit solutions" or "flexible increment" that comprise this collection.

Aerodynamic or Aeroelastic databases may be created and selected for data reuse. If an Aeroelastic database is selected for reuse, its companion Aerodynamic database will be automatically selected.

Note:	This form is valid for both solution types.

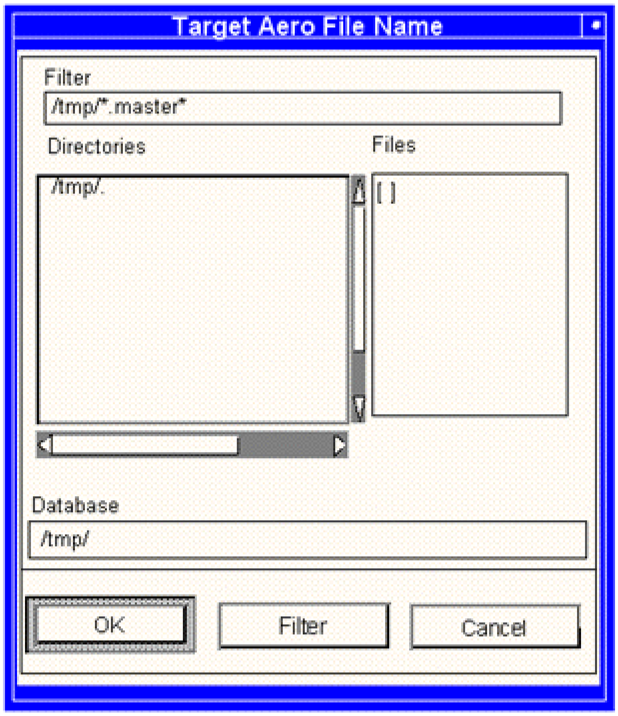


SAVE NEW DATA, when enabled, supports the creation of an aerodynamic and, optionally, an aeroelastic database, associated to a parent MASTER file. The File Name subform prompts the user for the MASTER name. The selected directory is searched for <name>.MASTER, <name>.ADB and, optionally, <name>.AEDB. The highest extension is identified and incremented to create the NEW DATA file names.

REUSE DATA, when enabled, supports the reuse of an aerodynamic and, optionally, an aeroelastic database, associated to a parent MASTER file. The Add Aerodynamic and Add Aeroelastic subforms prompt the user for the MASTER name. Add Aerodynamic only selects the Aerodynamic database (DBSET) associated with the MASTER file while Add Aeroelastic selects both the Aerodynamic and Aeroelastic database (DBSET's) associated with the MASTER file.

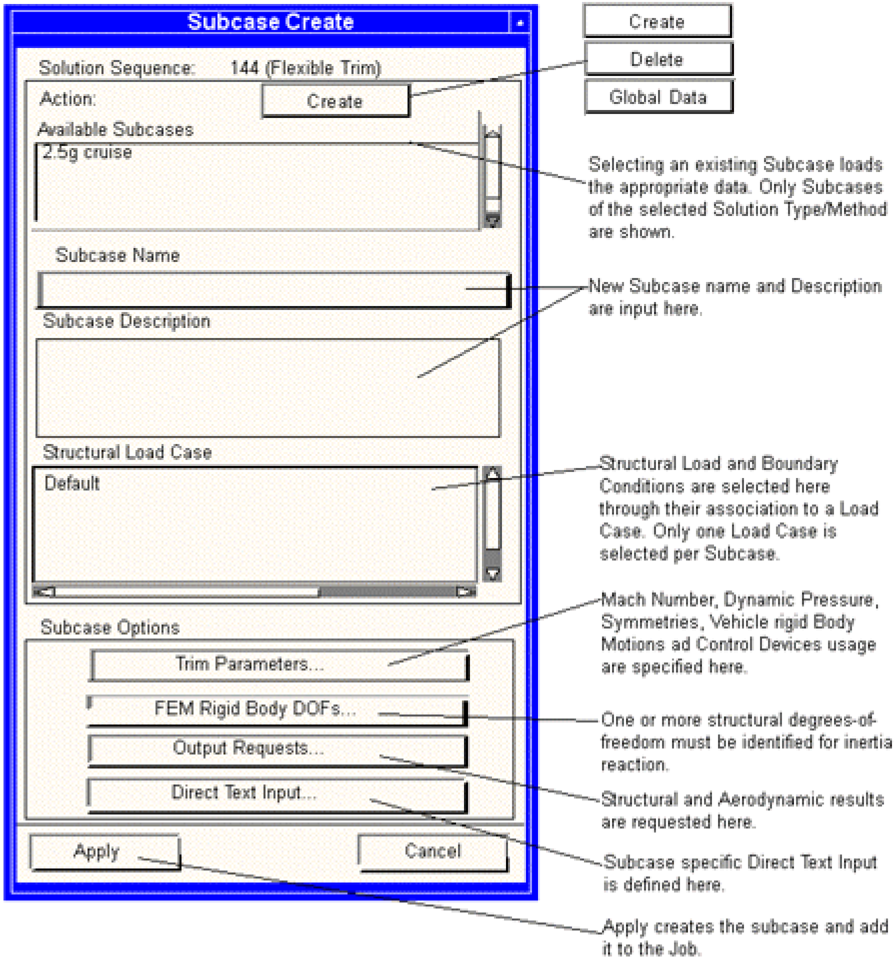
File Name or Add Aerodynamic or Add Aeroelastic

The label on the form changes based on which button on the Target Database form is selected. The form shown below is for the File Name. If Add Aerodynamic is chosen, the form title would read Select Aerodynamic MASTER File. If Add Aeroelastic is chosen, then the form title would read Select Aeroelastic MASTER File.



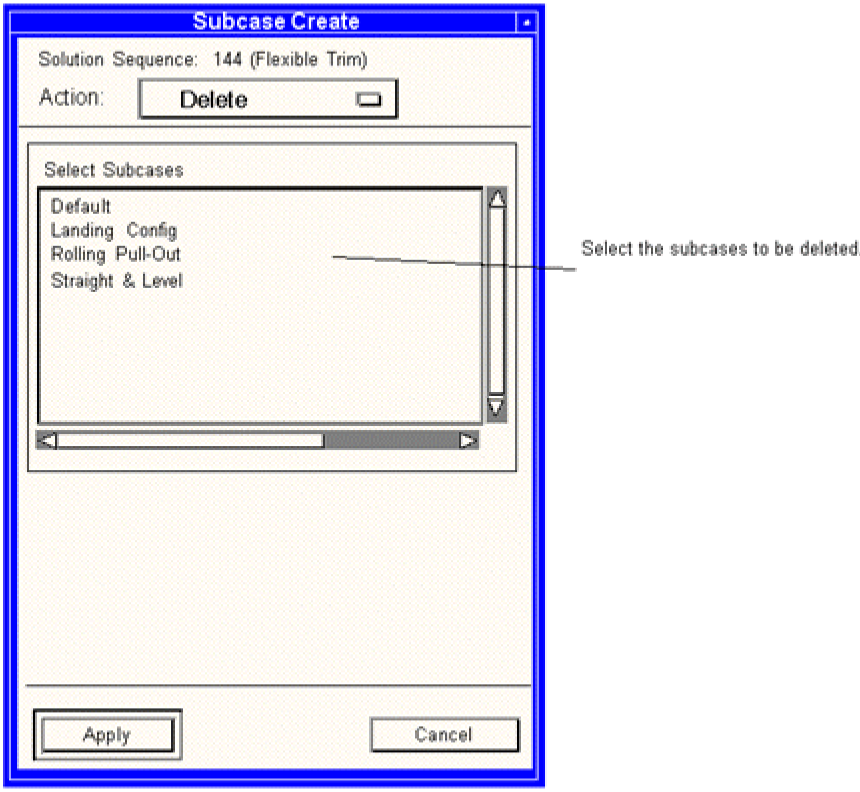
Subcase Create - (for Static Aeroelasticity) - Create

Subcase Create defines one or more analysis conditions based on the Selected Analysis Type/Method pair. The form shown below is valid for Flexible Trim, Flexible Increments and Rigid Trim Methods.



Subcase Create - (for Static Aeroelasticity) - Delete

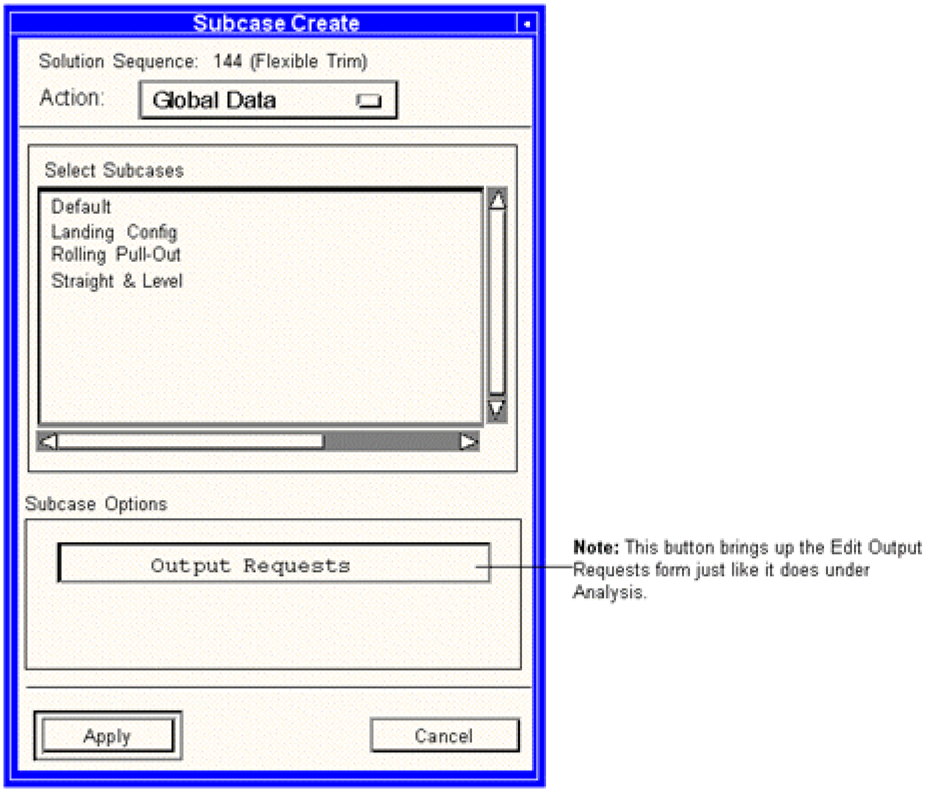
The form deletes any existing static Aeroelastic subcases from the database. This form is valid for all Methods and Flexible Trim, Flexible Increments, and Rigid Trim.



Note: This form is identical to the one under analysis which has had the "Action" option menu added to it.

Subcase Create - (for Static Aeroelasticity) - Global Data

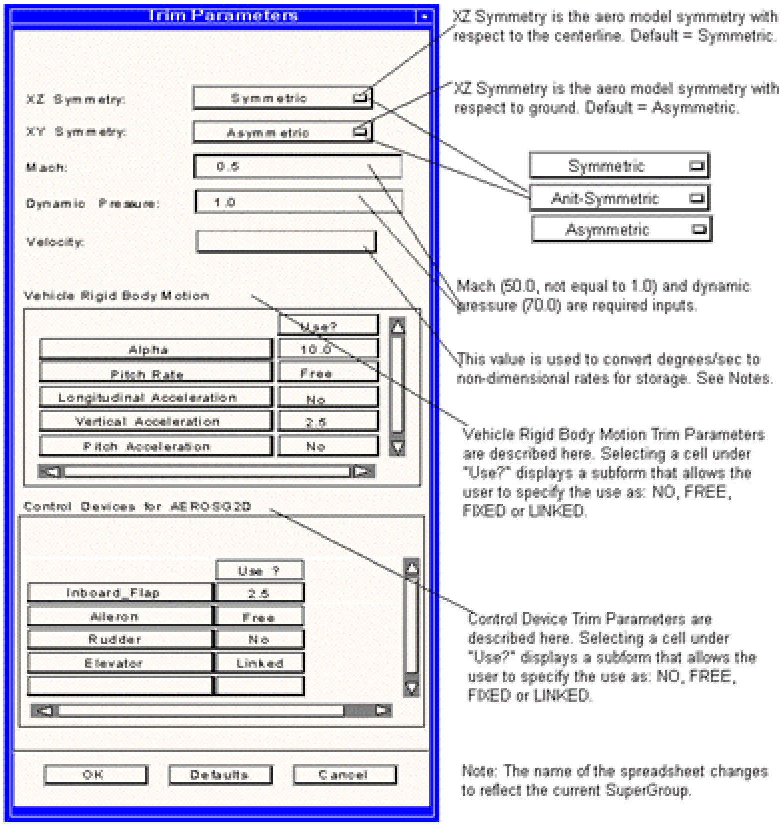
The form allows the users to globally change the Output Request for the selected Static Aeroelastic subcases. This form is valid for all Methods: Flexible Trim, Flexible Increments, and Rigid Trim.



Note: This form is identical to the one under analysis which has had the "Action" option menu added to it.

Trim Parameters

The Trim Parameters form, shown below, is valid for Flexible Trim, Rigid Trim and Flexible Increments.



Note:

1. Full models must have the XZ Symmetry option menu set to Asymmetric.
2. For symmetric and antisymmetric analysis, the model must be geometrically symmetric about the ZX plane of the aerodynamic (flow) coordinate system.
3. The Velocity is only enabled when Dimension, Angles, and Rates (on the options form) is toggled.
4. The Velocity takes a real number and uses it in the following equation to convert the entered degrees/sec in the spreadsheet to nondimensional rates before storing them in the database.

Equation:

$$\text{Dimensional Rate} = \frac{(\dot{deg}/s)(length)}{2 \times Velocity}\left(\frac{rad}{deg}\right)$$

where:

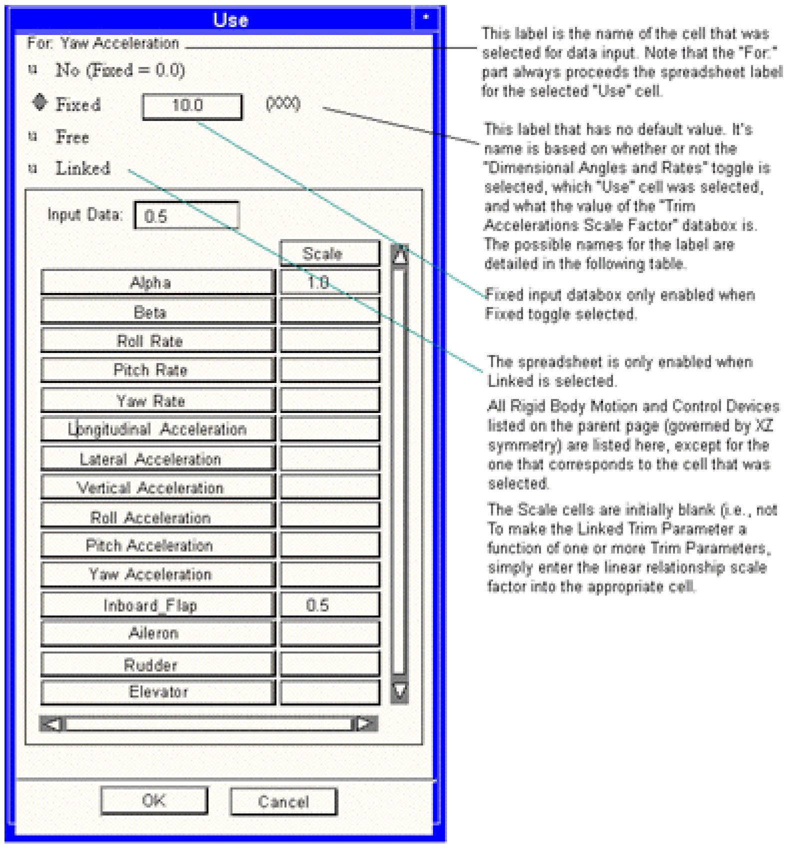
Length is the reference chord for symmetric rates and is the reference span for antisymmetric. The reference lengths are defined in the Aeroelastic - Model -> Global Data form.

5. The Vehicle Rigid Body Motion Trim Parameters listed on the lower half of this form, are dependent on the value of XZ Symmetry. The Symmetric parameters are: alpha (angle of attack), pitch rate, longitudinal acceleration, vertical acceleration and pitch acceleration. The Antisymmetric parameters are beta (side slip), roll rate, yaw rate, lateral acceleration, roll acceleration and yaw acceleration. Asymmetric parameters include those from both Symmetric and Antisymmetric. Only these Angles and Rates that have been created will be available on the spreadsheet.
6. The Control Device Trim Parameters listed on the lower half of this form shows the created control devices for the current SuperGroup.
7. Each Trim Parameter can have one of four uses: NO, FREE, FIXED, LINKED. Trim Parameters that use NO are excluded from the trim solution. FIXED Trim Parameters have a constant value during trim. FREE Trim Parameters are treated as unknowns during the trim analysis. LINKED Trim Parameters are determined by the values of one or more Trim Parameters through linear superposition:

   ```
   Linked Trim Parameter = C1*Trim_Parameter_1+C2*Trim_Parameter_2+...
   ```

Trim Parameter Use Subform

When focus is placed in a Trim Parameter “Use?” Cell on the preceding page, the following input form displays.

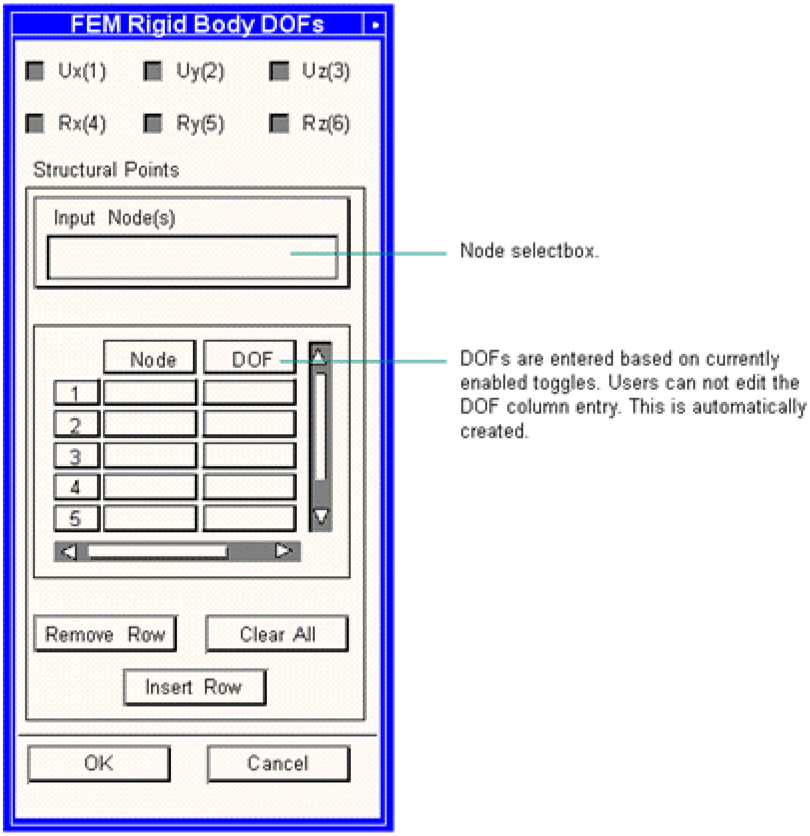


The following table displays the possible values for the label on the previous Use form.

Use Cell Name	Dimensional &Scale Factor = 1	Dimensional &Scale Factor != 1	NonDimensional &Scale Factor = 1	NonDimensional &Scale Factor != 1
Alpha	(Degrees)	(Degrees)	(Rads)	(Rads)
Beta	(Degrees)	(Degrees)	(pb/2v)	(pb/2v)
Roll Rate	(degs./sec.)	(degs./sec.)	(qc/2v)	(qc/2v)
Pitch Rate	(degs./sec.)	(degs./sec.)	(rb/2v)	(rb/2v)
Yaw Rate	(degs./sec.)	(degs./sec.)	(rb/2v)	(rb/2v)
Longitudinal Acceleration	(length/s/s)	(length/s/s)	(length/s/s)	(length/s/s)
Lateral Acceleration	(length/s/s)	(length/s/s)	(length/s/s)	(length/s/s)
Vertical Acceleration	(length/s/s)	(length/s/s)	(length/s/s)	(length/s/s)
Roll Acceleration	(degs./s/s)	(degs./s/s)	(Rads/s/s)	(Rads/s/s)
Pitch Acceleration	(degs./s/s)	(degs./s/s)	(Rads/s/s)	(Rads/s/s)
Yaw Acceleration	(degs./s/s)	(degs./s/s)	(Rads/s/s)	(Rads/s/s)
Control Devices	(Degrees)	(degs./s/s)	(Rads)	(Rads)

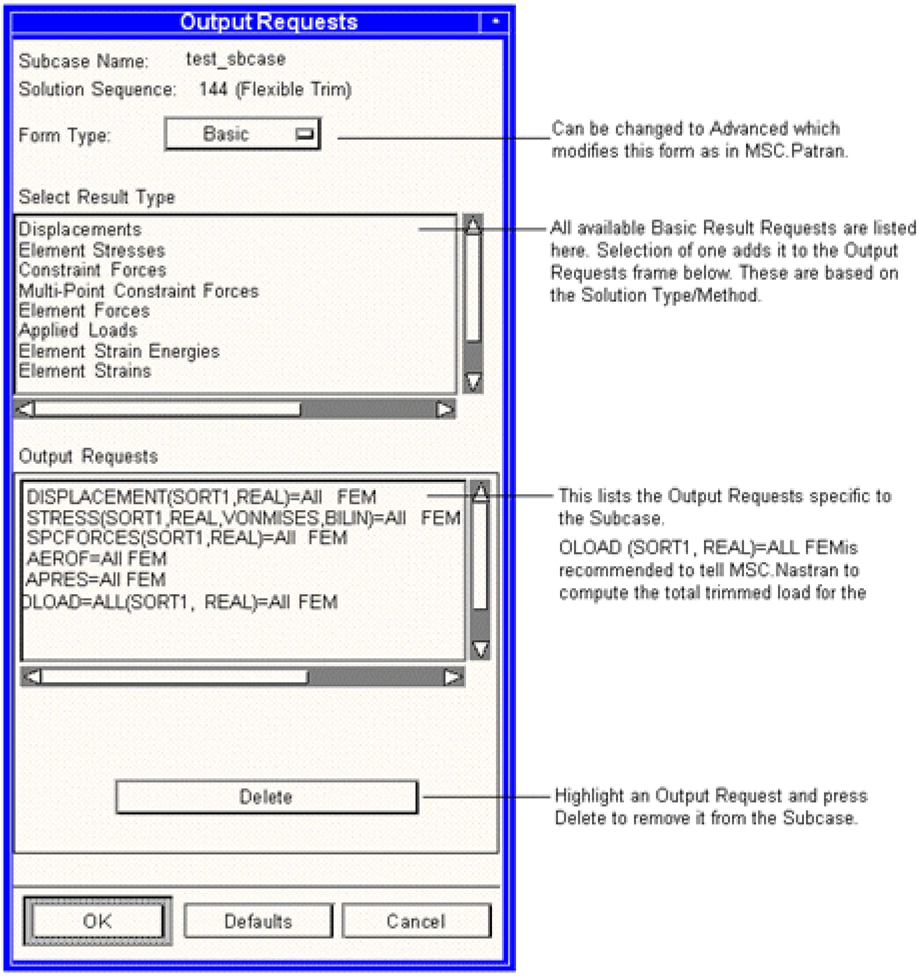
FEM Rigid Body DOFs

Static Aeroelastic analysis allows for restraint of one or more structural degree-of-freedom for inertia support. This DOF is associated with a structural node. The node should be associated with a heavy piece of structure as it is used to "support" the aircraft inertia during the specified condition. This form results in the creation of an MSC.Nastran SUPORT1 entry, referenced by the subcase.



Output Requests (Basic)

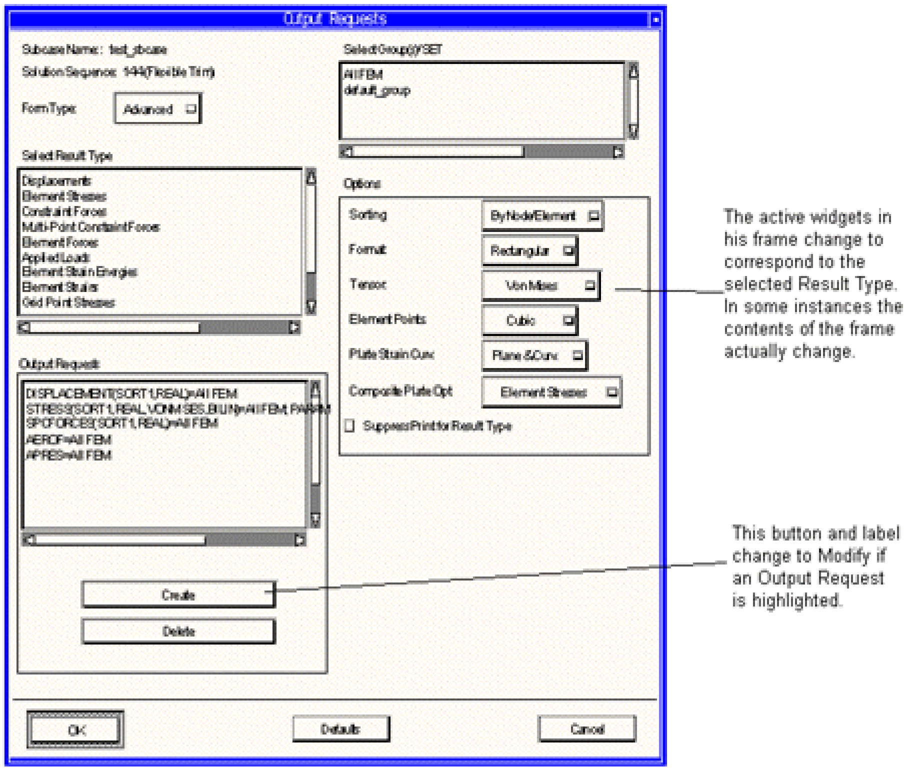
Basic Output Requests allow users to select aerodynamic and structural results for all applicable nodes and elements.



Note: The Output Request form is not available for Flexible Increments. These are automatically written to an XDB file unless otherwise indicated under Job Parameters (i.e., "None" causes the unit increments to not be stored on the XDB file).

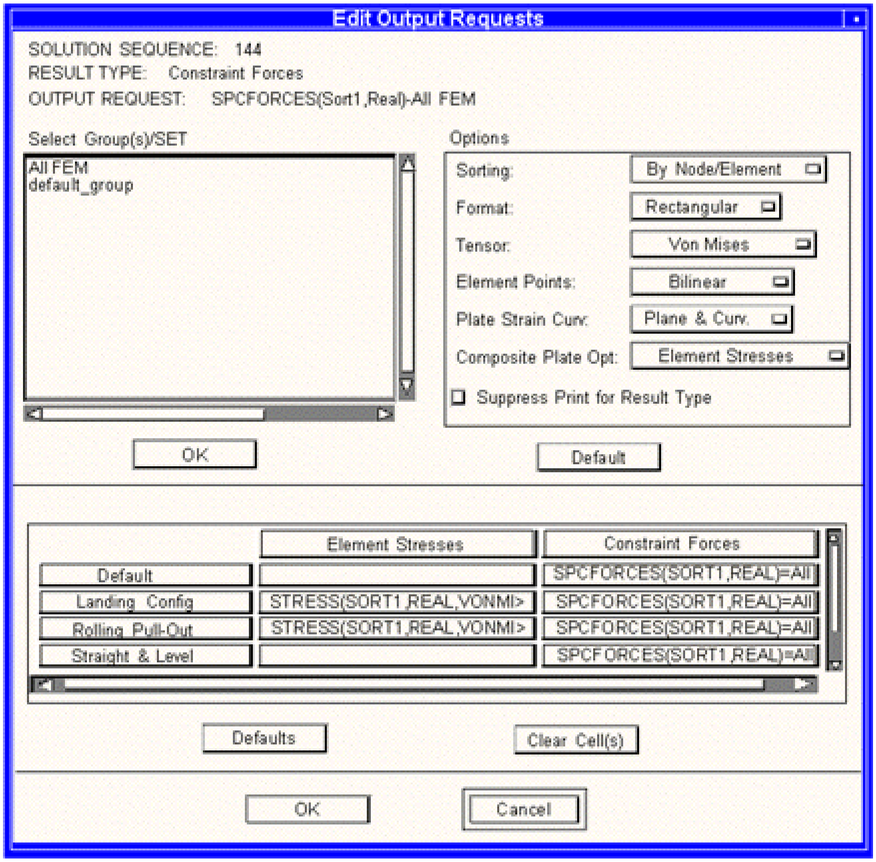
Output Requests (Advanced)

This form gives the user access to create the Output Request exactly as they need it.



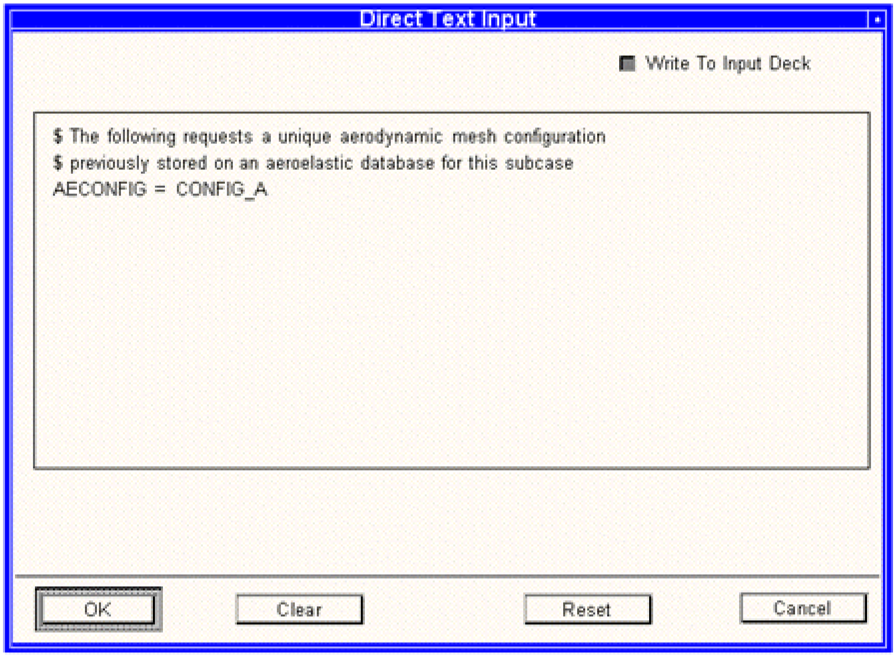
Global Data - Edit Output Request

This form allows users to modify defined Output Request for multiple subcases globally.



Direct Text Input

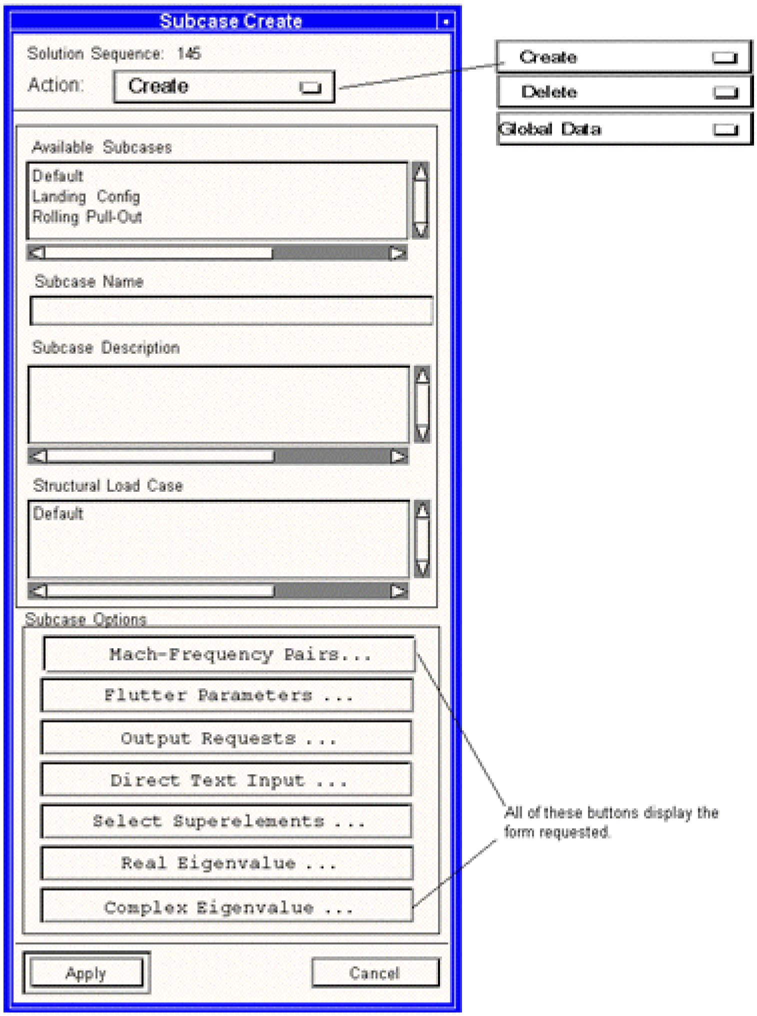
Users can provide one or more lines of Subcase specific lines for inclusion in the analysis job.



Note:	This form is identical to the current Direct Text Input form under Subcase Create in the Analysis application.

Subcase Create - (for Flutter) - Create

Subcase Create defines one or more analysis conditions on the selected Analysis Type.



Note:

1. Selecting an existing Subcase loads the appropriate data into the form.
2. The "Output Request" and "Direct Text Input" forms are the same as when the selected "Solution Type" is "Static Aeroelasticity".
3. The "Select Superelements" button is only active if Superelements are present in the DB. (This cannot be added to Static Aero until changes are made to MSC.Nastran.)
4. By default the "Complex Eigenvalue" button is grayed out. It is only active when the "Method" on the "Flutter Parameters" form is set to K.

Subcase Create - (for Flutter) - Delete

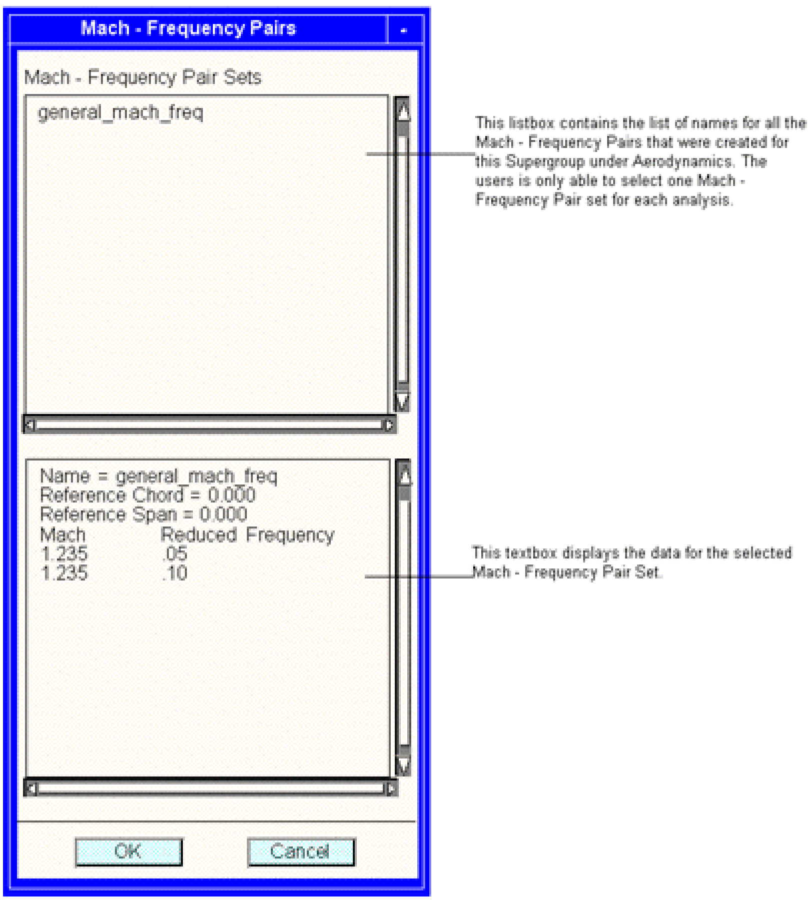
This form (not shown) is exactly like the one for Static Aeroelasticity except that it allows users to delete any existing Flutter Subcases from the database.

Subcase Create - (for Flutter) - Global Data

This form (not shown) is exactly like the one for Static Aeroelasticity except that it allows the user to globally change the output request for the Selected Flutter subcases.

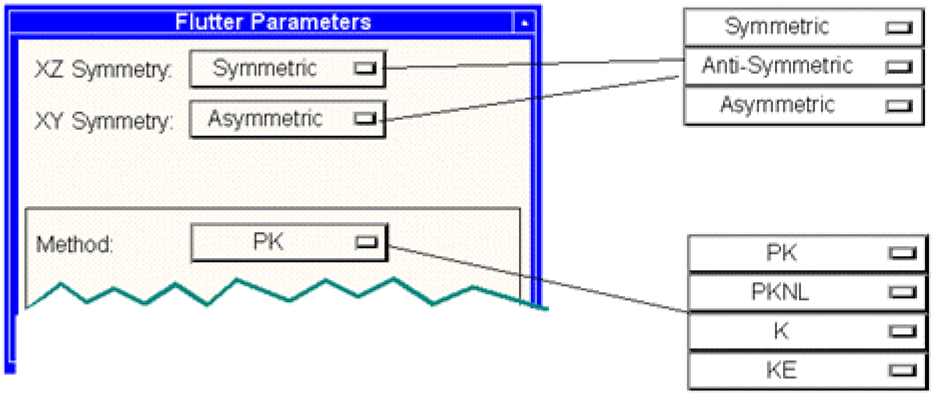
Mach/Frequency Pairs

This form is used to select which Mach-Frequency Pair is going to be used in the Flutter Analysis.

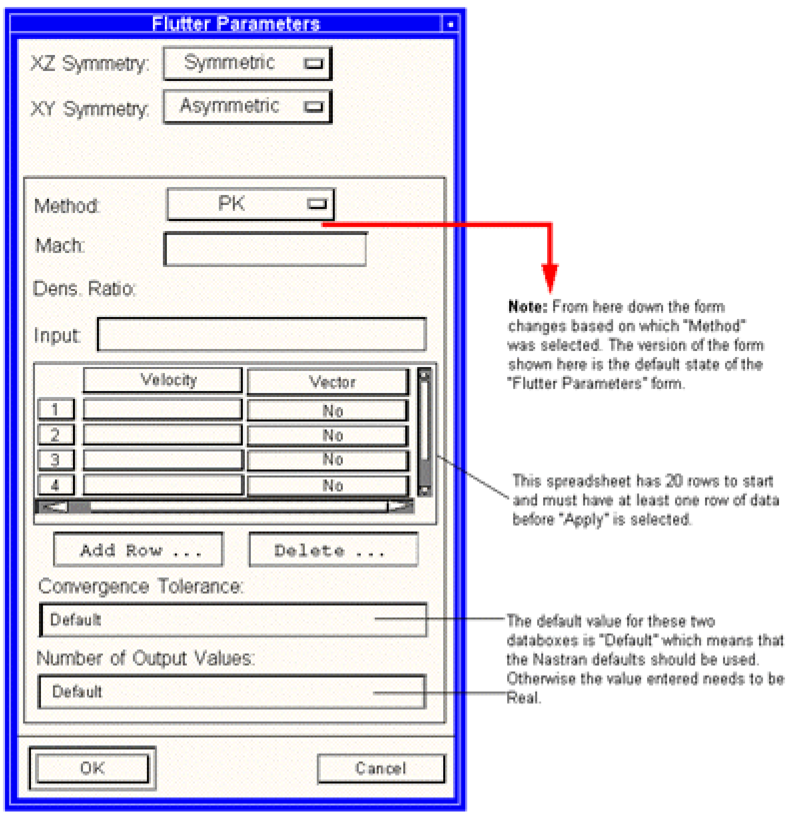


Flutter Parameters

The top half of the Flutter Parameters form, shown below, is the same for all four methods: PK, PKNL, K, and KE.



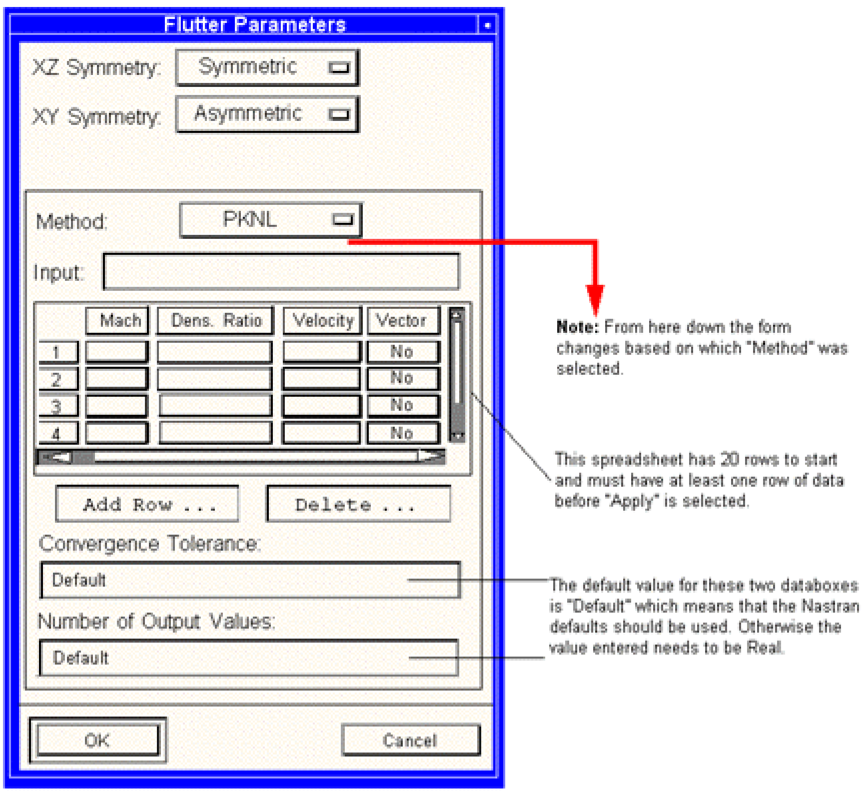
Flutter Parameters: PK Method



Note:

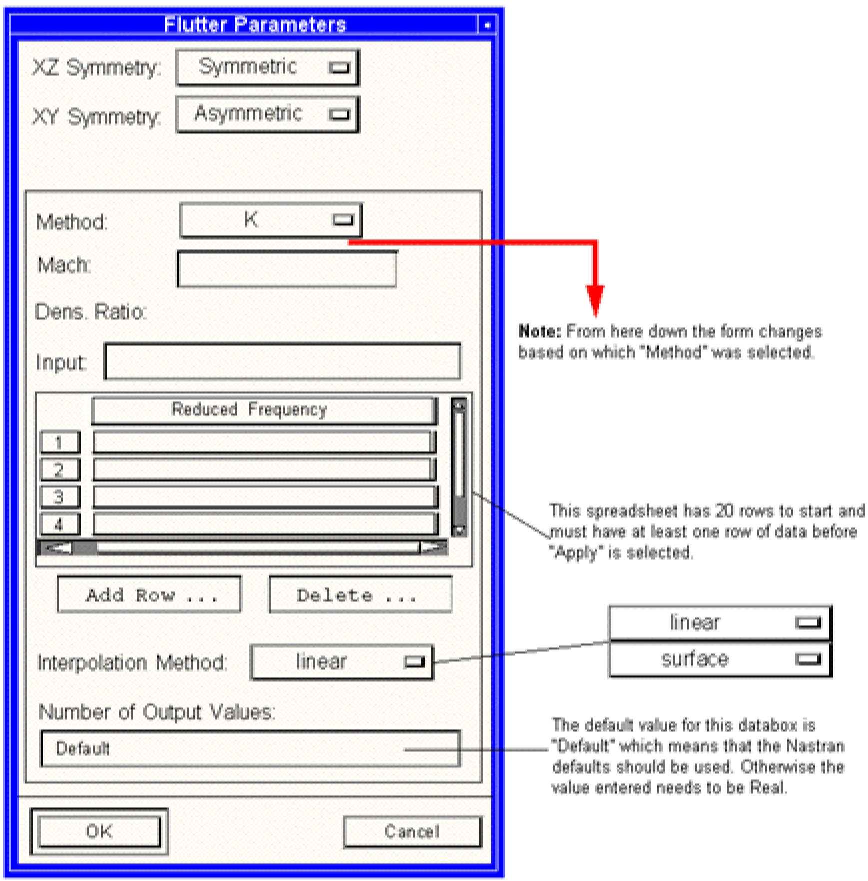
1. "Mach" and "Dens. Ratio" are real databoxes that are blank by default.
2. The "Vector" cells on the spreadsheet toggle between "No" and "Yes" when selected.
3. The "Input" databox accepts a positive real value that gets entered into the selected "Velocity" cell when the Return/Enter key is pressed. If the databox is empty when the Return/Enter key is pressed, the selected cell is cleared.
4. The "Add Row" button allows a new row to be added to the spreadsheet while the "Delete" button deletes the selected row.

Flutter Parameters: PKNL Method



Note:	
	1. The "Vector" cells on the spreadsheet toggle between "No" and "Yes" when selected. 2. The "Input" databox accepts a positive real value that gets entered into the selected "Mach", "Dens Ratio", or "Velocity" cell when the Return/Enter key is pressed. If the databox is empty when the Return/Enter key is pressed, the selected cell is cleared. 3. For the "Mach" cells zero is also an allowed input. 4. The "Add Row" button allows a new row to be added to the spreadsheet while the "Delete" button deletes the selected row.

Flutter Parameters: K and KE Method:



Note:	1. "Mach" and "Dens. Ratio" are real databoxes that are blank by default. 2. The "Input" databox accepts a positive real value that gets entered into the selected "Reduced Frequency" cell when the Return/Enter key is pressed. If the databox is empty when the Return/Enter key is pressed, the selected cell is cleared. 3. The "Add Row" button allows a new row to be added to the spreadsheet while the "Delete" button deletes the selected row. 4. This form is the same when **Method** = **KE**.

Output Requests

This form (not shown) is exactly like the one for Static Aeroelasticity and allows the user to create Output Requests.

Basic output request give the use the ability to select aerodynamic and structural results for all applicable nodes and elements.

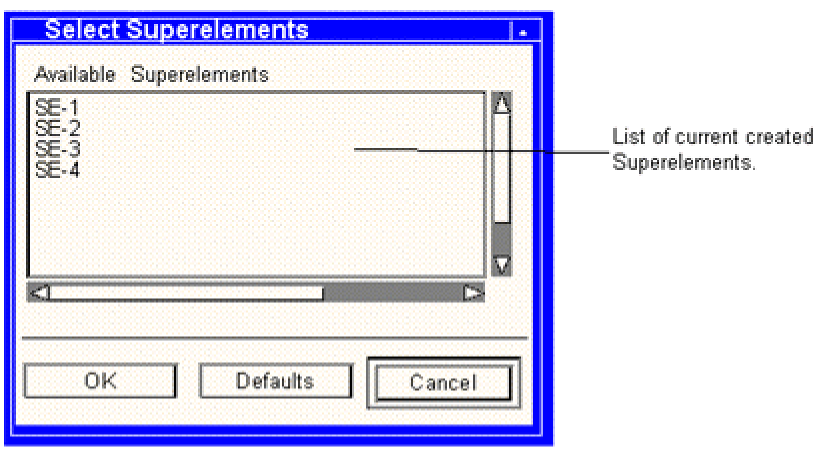
Direct Text Input

This form (not shown) is exactly like the one for Static Aeroelasticity and allows the user to create Output Requests.

Users can provide one or more lines of subcase specific lines for inclusion in the analysis.

Select Superelements

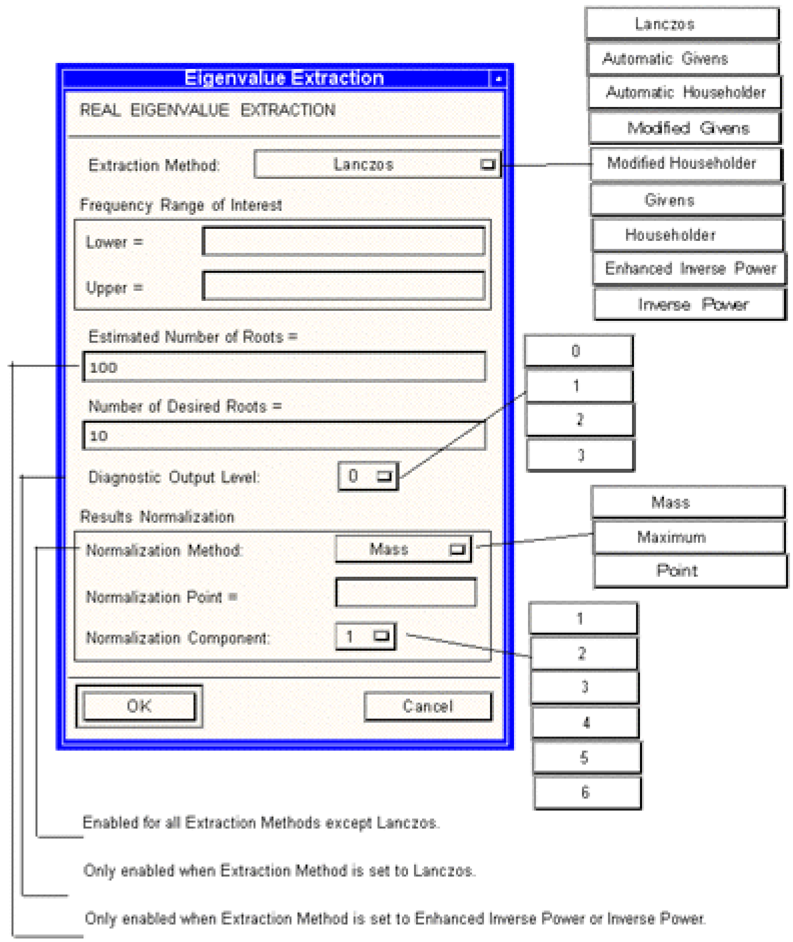
Users select which Superelements are to be included in the analysis job.



Note:	This form is identical to the current Select Superelement form under Subcase Create in the Analysis application.

Eigenvalue Extraction - Real

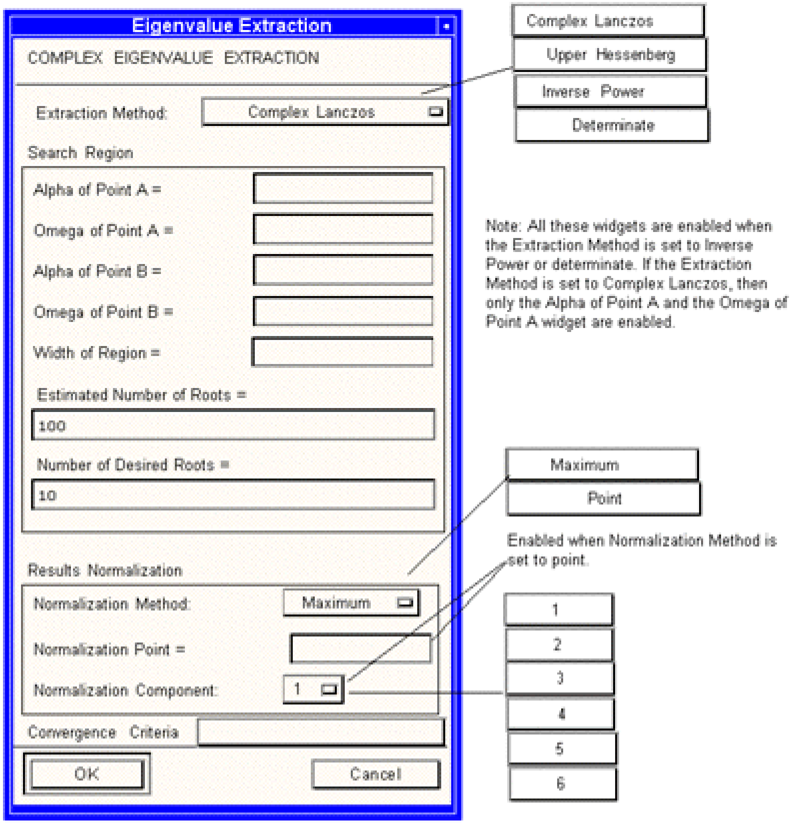
This form allows the users to define Real Eigenvalue data to be used in the analysis.



Note: This form is identical to the current Eigenvalue Extraction form that comes up off the Real Eigenvalue button selection under SOL 110 Subcase Parameters in the Analysis application.

Eigenvalue Extraction - Complex

This form allows the users to define complex Eigenvalue data to be used in the analysis.

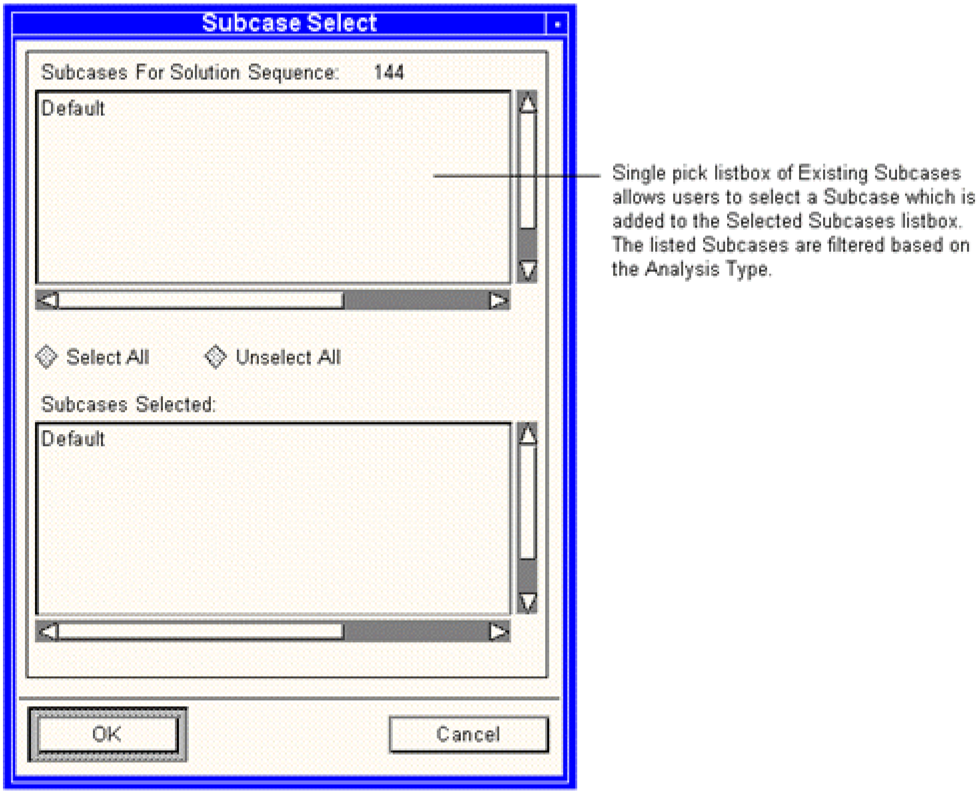


Note: This form is identical to the current Eigenvalue Extraction form that comes up off the Complex Eigenvalue button selection under SOL 110 Subcase Parameters in the Analysis application.

Subcase Select

This form allows users to review and alter the selected subcases prior to analysis. All created subcases are listed in the top listbox. As subcases are selected, they are added to the bottom listbox for subsequent use in analysis.

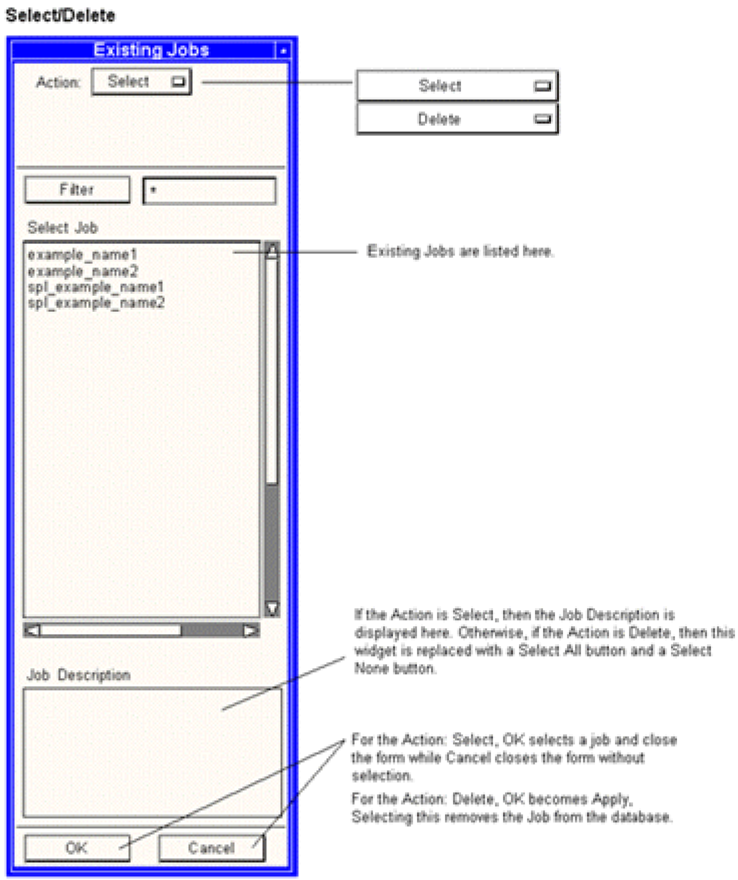
Trim subcases (Rigid and Flexible) can not be mixed with Flexible Increment subcases. After the first subcase is selected for analysis, the other type is automatically removed from the available listbox.



Existing Jobs

Previously created analysis Jobs may be selected for analysis, modified for subsequent analysis or removed from the database.

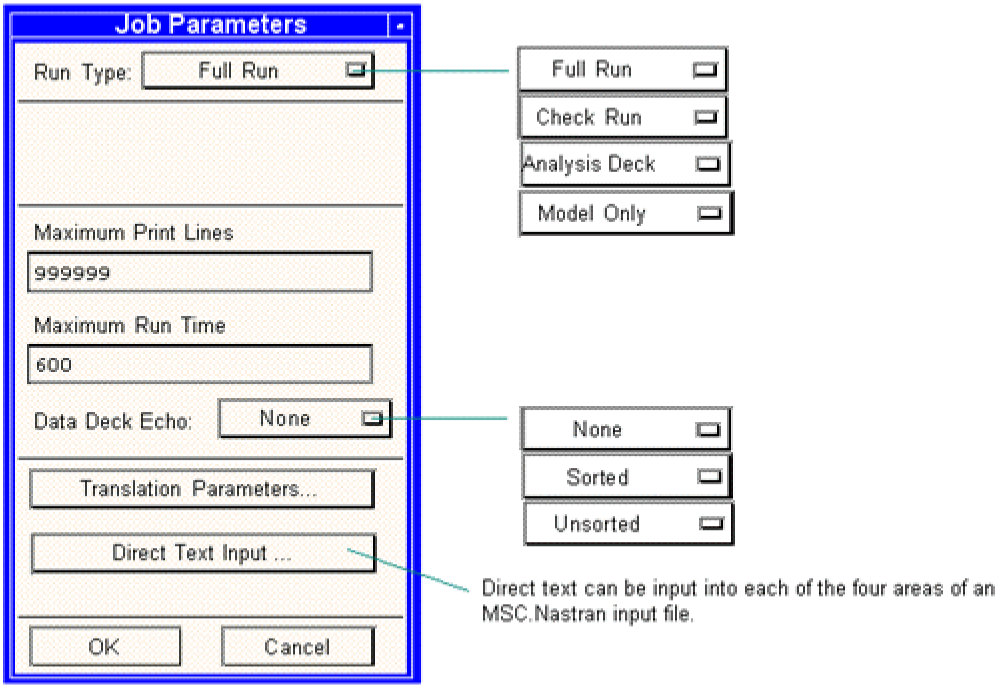
Select/Delete



Job Parameters

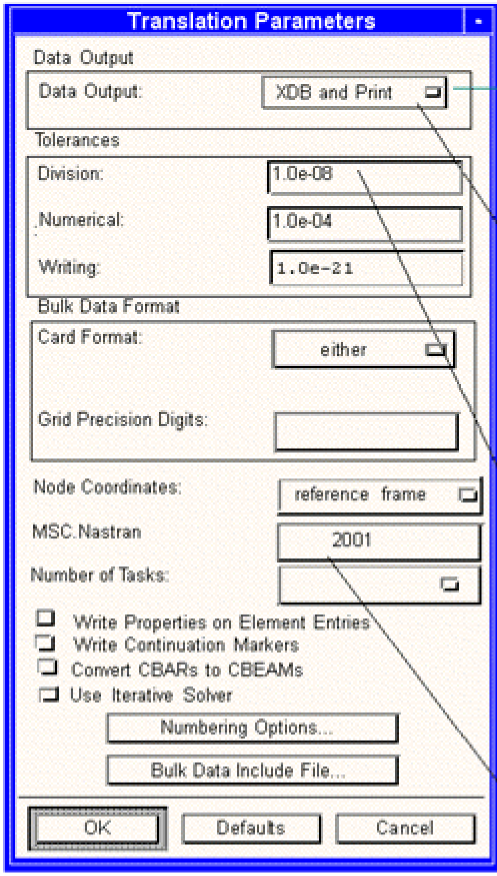
Job Parameters specify the Run Type (i.e., whether an analysis is performed or just the creation of an input file), the resource limitations, print behavior and input file creation behavior. An Include File can be specified in the Translation Parameters subform.

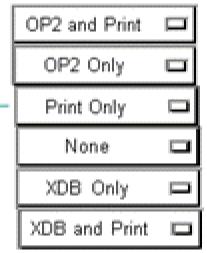
Four Run Types are available: Full Run, Check Run, Analysis Deck and Model Only. A Full Run causes an analysis to be performed while a Check Run instructs MSC.Nastran to check out the created input deck. Analysis Deck causes the creation of a complete input file while Model Only results in the creation of only the bulk data section.



Translation Parameters

This subordinate form appears when the Translation Parameters button is selected.





Defines type of data output. "Print" specifies output of data to the MSC.Nastran print file (`*.f06`). "XDB" specifies output of data to a MSC.ACCESS database (`*.xdb`). "OP2" specifies output of data to ON OP2 file.

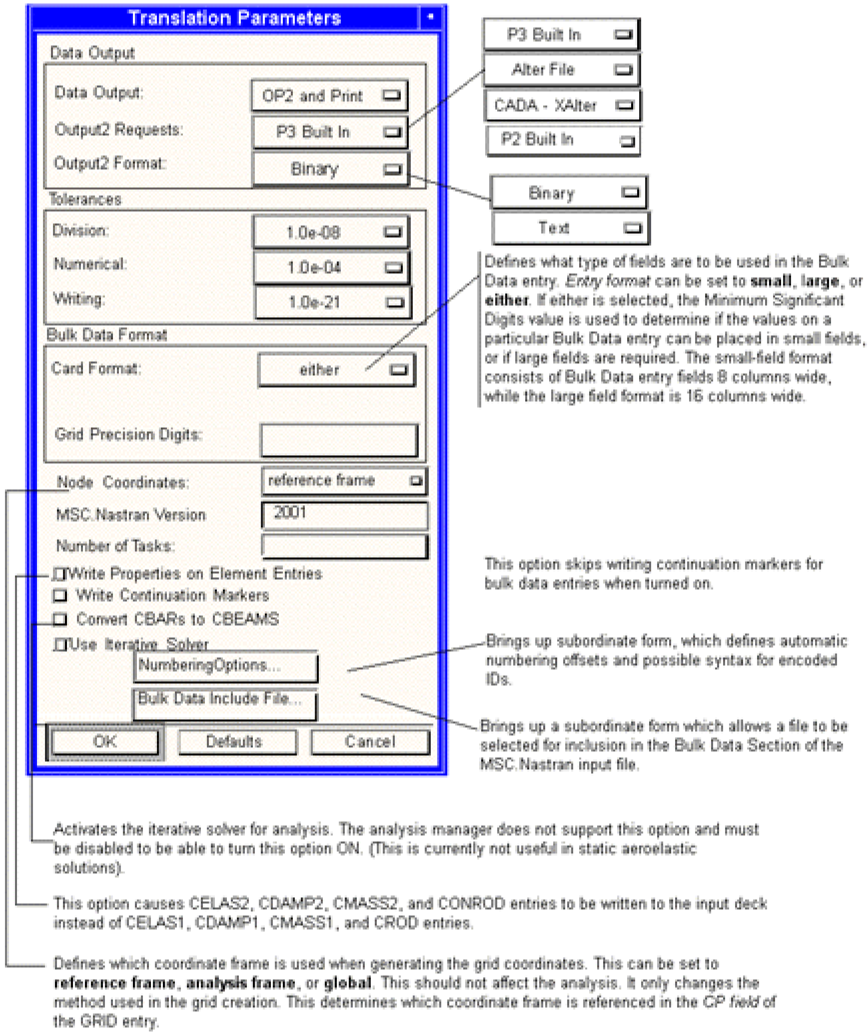
Note: Selection of "OP2 and Print" or OP2 Only" will cause the "OUTPUT2 Requests" and the "OUTPUT2 Format" options menus to be displayed in this frame. (See next page for options.)

Defines various tolerances used during translation.

1. *Division* is used to prevent divide-by-zero errors.
2. *Numerical* is used to determine if two real values are equal.
3. *Writing* is used to determine if a value is approximately zero when generating a Bulk Data entry field.

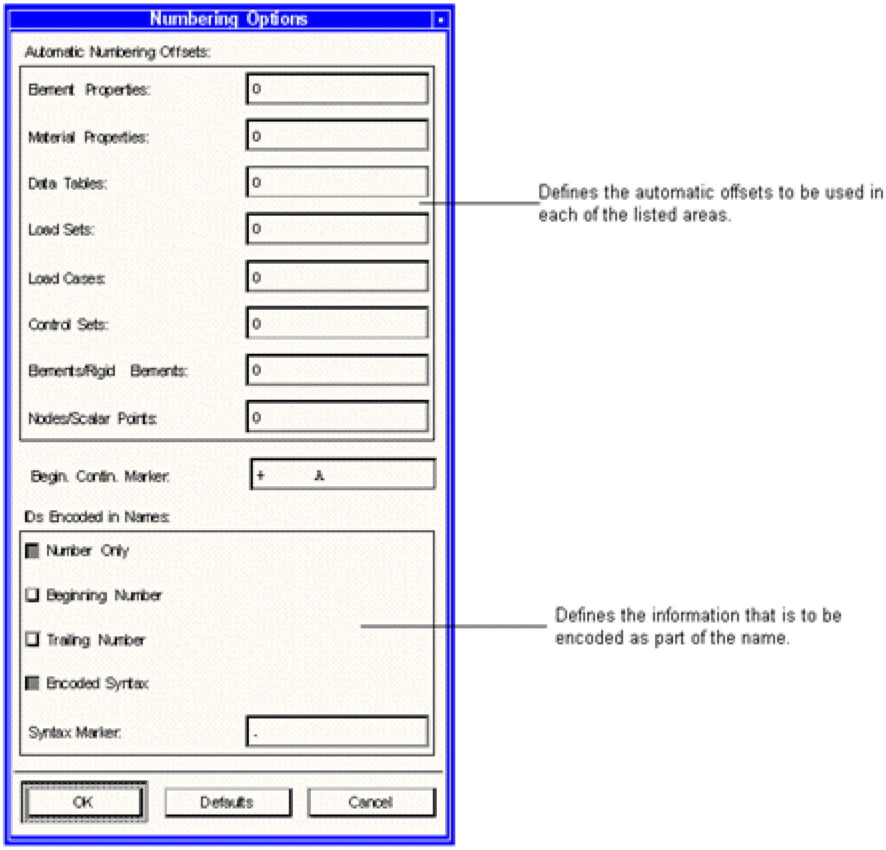
MSC.Flightloads and Dynamics requires this Version be no less than 70.6.

V2001 requires Version 2001.



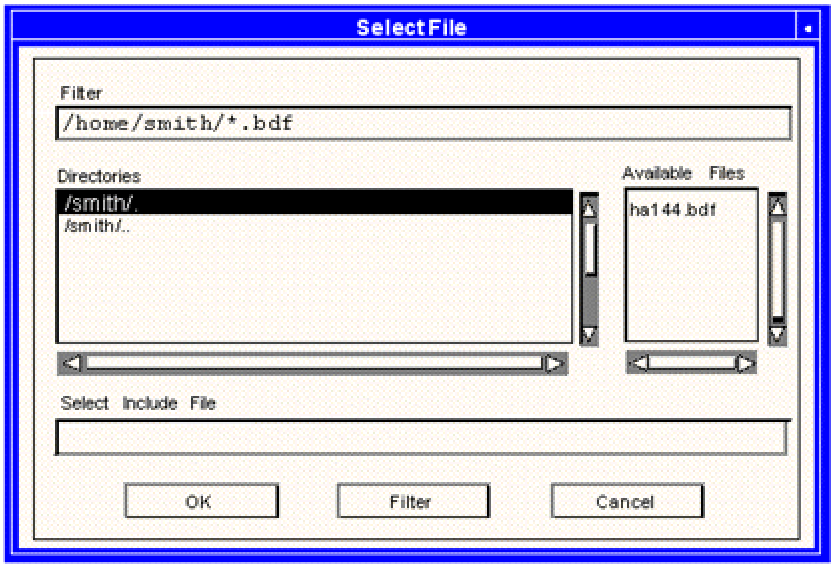
Numbering Options

This subform appears when the Numbering Options button is selected.



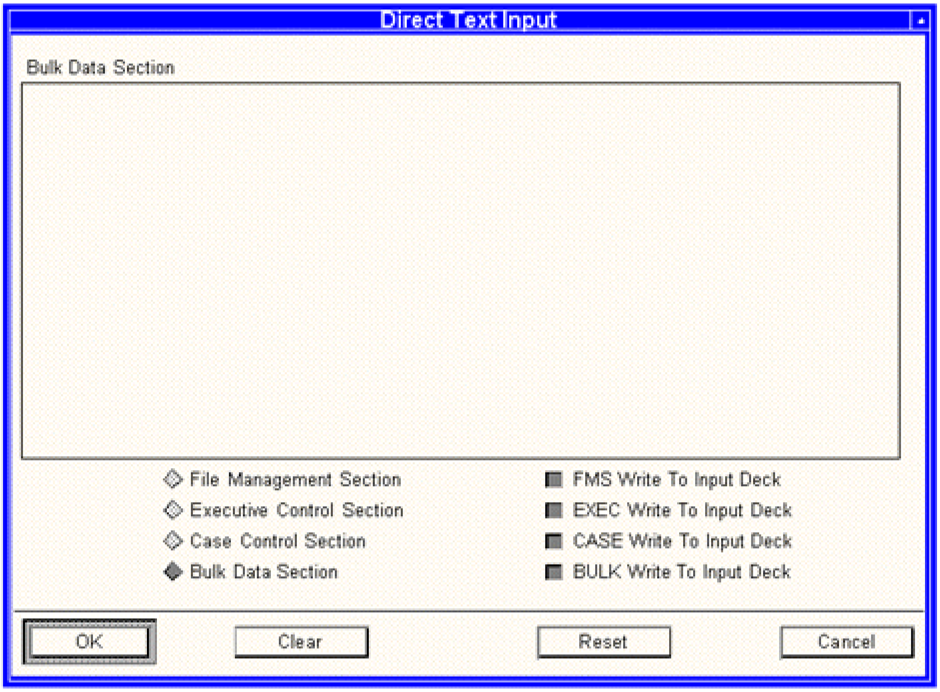
Bulk Data Include File

This subform appears when the Bulk Data Include File button is selected.



Direct Text Input

One or more lines of text can be defined for the four sections of the MSC.Nastran input file: File Management Section (FMS), Executive Case Control (EXEC), Case Control (CASE) and Bulk Data.


Direct Text Input
Bulk Data Section
File Management Section
Executive Control Section
Case Control Section
Bulk Data Section
FMS Write To Input Deck
EXEC Write To Input Deck
CASE Write To Input Deck
BULK Write To Input Deck
OK
Clear
Reset
Cancel

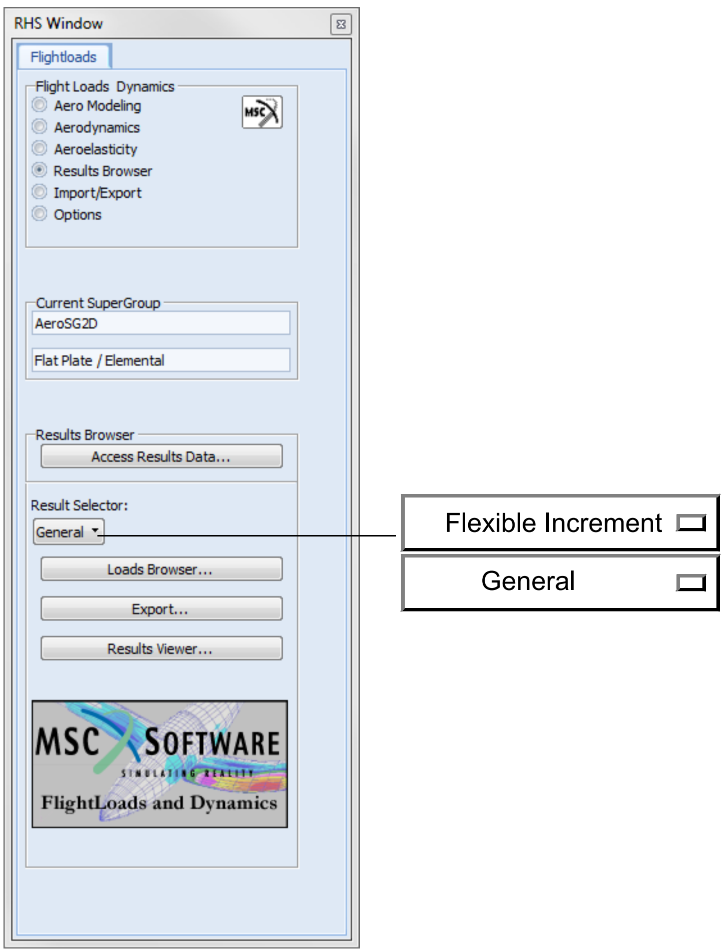
6 Results Browser

Introduction

The Results Browser allows users to query previously computed external loads information.

The module is separated into four Results Browser applications and one Data Attachment application that are selectable by clicking on one of the ellipsis buttons at the bottom of the form.

- Attach Results Data - Accesses loads data from XDB Results files and aerodynamic/aeroelastic databases.
- Flexible Increment Filter - Filters results down to a single unit solution.
- Loads Browser - Does Running Load, XY plots and Load Summations.
- Export - Exports loads data in the form of MSC.Nastran Force/Moment entries.
- Results Viewer - Views the Results for the selected unit solution or for the General case.



Parameter	Description
Results Selector	Gives you the option for two different tool choices: Flexible Increment or General.

Note:	Four Results Browser Application Sections: ■ Flexible Increment Filter ■ Loads Browser ■ Export ■ Results Viewer

Attach Results Data

The first step in browsing loads is to select the data source. If the data resides external to the MSC.Patran database, it must first be "accessed". External data sources include Aerodynamic/Aeroelastic Databases and aeroelastic analysis results files in the HDF5 and XDB formats.

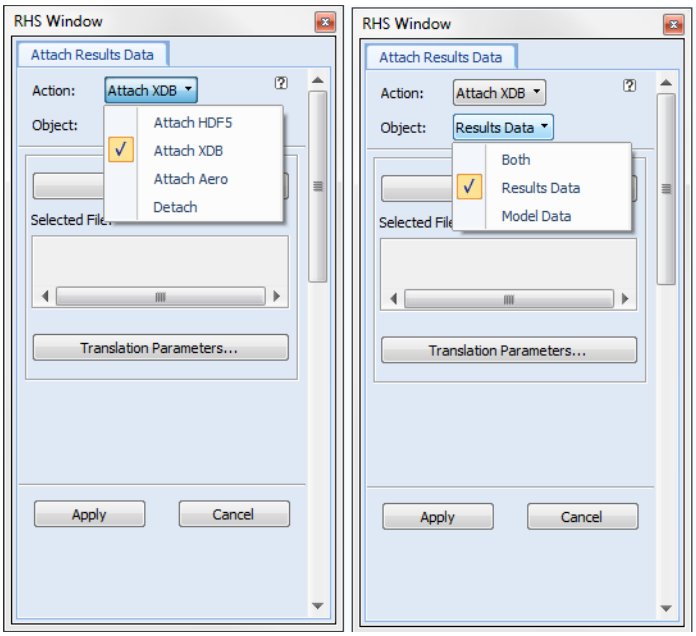
Loads Data, Model Data or both Loads and Model Data can be accessed from an external file. It is recommended that Model Data be accessed from an empty MSC.Patran database. Loads data refers to all results data, not merely the loads. Existing MSC.Patran results tools can be used for all results following the attachment in MSC.FlightLoads. Model data brings in a grid-element representation of both the structural and aerodynamic models. Neither are “complete” in the sense of direct creation, but are complete in regards to visual action of results. To bring “complete” models, use the Import function.

Accessing Aero data causes the automatic creation of an XDB file and subsequent attachment. This is transparent to the user.

If the Loads Data to be processed already resides in the MSC.Patran database as LBCs or results, it does not have to be accessed.

Overview

When Attach Results Data is selected, the default options that appear on the form are Action: **Attach XDB** and Object: **Results Data**. The following are all the different possible Actions and Objects.



Note: Not all actions have an Object Menu.

Attach Results Data

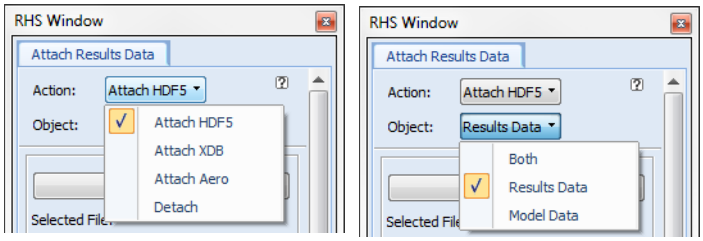
The following is a list of the Results Browser Forms and subforms related to Attach Results Data.

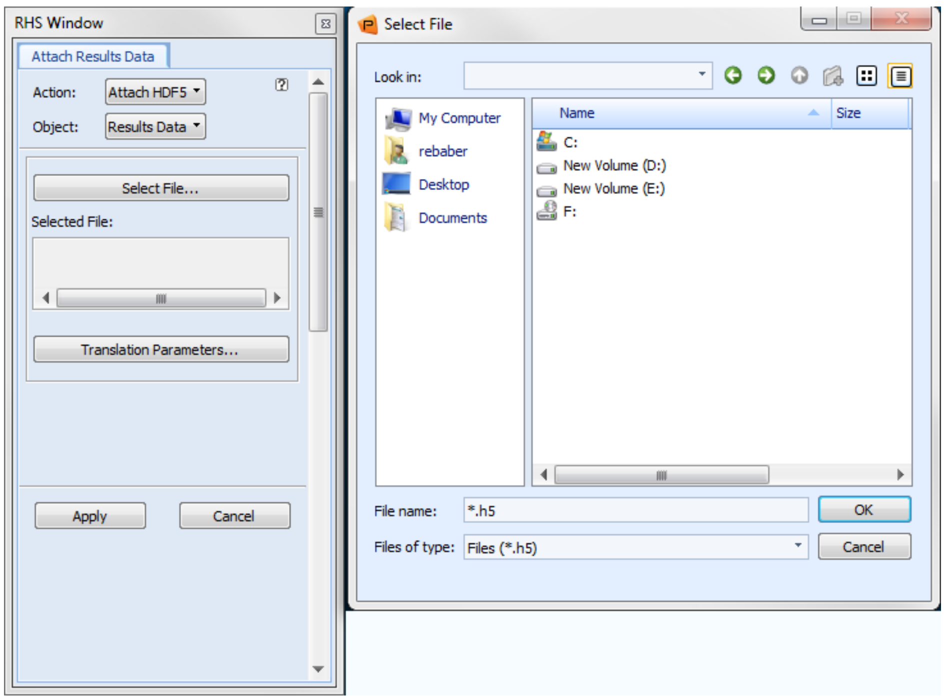
- Attach HDF5
  - Results Translation Parameters Subform
- Attach XDB
  - Results Translation Parameters Subform
- Attach Aero
- Detach

Attach HDF5

This form illustrates the selection of a HDF5 (h5) results file for attachment. The h5 file is created by including the MDLPRM,HDF5,1 entry in the bulk data section of the input deck.

Import of Model data is supported only for nodes and elements from h5 file thus it is recommended to import the corresponding input deck to get the complete model data before importing the h5 file for result entities.

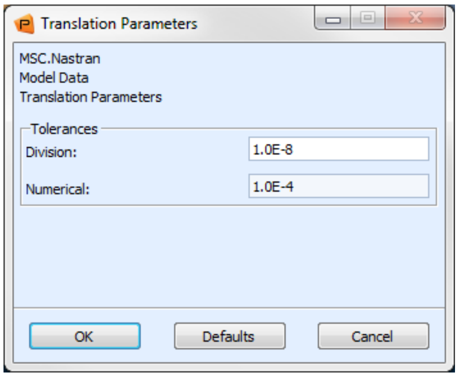




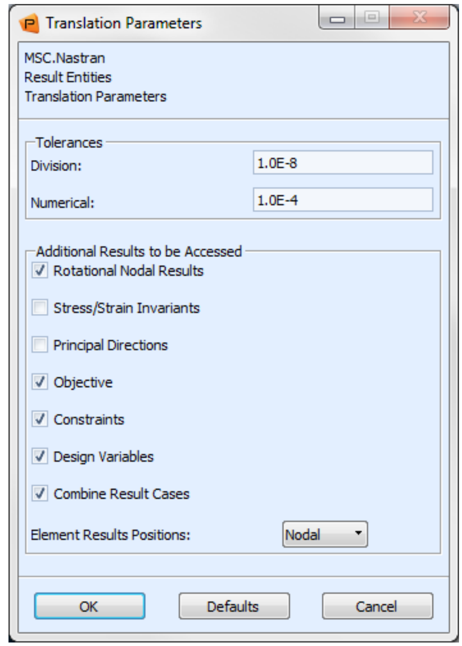
Item	Description
Method	The **Method** choices for accessing HDF5 files are: ***Result Entities***, ***Model Data***, or ***Both***. For HDF5 files, when selecting ***Model Data*** or ***Both***, only the actual nodes and elements of the model are imported into the Patran database
Select File...	This brings up a file browser to select the results file to access. By default the file name is the ***jobname.h5,*** Therefore it is only necessary to select a file if it differs from the jobname.
Translation Parameters...	Defines the parameters used to control the results or model translation. See Results Translation Parameters Subform, 214.

Results Translation Parameters Subform

This subordinate form appears when the **Translation Parameters** button is selected. This form affects import of objects as noted below:



Model Data



Result Entities and Both

Item	Description
Tolerances ■ Division ■ Numerical	Defines the tolerances used during translation. The division tolerance is used to prevent *division by zero* errors. The numerical tolerance is used when comparing real values for equality.
Additional Results to be Imported	Indicates which of the given results categories are to be filtered out during translation. Items selected will be translated. Items not selected will be skipped. By default, ***Rotational Nodal Results, Objective, Constraints, Design Variables,*** and ***Combine Result Cases*** get translated while ***Stress/Strain Invariants,*** and ***Stress/Strain Tensor Principal Directions*** results are ignored. ***Combine Result Cases*** combines all Superelement results into a single result case when ON, but places each Superelement in separate result cases when OFF.
Element Results Positions	If an element has results at both the centroid and at the nodes, this filter will indicate which results are to be included in the translation.

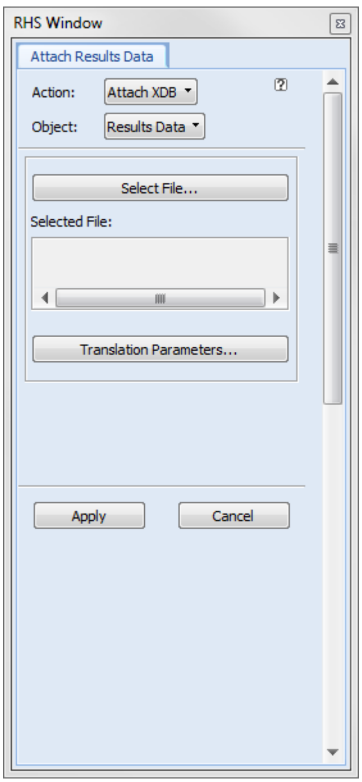
Supported HDF5 Result Quantities

The following are the result quantities supported through HDF5 in the Flightloads:

- Aeroelastic Forces, Elemental Component
- Aeroelastic Moments, Elemental Component

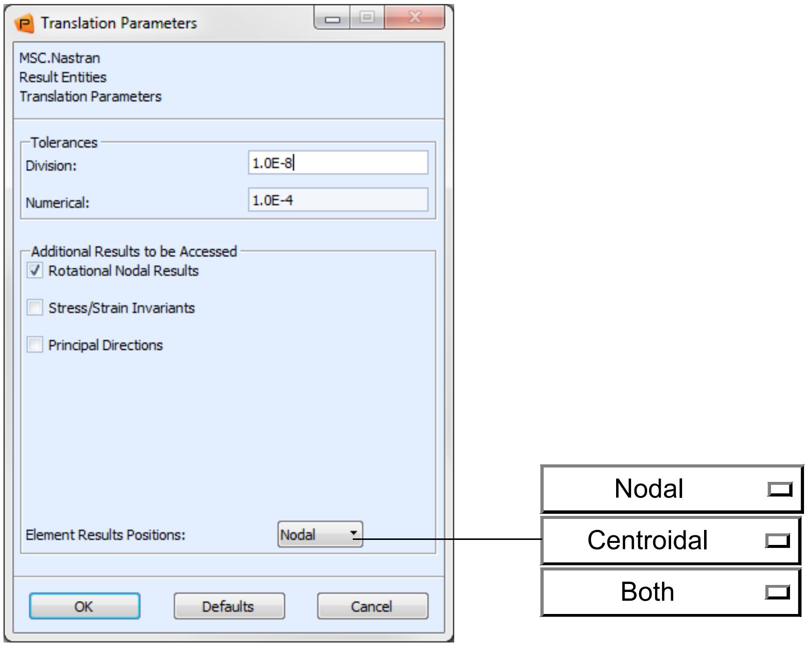
Attach XDB

This form illustrates the selection of an XDB results file for attachment. Typically, the default Translation Parameters are adequate and can be ignored. Note that the Element Results Position is by default set to Nodal. This should be set to Both for FlightLoads because aerodynamic forces are computed at the centroid of aerodynamic elements and the structural forces are computed at the nodes.



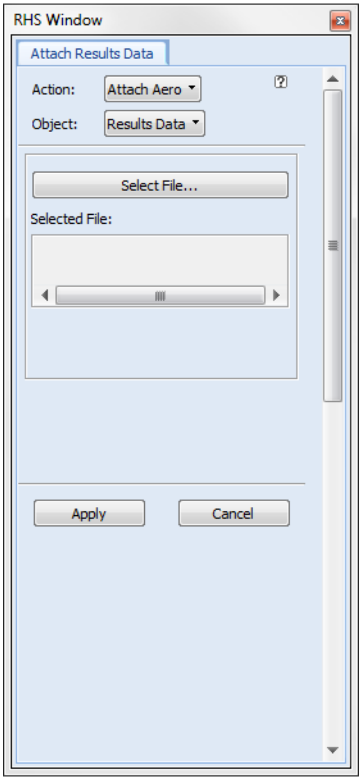
Parameter	Description
Translation Parameters	This button displays a slightly different form based on what the object is set to.

Results Translation Parameters Subform



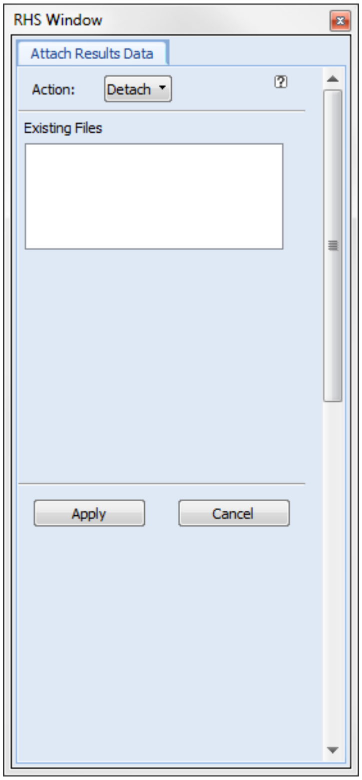
Attach Aero

This form illustrates the selection of an Aero results file for attachment. The XDB attachment in MSC.FlightLoads has the "Rotational Nodal Results" selected by default to ensure that nodal rotations are present for spline verification and that moments are present for load summations.



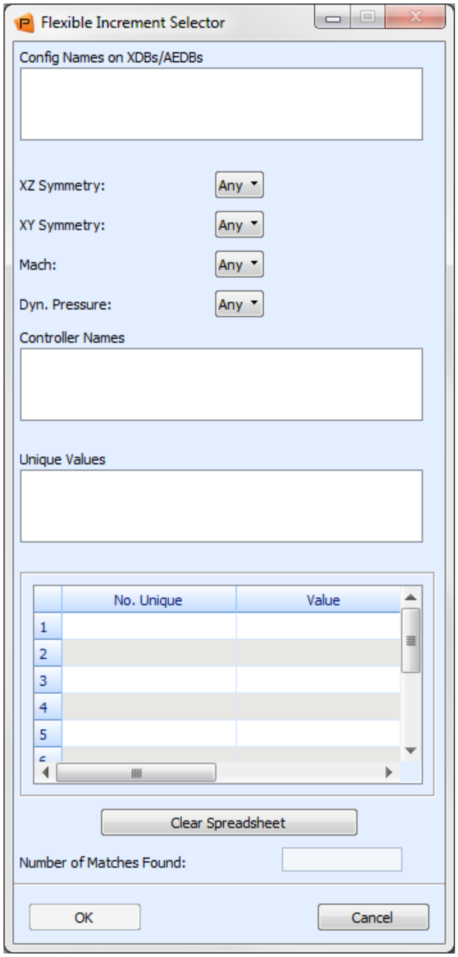
Detach

This form detaches the previously attached Loads Data files.



Flexible Increment Filter

This is a new option on the Results Browser menu that allows you to choose a single unit solution to be used instead of the general solution. This form is only available when the Result selector on the main form is set to Flexible Increment.



Parameter	Description
Config Names on XDBs and AEDBs	Select a value first or the remainder of the form will remain disabled.
Controller Names	All the possible values found in the selected Config Name and not filtered out by the other widgets are listed as options in the listbox.
Unique ANGLEA Values	The label for this listbox is “Unique XXX Values” where XXX is determined by the selected Controller Name.
Number of Matches Found	The user is not able to select OK until the number of matches found is down to 1.

Note:	
	1. All the possible values found in the selected Config Name and not filtered out by the other widgets are listed as options in the option menu. The default value is "Any". 2. After a value is set, it is used to filter the other values on this form. A value of "Any" implies that a choice has not yet been made for that value.

Loads Browsers

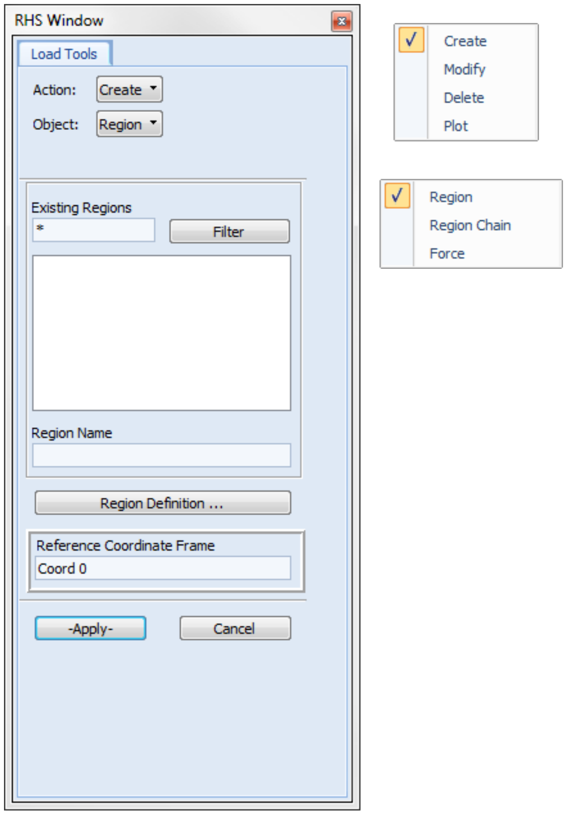
Visualizing resultant loads along the bending axis of a wing can be very helpful for verifying that the correct loads have been applied to a model. MSC.FlightLoads and Dynamics provides a set of XY Plotting capabilities that allow for plotting overall vehicle applied shear, bending moment, and torque (SBMT) diagrams based on the applied loading. This functionality allows for plotting overall vehicle applied SBMT diagrams along different axes for different regions of the structures.

Load Summation provides a convenient method of summing the applied vehicle loads about a spatial location.

The Loads Browser in the Results Browser module allow users the ability to create either Running Loads Plots or perform Load Summations

Overview

When Loads Browser is selected, the first form displayed is Create/Region. Shown adjacent to the form below are all the different Actions, Objects, and Method Options available.



Not all combinations of Action and Object are valid for this form. The Method option menu is only available when the Action is set to Plot or if the Action is set to Create and the Object is set to Force.

Most of the Loads Browser forms are shown and annotated in the following pages, grouped by Action as follows:

- Create (Region, Region Chain, and Force)
- Modify (Region and Region Chain)
- Delete (Region and Region Chain)
- Plot - Running Loads (LBCs, Load Cases, and Results)
- Plot - Load Summation (LBCs, Load Cases, and Results)

Loads Browser

Following is a list of the Results Browser forms and subforms related to the Loads Browser:

- Create/Region
- Create/Region Chain

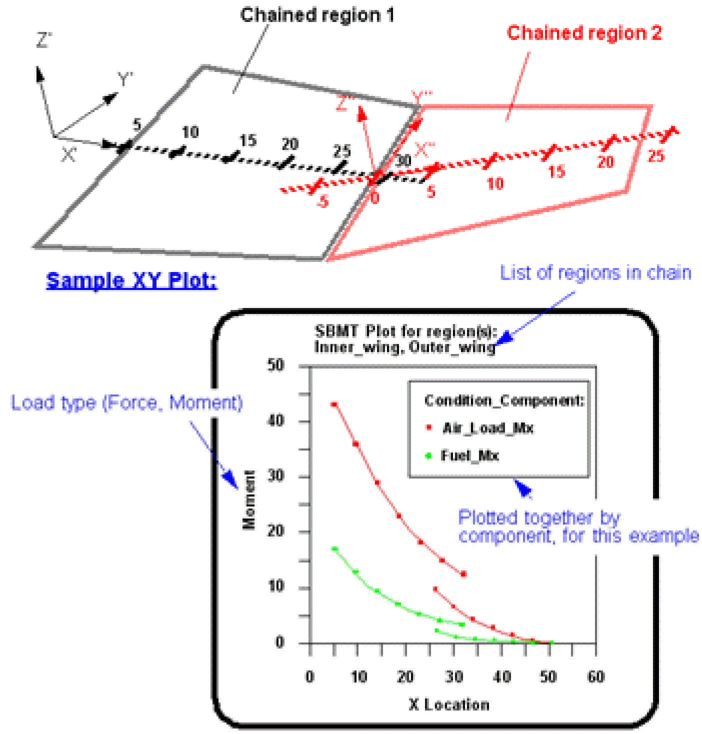
- Create/Force
- Modify/Region
- Modify/Region Chain
- Delete/Region
- Delete/Region Chain
- Plot/LBCs/Running Loads
- Plot/Load Cases/Running Loads
- Plot/Results/Running Loads
- Plot -- Running Load Subforms
- Plot/LBCs/Load Summation
- Plot--Load Summation Subforms

Creation of Running Loads plots and Load Summations requires the identification of a section of the model over which the plot is to be created. This section can be described using a Region, Region Chain or on-the-fly through interactive node and element selection.

Visualizing resultant loads along the bending axis of a wing can be very helpful for verifying that the correct loads have been applied to a model. MSC.FlightLoads and Dynamics provides a set of XY Plotting capabilities that allow for plotting overall vehicle applied shear, bending moment, and torque (SBMT) diagrams based on the applied loading. This functionality allows for plotting overall vehicle applied SBMT diagrams along different axes for different regions of the structures.

Load Summation provides a convenient method of summing the applied vehicle loads about a spatial location.

The following picture illustrates the methodology behind the loads data for plots.



Load Summations can be performed using previously defined Regions or on-the-fly selection of nodes and elements.

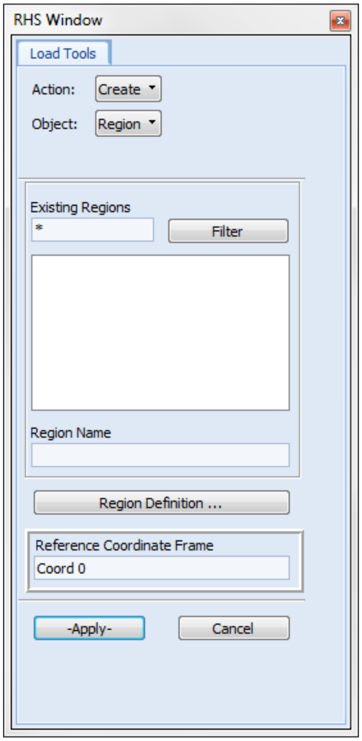
Both Running Loads and Load Summation support the following kinds of external loads:

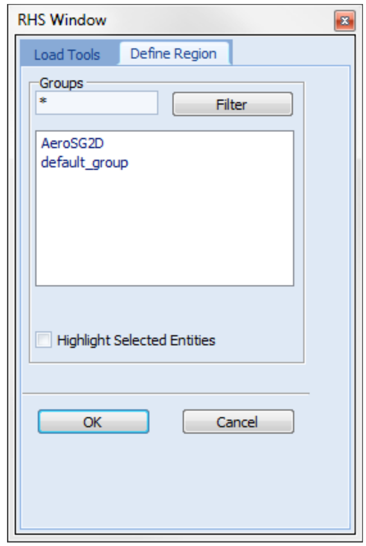
- Forces and Moments
- Pressure
- Inertia
- Distributed Edge Loads

Typically, Force and Moment loads are evaluated because they appear in the Aerodynamic/Aeroelastic database and Aeroelastic analysis results files.

Create/Region

This form describes the creation of model Regions. These can describe either aerodynamic or structural Regions. A Region should only pertain to either the structural or aerodynamic model and not both.

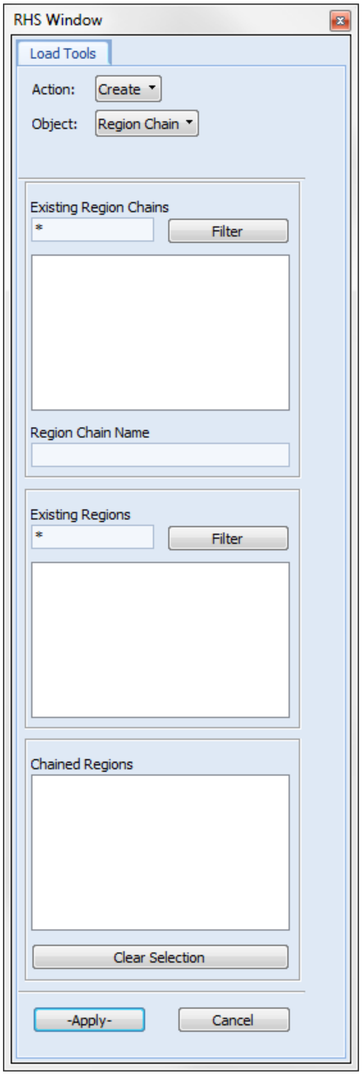




Parameter	Description
Region Name	Used to enter a new Region name.
Reference Coordinate Frame	Defines the Region's Reference Coordinate Frame.
Define Region	Used to select the Group(s) to be included in the Region definition.
Highlight selected entities	If this toggle is set, then all of the entities associated with groups that define the Region will be highlighted in the graphics viewport.

Create/Region Chain

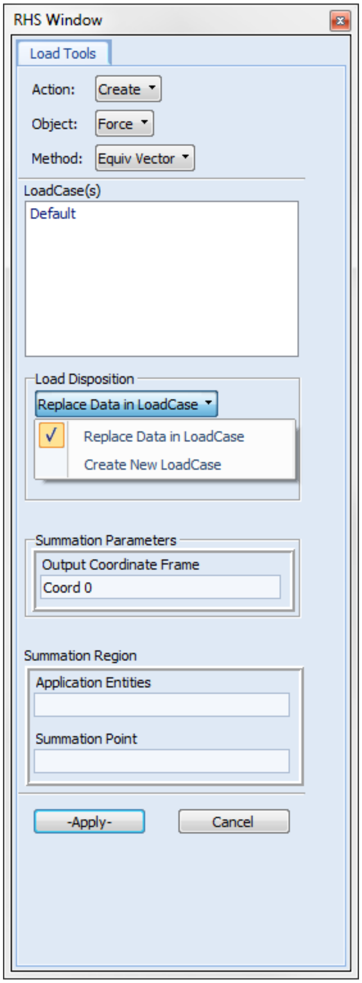
Two or more Regions can be chained together and referenced as a Region Chain. A Region Chain can describe Regions with a discontinuity at their intersections (i.e., two Regions that follow a wing spar break). Region Chains can be referenced in Running Load plots. The following form is used to combine individual regions into a region chain that can be plotted in a single graph.


RHS Window
Load Tools
Action:
Create
Object:
Region Chain
Existing Region Chains
*
Filter
Region Chain Name
Existing Regions
*
Filter
Chained Regions
Clear Selection
-Apply-
Cancel

Parameter	Description
Existing Region Chains	Previously created Region Chains.
Existing Regions	Current Regions in the model.
ChainedRegions	Defines the Regions in the Chain and the order in which they are to be plotted. Loads from these regions are calculated from the maximum X of the last chained Region back to the minimum X of the first chained Region. Loads from Regions further out on the Chain are added to Regions further in on the Chain at the maximum X after they are transformed to the new coordinate system.
Clear selection	Clears all the items in the Chained Region(s).

Create/Force

The Create Force functionality enables a user to create an equivalent force vector that is derived from all of the nodal forces within any combination of existing load cases. The user has the ability to define the load summation point where the equivalent force is applied as well as the region over which the load summation is performed. The resultant force can either be used to replace the contents of an existing load case or to create an entirely new load case.



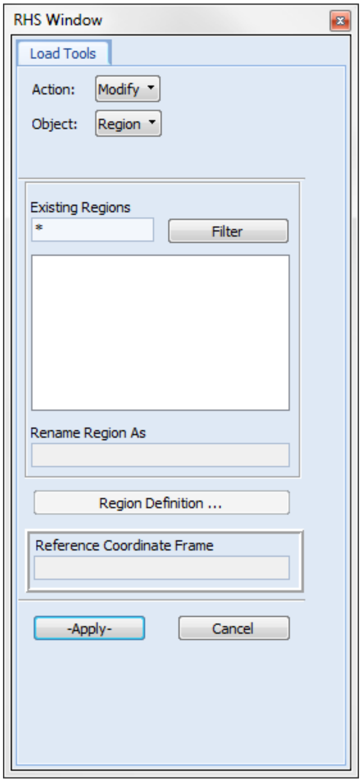
Parameter	Description
LoadCase(s)	Available Load Cases in the database currently open. Load Cases selected by the user for load summation are highlighted.
Load Disposition	User has the option to either create a new load case or replace the data in an existing load case. The form that is displayed for the later option is shown on the next page.

Parameter	Description
New LoadCase Name	User specifies the name used for any new load cases.
Summation Parameters	User specifies the coordinate frame that will be used to define the components of the equivalent force vector. The default is the global rectangular.
Application Entities	User specifies those entities over which the load summation is performed. The user must ensure that the load cases selected have nodal loads that are applied to the selected entities else an equivalent force will not be created. Consequently, element base forces such as pressures are ignored.
Summation Point	User specifies the node where the equivalent force will be applied. Only a single node may be specified. Any additional nodes will be ignored.

The following form is displayed if the user selects the load disposition switch to enable the Replace Data in Load Case option. This option will cause all of the loads contained in the selected load case to be replaced with the newly created equivalent force.

Modify/Region

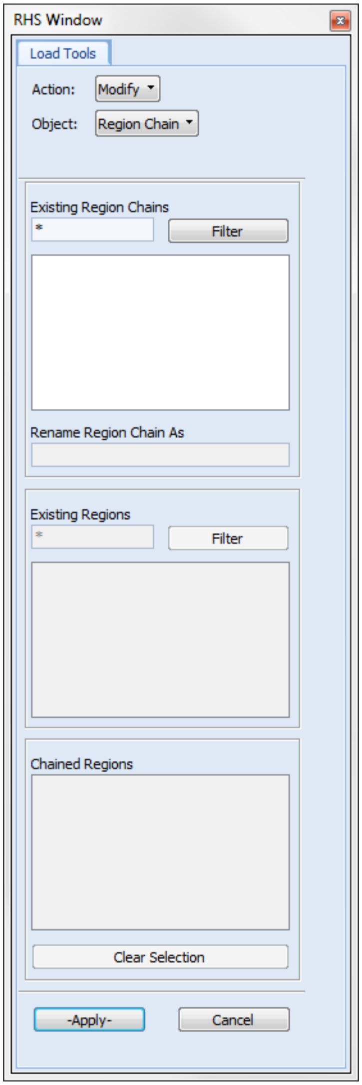
Previously created Regions can be modified.



Parameter	Description
Region Name	Used to enter a new Region name, or to modify the name of the selected Region.
Region Definition	Used to enter a new Region name, or to modify the name of the selected Region.
Define Region	Used to select the Groups to be included in the Region definition.
Reference Coordinate Frame	Defines the Reference Coordinate Frame of the Region.

Modify/Region Chain

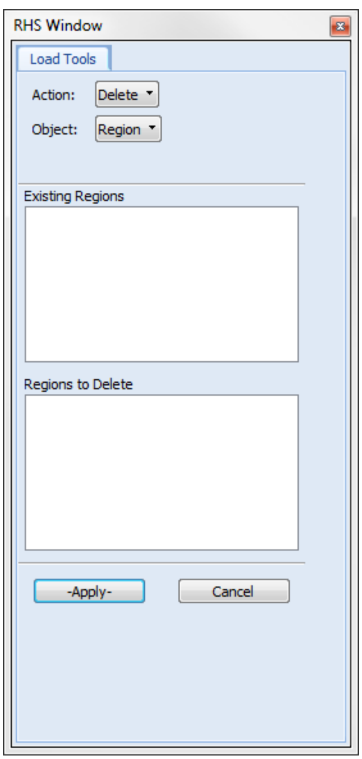
Previously created Region Chains can be modified.


RHS Window
Load Tools
Action:
Modify
Object:
Region Chain
Existing Region Chains
*
Filter
Rename Region Chain As
Existing Regions
*
Filter
Chained Regions
Clear Selection
-Apply-
Cancel

Parameter	Description
Existing Region Chains	Previously created Region Chains are listed here. The Region Chain selected for modification is highlighted.
Rename Region Chain As	The user has the option to change the name of the region chain being modified.
Existing Regions	Current Regions in the model. User selects those regions that will be contained in the revised region chain definition.
Chained Regions	Defines the Regions in the Chain and the order in which they are to be plotted. Loads from these regions are calculated from the maximum X of the last chained Region back to the minimum X of the first chained Region. Loads from Regions further out on the Chain are added to Regions further in on the Chain at the maximum X after they are transformed to the new coordinate system.
Clear selection	Clears all the items in the Chained Region(s).

Delete/Region

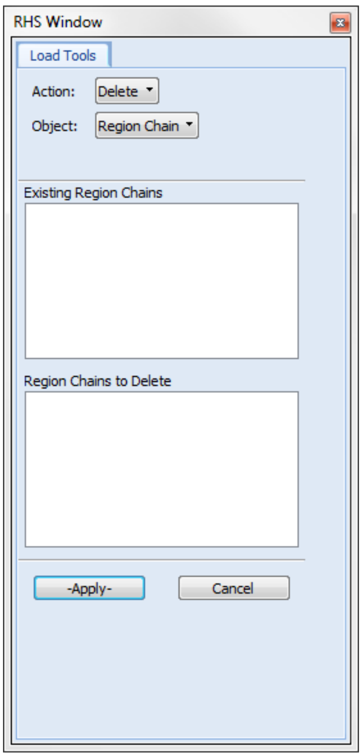
Previously created Regions can be deleted from the database if they are no longer useful.



Parameter	Description
Existing Regions	Select Region and Apply to delete region.

Delete/Region Chain

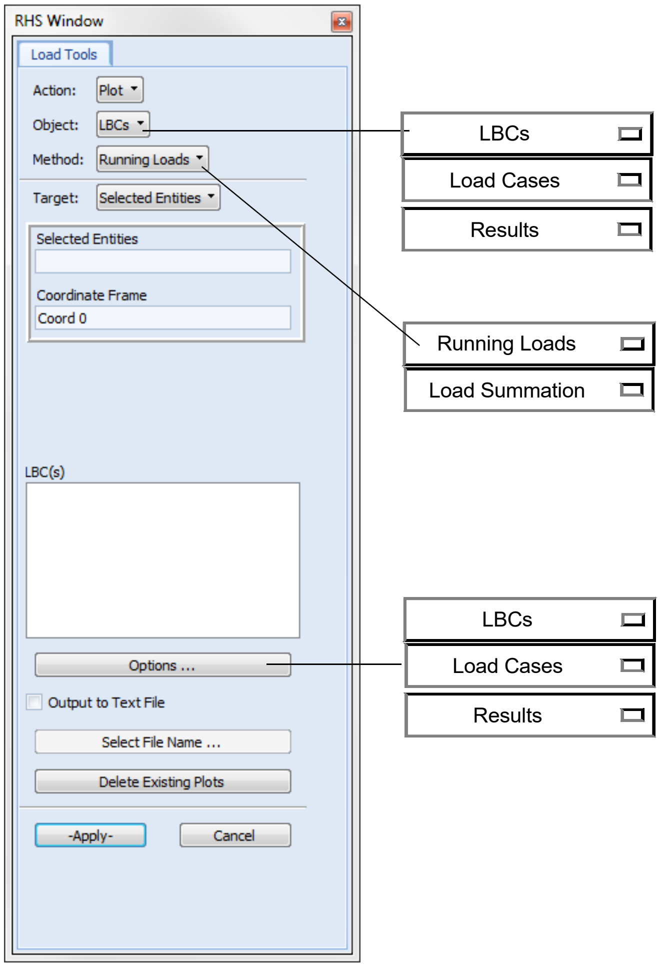
Previously created Region Chains can be removed from the database when they are no longer useful.



Parameter	Description
Existing Region Chains	Select Region Chain and Apply to delete chain.

Plot

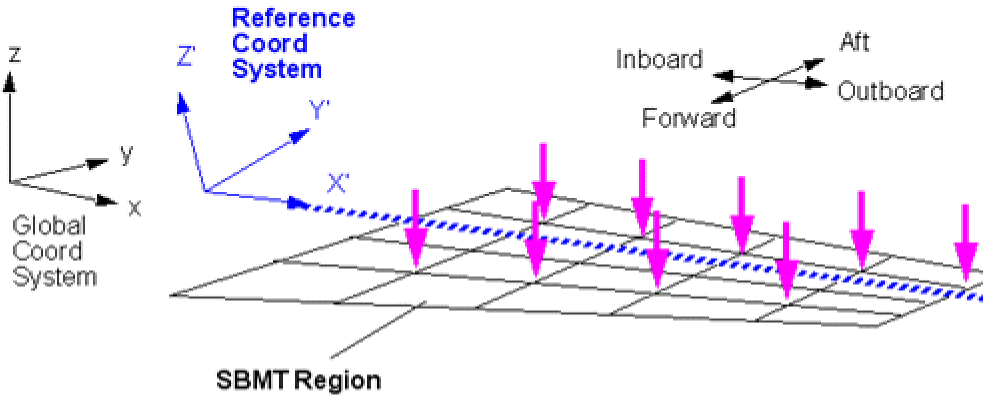
When the Action is set to Plot, the user has the ability to plot various kinds of Running Loads or Load Summation. The possible options for the form at this point are:



Plot Running Loads

To Plot Running Loads, set the Action to Plot on the Loads Browser form. The form layout is set up so that the user sets up the plot in a logical order from the top of the form to the bottom. Several sub-forms are also used to define the parameters of the plots to be generated. Users have the optional choice of writing this information to a report file.

After identifying that a Running Load plot is desired, the model's nodes and elements must be identified either by membership to a Region, Region Chain or Selected on-the-fly.

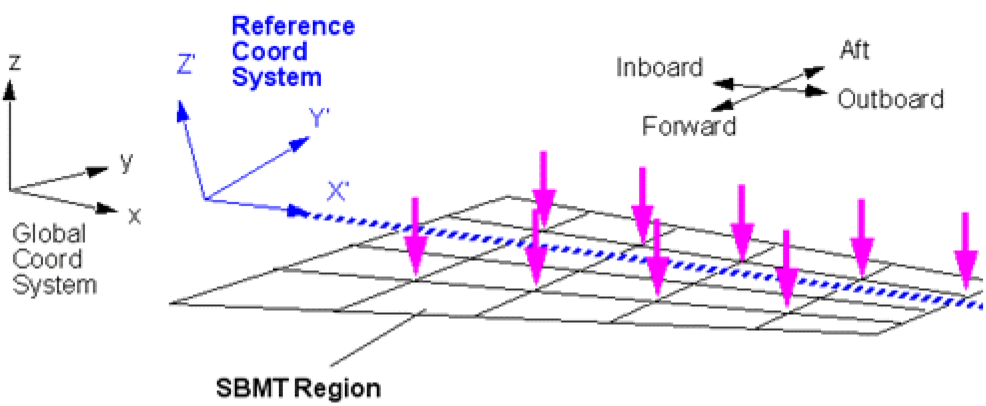


Running Loads - Methodology

This section covers the theory and methodology behind the creation of Running Load plots and also shows the overall mathematical procedure that is used to generate the data for these plots.

Regions

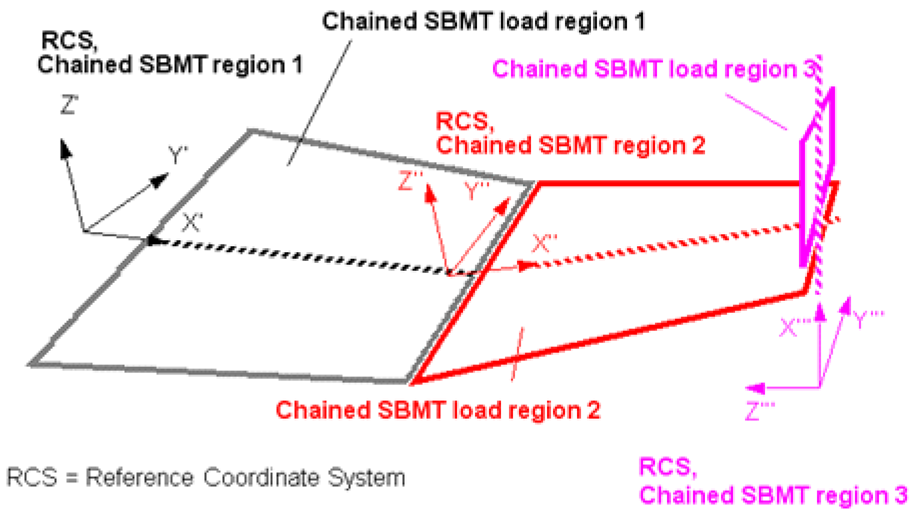
The first item to be defined is the Region. The Region consists of finite elements and a reference coordinate system (see figure below).



All data reported on a Running Load plot for this Region is given in terms of the reference coordinate system. The Region covers a certain range of ordinates within the reference coordinate system. This range is from the minimum X-axis value in the reference coordinate system of the FEM nodes in the region to the maximum X-axis value in the reference coordinate system of the FEM nodes in the Region. Correspondingly, there is a Y and Z range. X is important because summation occurs along X.

Region Chains

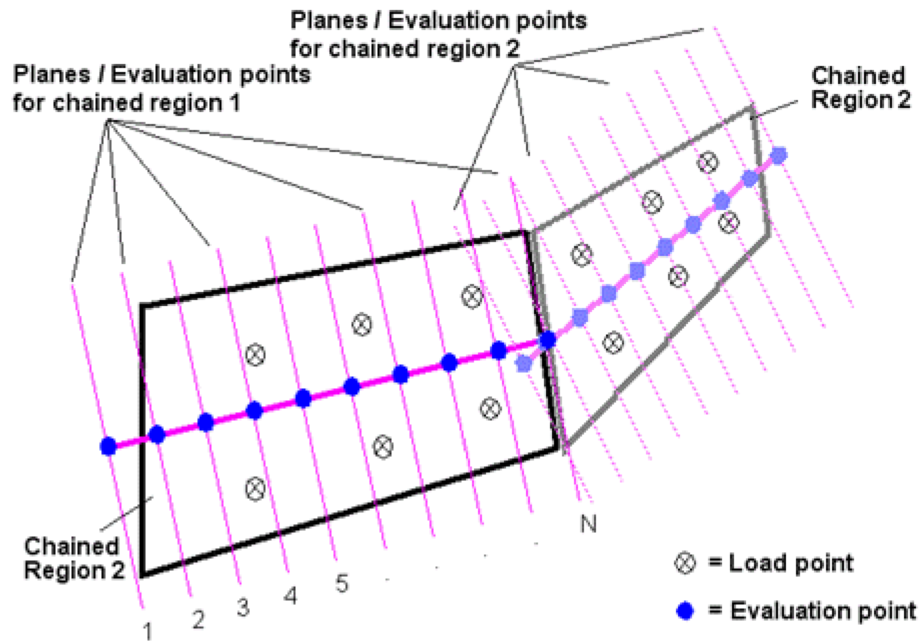
A Region Chain may be defined. This Chain consists of an ordered list of Regions. These Regions should be adjacent and attached pieces of structure for an Running Load plot to give realistic results (see figure below). There are no checks in MSC.FlightLoads and Dynamics that prevents Region Chains from containing unattached pieces of structure.



Running Load Plots

The procedure for generating the Running Load plot data consists of accumulating the sum of the loads from the maximum X-axis location in the last Region in the Chain back to the minimum X-axis value of the first Region in the Chain. These accumulating summed loads are known as Running Loads. Each point on the resulting curve is equivalent to a load summation done at the X ordinate for all parts “outboard” (larger X value) in the region or region chain.

Running Loads are calculated at “evaluation points” for each Region. The number of points at which applied loads are summed and reported is determined by the evaluation points box on the plot form. This number is the same for all of the regions. Loads applied to the model that lie within the X-axis region of the specified coordinate system(s) are included in the Running Load plot (see figure below).



The equations used to sum the loads together for an evaluation point are as follows.

$$FX_{eval} = \sum_{i=1}^{loads} FX_i$$

$$FY_{eval} = \sum_{i=1}^{loads} FY_i$$

$$FZ_{eval} = \sum_{i=1}^{loads} FZ_i$$

$$MX_{eval} = \sum_{i=1}^{loads} MX_i + FY_i z_i + FZ_i y_i$$

$$MY_{eval} = \sum_{i=1}^{loads} MY_i + FX_i z_i + FZ_i (x_i - x_{eval})$$

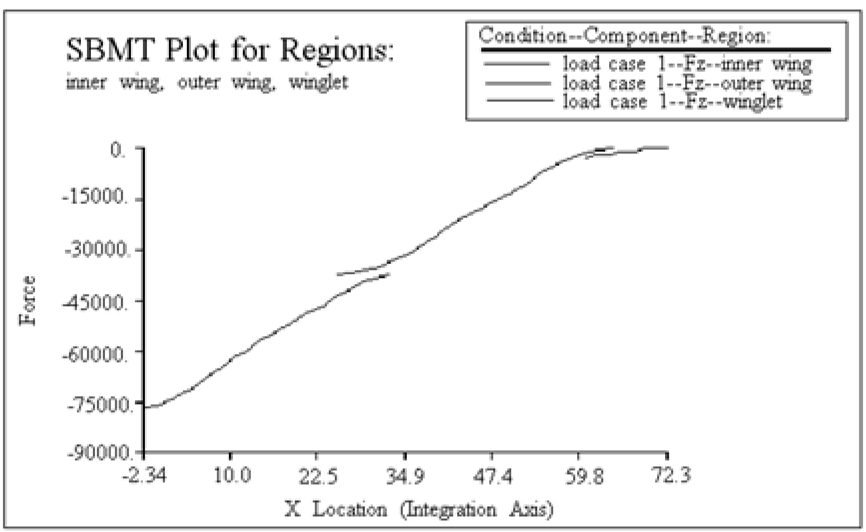
$$MZ_{eval} = \sum_{i=1}^{loads} MZ_i + FX_i y_i + FY_i (x_i - x_{eval})$$

To find the beginning Running Load (load at the maximum evaluation point) for a Region, the Running Load at the minimum evaluation point of the next Region in the Chain is transformed from the reference coordinate system of the next Region in the Chain to the reference coordinate system of the current Region and then translated to the maximum evaluation point of the current Region. If the Region is the last Region in a Chain then the beginning Running Load is zero for all terms.

For the reporting Region data on Running Load plots, the first Region is plotted using its X-coordinates directly. The next Chained Regions are included by adding their X-value onto the end of this, with the origin of the second system located its value in the first coordinate system. For example, suppose a first Chained Region is defined from X'=5 to X'=32, and the second Chained Region covers X"=-3 to X"=23. The origin of the second Chained Region located at X'=29 in the first coordinate system. Data from the first Chained Region occupies the graph from x=5 to x=32, and the second Chained Region is plotted over x=26 to x=51.

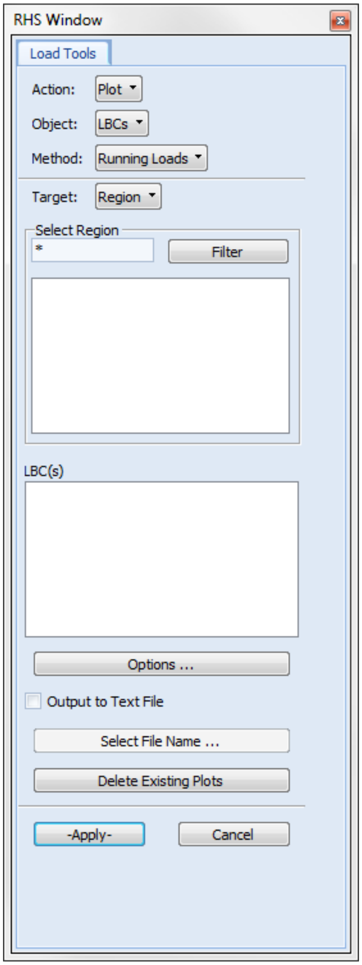
It is very likely that two adjacent Regions will contain identical nodes. Therefore, the code checks to see if regions next to each other in a Region Chain contain identical nodes and place forces attached to only those nodes in the Region appearing first in the list. This will prevent double counting of the forces applied to these nodes. This check is only done for adjacent Regions in a Region Chain. No check is done for duplicate elements, therefore pressures applied to elements present in more than one Region in an Region Chain will be summed twice in the Running Load calculation.

A sample Running Load plot from is shown below:



Plot/LBCs/Running Loads

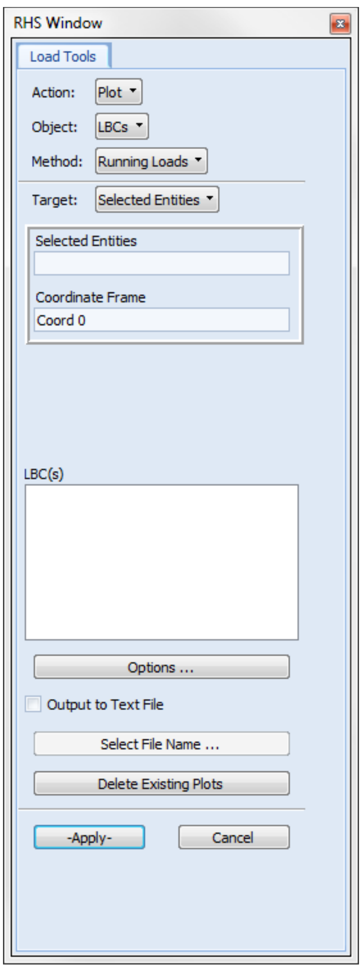
One or more MSC.Patran Loads/Boundary Conditions can be integrated over a model to produce a Running Load plot.



Parameter	Description
Target	Option for Region and Region Chain are very similar.
Select Region	Label changes to reglect whether the option is set to Region or Region Chain. **Note:** All available regions or region chains are listed. User selections are highlighted.

Parameter	Description
LBC(s)	List of available LBCs.
Options...	Refer to the Plot Options form in this section for more information on this selection.
Select File Name	Select File Name is only active when Output to Text File switch is enabled. This displays the standard File Name Definition form.
Delete existing plots	All running loads plots and any corresponding XYPLOT windows are deleted from the database.

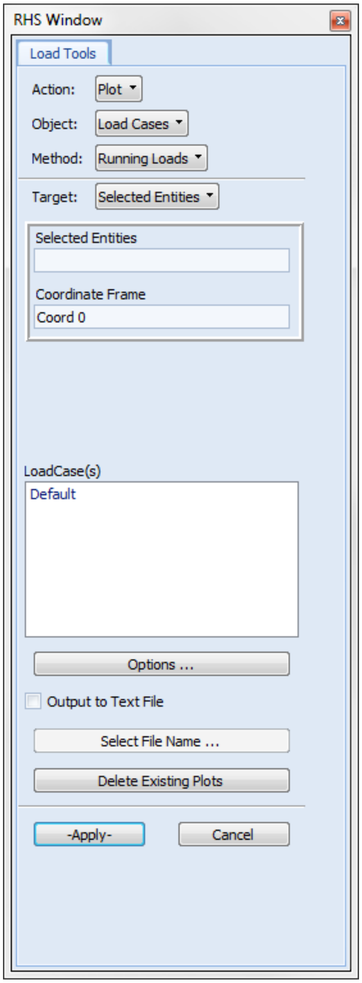
This form shows the changes that occur when the "Option" option menu is changed to "Selected Entities". Notice that the changes are valid for all Objects (LBCs, Load Cases, Results).



Parameter	Description
Selected Entities	Nodes or specific element types may be specified.
Coordinate Frame	The coordinate frame is specified for determining how force and moment components are defined.
LBC(s)	All existing LBCs contained in the model are listed. Any combination of LBCs may be specified. User selections are highlighted.
Options...	Refer to the Plot Options form in this section for more information on this selection.
Delete existing plots	All running loads plots and any corresponding XYPLOT windows are deleted from the database.

Plot/Load Cases/Running Loads

One or more MSC.Patran Load Cases, which may reference multiple LBCs, can be integrated over a model to produce a Running Load plot.

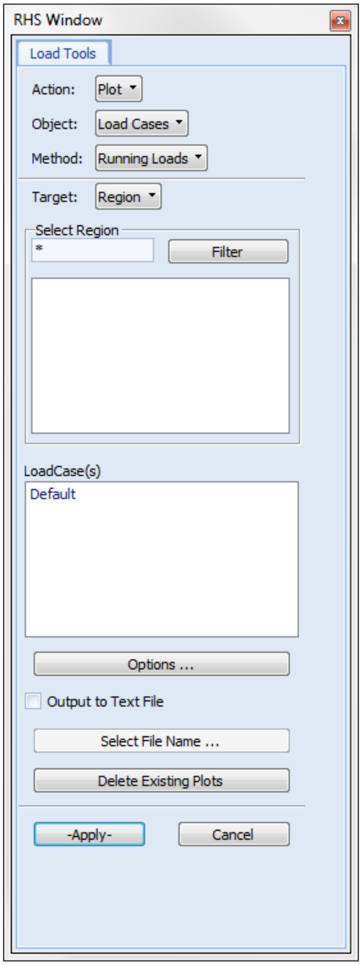


Parameter	Description
Select Region	This portion only appears if the Region option is selected. multiple regions may be selected and these will be highlighted.

Parameter	Description
LoadCase(s)	All available Load Cases are listed here. Multiple Load Cases may be selected and these will be highlighted.
Options...	Refer to the Plot Options form in this section for more information on this selection.
Select File Name	Select File Name is only active when Output to Text File switch is enabled. This switch will then display the standard File Name Definition Form.
Delete existing plots	All running loads plots and any corresponding XYPLOT windows are deleted from the database.

Plot/Results/Running Loads

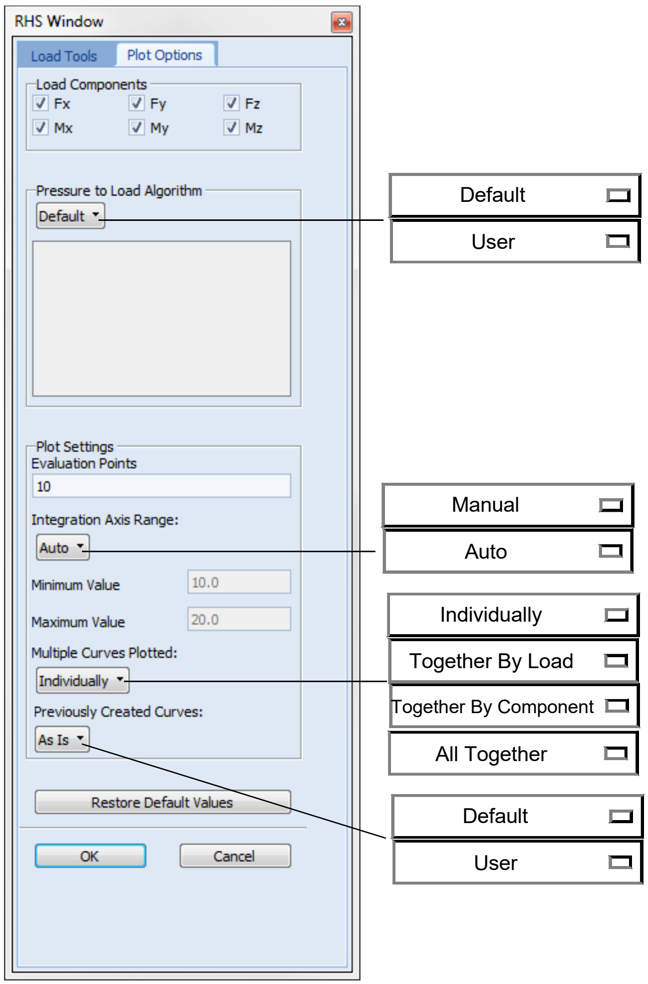
One or more Results can be integrated over a model to produce a Running Load plot. For MSC.FlightLoads and Dynamics, an XDB results file is automatically created by attachment to an Aerodynamic/Aeroelastic database. Therefore, Running Loads plots of this loads data require "Results" selection.


RHS Window
Load Tools
Action:
Plot
Object:
Load Cases
Method:
Running Loads
Target:
Region
Select Region
*
Filter
LoadCase(s)
Default
Options ...
Output to Text File
Select File Name ...
Delete Existing Plots
-Apply-
Cancel

Parameter	Description
Select Region	All existing regions are listed. User selections are highlighted.
Result Case(s)	All available Results Cases are listed here. Multiple Results Cases may be selected and these will be highlighted.
Nodal Vector Results	Only the Nodal Vector results contained in the selected Results Cases are displayed. If a Result Case contains no nodal results, then all running loads plots will be displayed as a value of zero that runs the full length of the regions. The same behavior will be exhibited for selected entities and region chains.
Options...	Refer to the Plot Options form in this section for more information on this selection.
Select File Name	Select File Name is only active when Output to Text File switch is enabled. This switch will then display the standard File Name Definition Form.
Delete existing plots	All running loads plots and any corresponding XYPLOT windows are deleted from the database.

Plot Running Loads - Plot Options

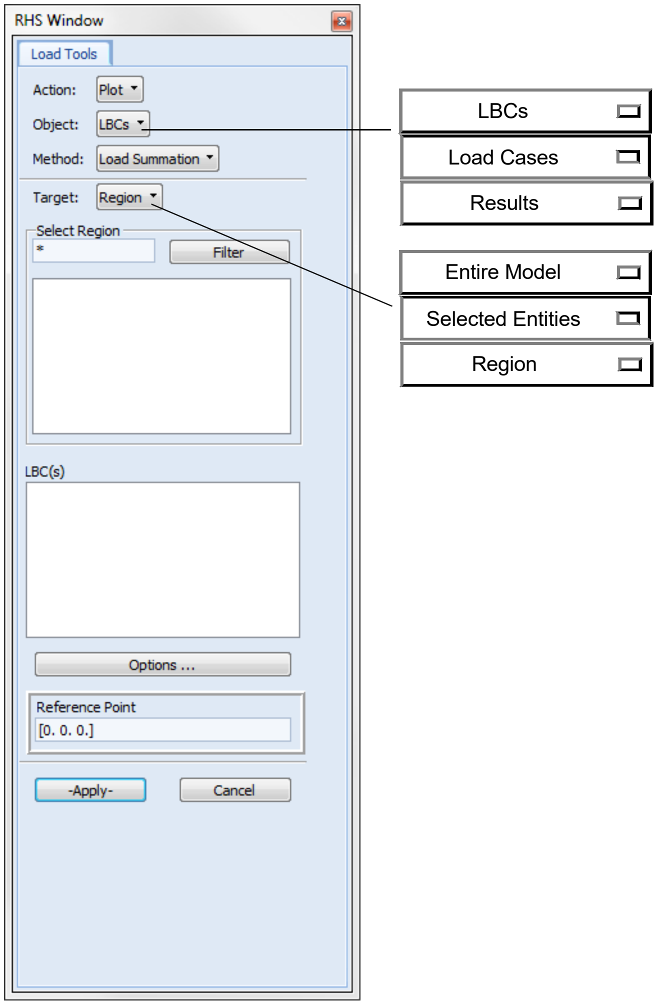
This subform is valid for all Objects (LBCs, Load Cases, and Results) and is accessed from the Running Loads main form. It controls the appearance of the XY plot, as well as the status of previously created plots.



Parameter	Description
Load Components	These are the components that are reported in the Running Load plot. By default, these are all ON.
Evaluation Points	Number of evaluation points for each Region. The default is 10 points.
Integration Axis Range	Minimum Value and Maximum Value are active when option "Integration Axis Range" is Manual; otherwise, these are greyed out for Auto.
Multiple Curves Plotted	Multiple curves can be Individually plotted to separate XY windows, plotted Together by Load/Set, plotted Together by Component or all plotted on one XY plot.
Previously Created Curves	Previously created Running Load plots can be left "As Is" or automatically deleted (Remove All).

Plot Load Summation

Load Summations are accessed by setting the Plot Method to Load Summation. These may not be performed over Region Chains. They can only be performed on Regions or selected on-the-fly.



Load Summation - Methodology

For forces, the Load Summation calculation is simply the summation of the force vector components (transformed if necessary). The moments are calculated by determining the cross product of the force vector and the moment arm vector where the moment arm vector is a vector from the reference point to the node on which the force vector is applied.

$$\begin{bmatrix} M_x \\ M_y \\ M_z \end{bmatrix} = \begin{bmatrix} F_x & F_y & F_z \end{bmatrix} \times \begin{bmatrix} d_x \\ d_y \\ d_z \end{bmatrix} \tag{6-1}$$

For pressure loads, the equivalent nodal loads must be calculated. The first step is to transform the element face to a local coordinate system whose normal, the vector $\bar{w}$, is defined by the cross product of a vector from node1 to node 2, the vector $\bar{u}$, of the face and a vector from node 1 to node 4,

$$\begin{bmatrix} u_x \\ u_y \\ u_z \end{bmatrix} = \begin{bmatrix} v^{1\text{-}2}{}_x \\ v^{1\text{-}2}{}_y \\ v^{1\text{-}2}{}_z \end{bmatrix} \tag{6-2}$$

and

$$\begin{bmatrix} w_x \\ w_y \\ w_z \end{bmatrix} = \begin{bmatrix} v^{1\text{-}4}{}_x & v^{1\text{-}4}{}_y & v^{1\text{-}4}{}_z \end{bmatrix} \times \begin{bmatrix} u_x \\ u_y \\ u_z \end{bmatrix} . \tag{6-3}$$

The second local direction, the vector $\bar{v}$, is defined by taking the cross product of $\bar{w}$ and $\bar{u}$,

$$\begin{bmatrix} v_x \\ v_y \\ v_z \end{bmatrix} = \begin{bmatrix} w_x & w_y & w_z \end{bmatrix} \times \begin{bmatrix} u_x \\ u_y \\ u_z \end{bmatrix} . \tag{6-4}$$

The transformation matrix, $\bar{\lambda}$, is created from the direction cosines between the local system and the global system. The nodal coordinates are then multiplied by the transformation matrix.

The pressure load, like the displacements in an isoparametric finite element, is defined anywhere in the element (or on the element face for 3D elements) by

$$q(u, v) = \sum_{i=1}^{n} q_i \Psi_i \tag{6-5}$$

$q(u, v)$	=	=the spatial distribution of the pressure load in the local element coordinate system.
q_i	=	=the pressures at the nodes
Ψ_i	=	=the element interpolation functions.

The equivalent nodal forces are calculated from the exact integral evaluated using Gauss-Legendre quadrature

$$\bar{q} = \int_{\Omega_R} q(\xi, \eta) d\xi d\eta \cong \sum_{I=1}^{M} \sum_{J=1}^{N} q(\xi_I, \eta_J) \det(J) W_I W_J \tag{6-6}$$

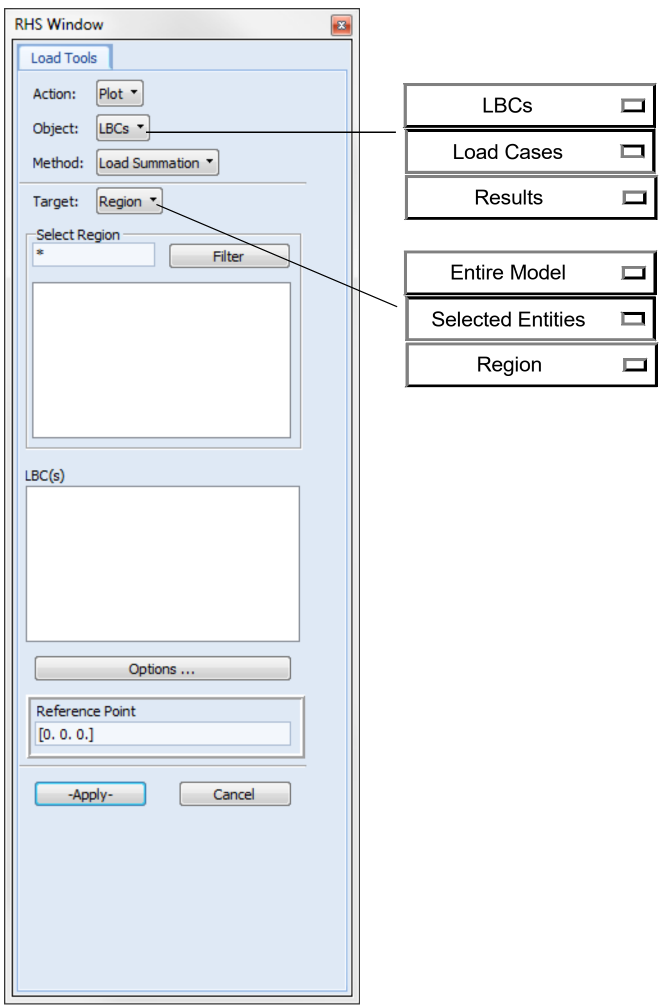
ξ, η	=	=the element's parametric coordinates.
$q(\xi, \eta)$	=	=the pressure distribution in the element's parametric coordinate system.
$\det(J)$	=	=the determinate of the Jacobian.

$q(\xi_I, \eta_J)$	=	=the pressure at the integration points ξ_I, η_J .
W_I, W_J	=	=the Gauss-Legendre integration weights.

Once calculated, the nodal loads are transformed into the global system using the transpose of the transformation matrix.

Plot/LBCs/Load Summation

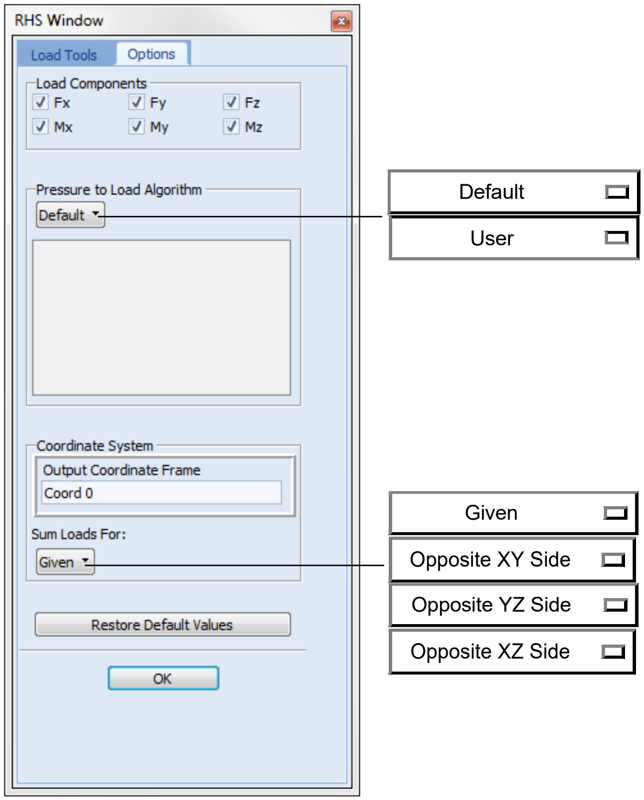
This form is the same for all Objects (LBCs, Load Cases, and Results). The only change is the label on the "Select LBC(s)..." button and the subform it displays. See the previous pages where Method=Running Loads for button labels and subforms.



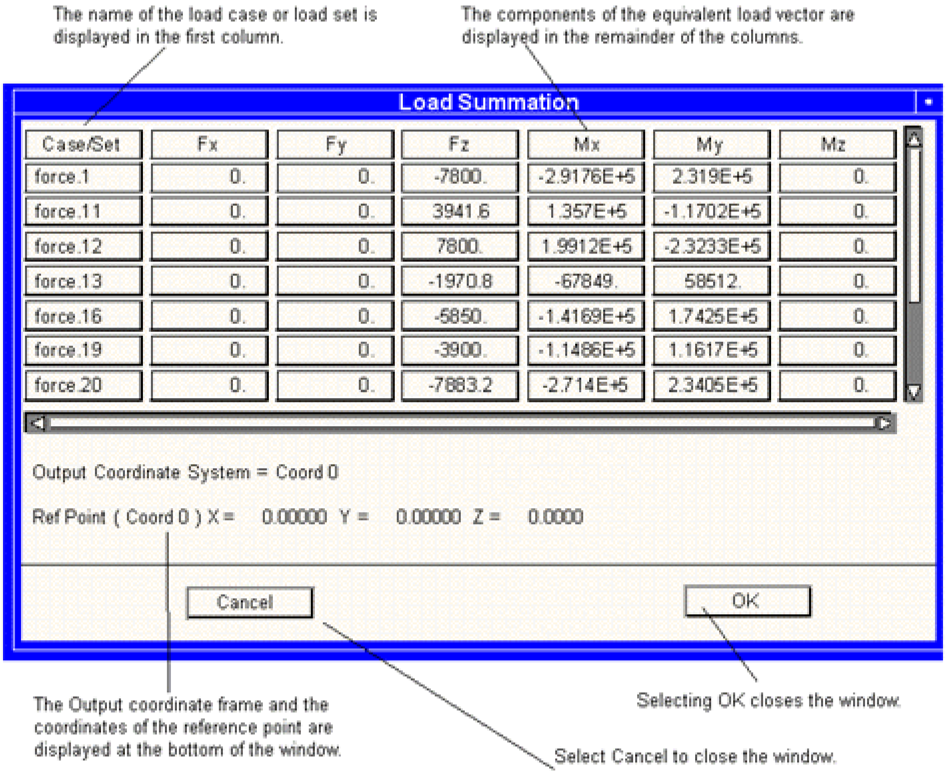
Parameter	Description
Select Region	This widget is available on all the Load Summation forms whenever the Option is set to Region.
LBCs	This listbox is replaced with the "Load Case(s)" listbox when the Object is set to LoadCases. When the Object is set to Results, this listbox is replaced with two listboxes, one called "Result Cases(s)" and one called "Nodal Vector Results". Refer to the various running loads forms for information on what these listboxes are used for.
Reference Point	Reference Point that defines the location that load summation will be performed relative to. May be either a physical location or a node.
Selected Entities	This widget is available on all Load Summation forms whenever the option is set to Selected Entities.

Plot Load Summation - Plot Options

This form is valid for all Objects (LBC, Load Cases, and Results) and is accessed from the Load Summation main form.



Select the Apply button on any of the Plot/Load Summation main forms to display the following spreadsheet.



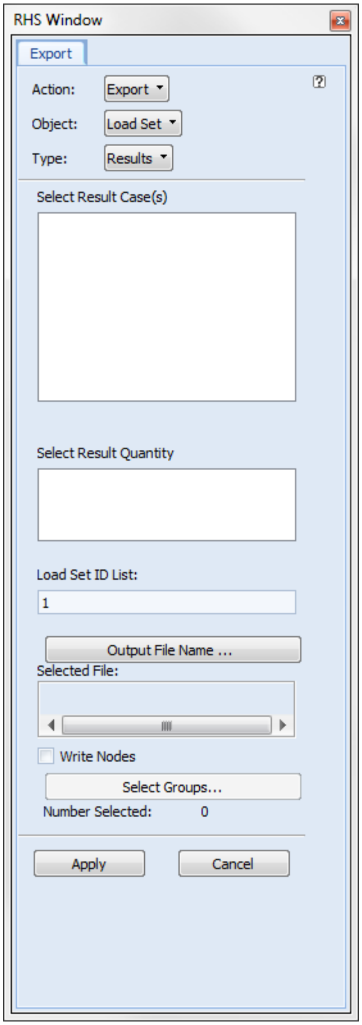
Case/Set	Fx	Fy	Fz	Mx	My	Mz
force.1	0.	0.	-7800.	-2.9176E+5	2.319E+5	0.
force.11	0.	0.	3941.6	1.357E+5	-1.1702E+5	0.
force.12	0.	0.	7800.	1.9912E+5	-2.3233E+5	0.
force.13	0.	0.	-1970.8	-67849.	58512.	0.
force.16	0.	0.	-5850.	-1.4169E+5	1.7425E+5	0.
force.19	0.	0.	-3900.	-1.1486E+5	1.1617E+5	0.
force.20	0.	0.	-7883.2	-2.714E+5	2.3405E+5	0.

Export

Export supports the seamless transfer of loads data, residing as aeroelastic results, to the structural model. The most common method of applying external loads are supported: loads described as MSC.Nastran FORCE/MOMENT entries in a text file. This file may be either "INCLUDED" in future (simple statics) structural analyses or read into an MSC.Patran database as LBCs.

Overview

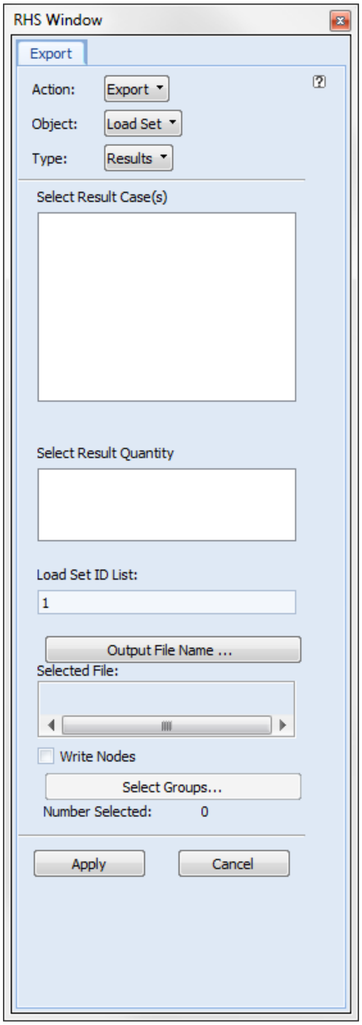
When Export is selected, the Export/LoadSet/Results form is displayed. Note that there are no other Action/Object/Method combinations available for this form.



Exporting loads causes the creation of a text file contained in MSC.Patran FORCE/MOMENT entries. If multiple loads from a single result case are selected, they are linked together by a LOAD entry.

Export

External loads are Exported into a text file containing MSC.Nastran FORCE/MOMENT entries by setting the Action to Export. One or more Result Cases/Quantities can be Exported. An XDB must have been previously attached to provide the aeroelastic loads results used by this capability.

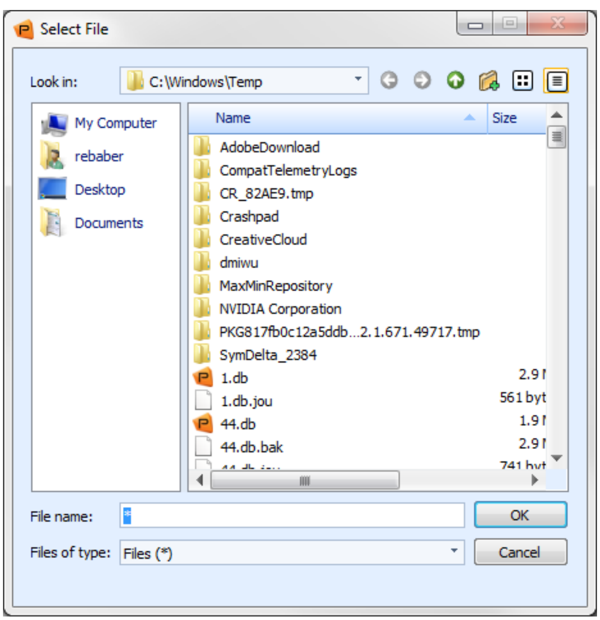


Parameter	Description
Select Result Case(s)	Select one or more Result Cases.
Select Result Quantity	Select one or more Result Quantities. The available Result Quantities are based on those associated with the selected Result Case(s).
Load Set ID List	Controls the output Load Set IDs. The format of this databox is "From: To: By". Default is 1.

Parameter	Description
Output File Name...	Displays the Select File subform.
Write Nodes	"Write Nodes" is off by default. If enabled, then the MSC.Nastran nodes (GRIDs) are included in the text file in the Basic Coordinate System. When you choose to put out the nodes, the resulting file can be reimported into MSC.Patran and an "equivalent node" transfer performed to move the forces to grids at the same location as the current model irrespective of their ID's. Otherwise you can only apply the loads to grids of the same ID.
Select Groups...	Select the structural groups to export. This widget is available when the write nodes toggle is set.
Apply	Apply creates a text file of MSC.Nastran FORCE/MOMENT entries. If more than one Result Quantity is selected, a LOAD entry is also generated for each Result Case. If nodes are selected, then the appropriate GRID entries are written to the text file.

Export - Output File Name Subform

The user selects the name of the text file for exportation.



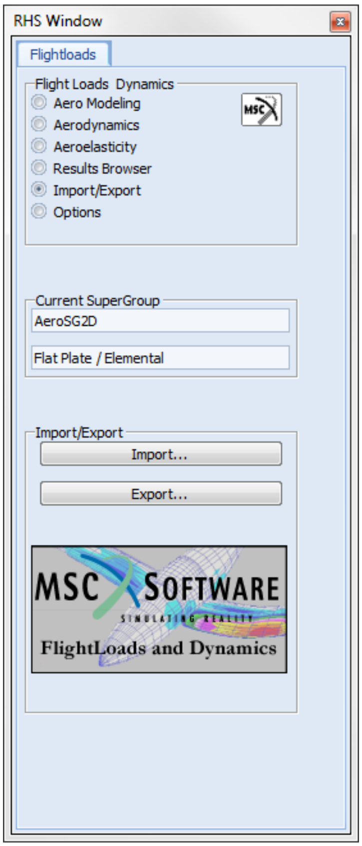
Results Viewer

This button displays the Patran Results forms using the Flexible Increment Filter to filter through the available results when requested.

7 Import/Export

Introduction

Import/Export allows users to communicate model and loads data to and from the current MSC.Patran database.

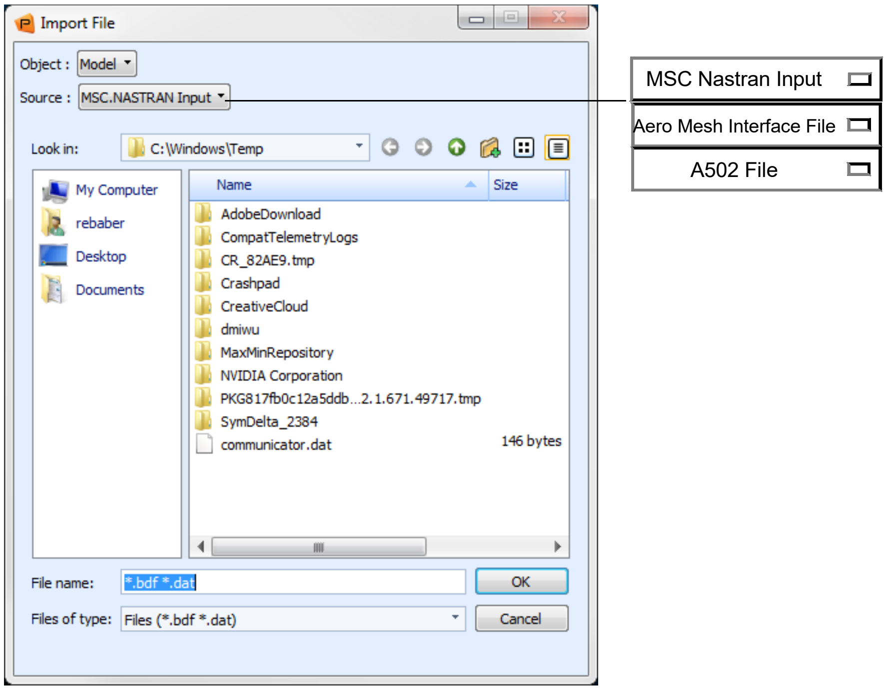


Parameter	Description
Import/Export	Two Import/Export application sections.

Import

The Import capability allows the user to retrieve an aerodynamic model into the current MSC.Patran database described in an MSC.Nastran input file, an Aero Mesh Interface File, or an A502 File.

Model data, including lifting surface, body, control device and splines can be imported by reading an MSC.Nastran input file (.bdf or .dat file). 3D model data can be imported in two different ways. The user can read in an Aero Mesh Interface File (.ami file), see Appendix F for the user can read in an A502 File (.inp file). The import form shown here gives the user access to all three import methods.



Parameter	Description
Object	Determines what is being imported. Currently there is only this one value.
Source	Determines the source of the import file. Its default value is “MSC.Nastran Input”.
SuperGroup Name	Only enabled when “Source” is set to “A502 File”.
Filter	This value gets modified based on the value of the “Source” option menu. The default value is “*.bdf”. If the “Source” option menu is set to “Aero Mesh Interface File”, then this value is changed to “*.ami*” and if it is set to “A502 File”, then this value is changed to “*.inp”.

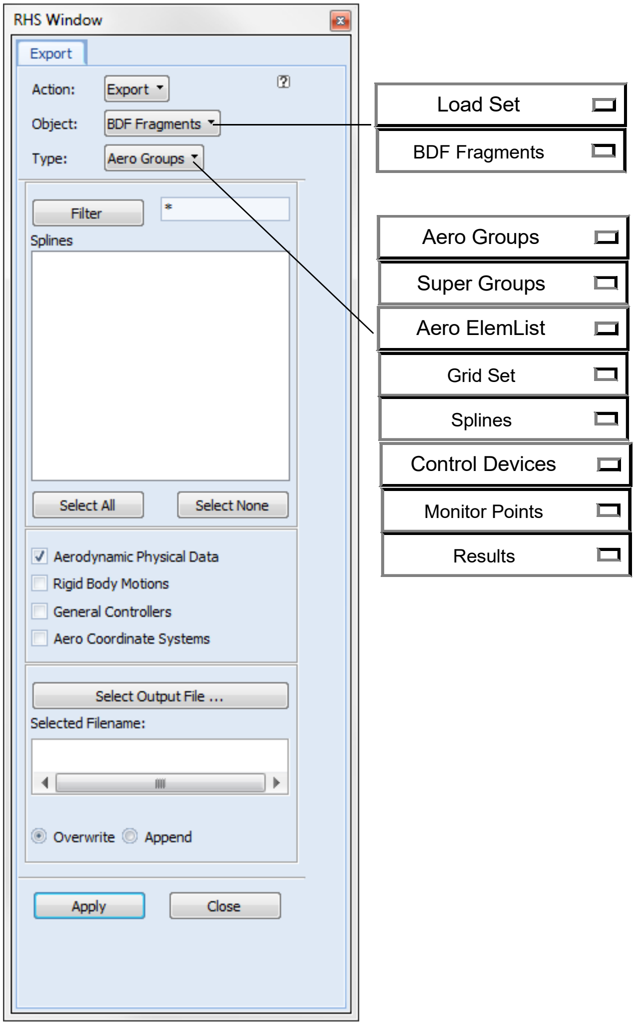
Note:	
	1. Upon reading in a MSC.Nastran input file, the appropriate entities are created in the database as if the user had performed the construction via the user interface. The appropriate session file commands mimicking this construction are created. 2. The aerodynamic coordinate system from the bulk data file as well as hinge line coordinate systems are copied into the MSC.Patran database. However, if a coordinate system of the same ID is already in the database, it is assumed that this coordinate system is to be used and the one from the bulk data file is not imported. 3. For the spline information to be imported, the structural nodes must already exist in the database. If you are importing both the structural and aerodynamic models into the database, you must import the structure first (using the File/Import option of the MSC.Patran Structural Preference) before using the MSC.FlightLoads Import option to obtain the aerodynamic model. If the structural nodes are not present, the aerodynamic model is imported but not the aeroelastic model. 4. Importing of 3D model data causes the appropriate entities to be created in the database. Currently, there is no user interface to allow the users to create 3D models from scratch so no commands are written to the session file.

Export

The ability to export loads data or partial bulk data files is important to help users get data out of the GUI and into some other application.

Overview

When Export is selected, the first form displayed is Export/BDF Fragments/Aero Groups. Shown adjacent to this form are all the different possible Actions, Objects, and Methods for Export.



Not all combinations of Object and Method are valid for this form. Most of the Export forms are shown and annotated in the following pages, grouped by Object as follows:

- Load Set (Results)
- BDF Fragments (all Methods except for Results)

Export

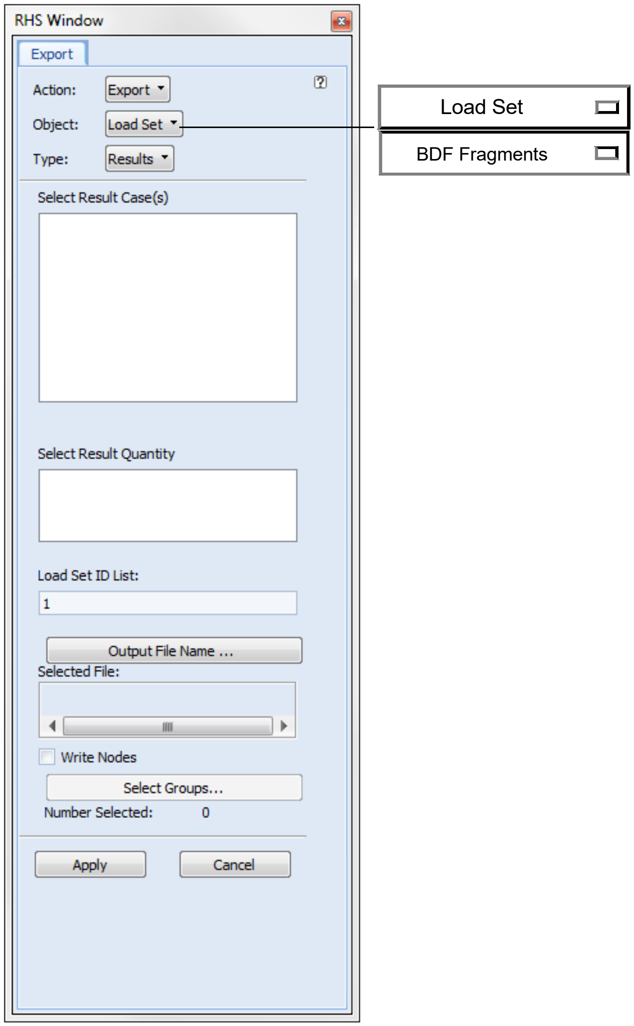
Following is a list of the Export forms and subforms.

- Export/Load Set/Results

  - Select File Subform for Load Set
- Export/BDF Fragments/Aero Groups
- Export/BDF Fragments/Super Groups
- Export/BDF Fragments/Aero Elem List
- Export/BDF Fragments/Grid Set
- Export/BDF Fragments/Splines
- Export/BDF Fragments/Control Devices
- Export/BDF Fragments/
- Export/BDF Fragments/Monitor Points
  - Select File Subform for BDF Fragments
  - Select Surface/Select Groups Subform

Export/Load Set/Results

External loads are exported into a text file containing MSC.Nastran Force/Moment entries by setting the Action to Export. One or more Result Cases/Quantities can be exported. An XDB must have been previously attached for this capability to properly fction.

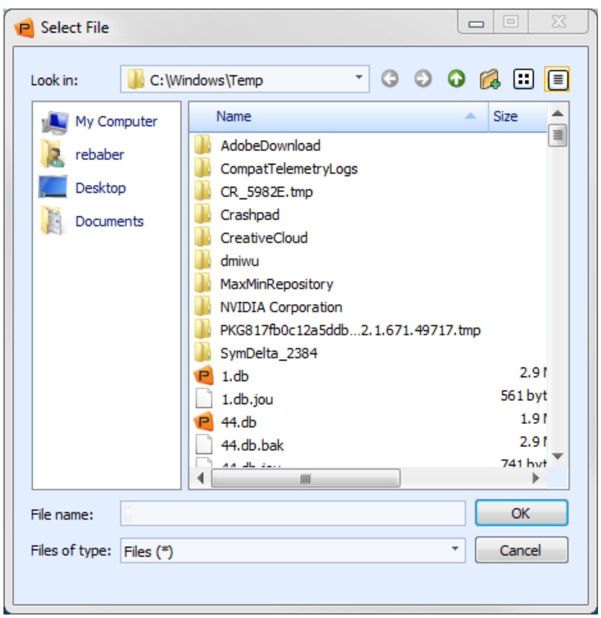


Parameter	Description
Select Result Case(s)	Select one or more Result Cases.
Select Result Quantity	Select one or more Result Quantities. The available Result Quantities are based on those associated with the selected Result Case(s).
Load Set ID List	Controls the output Load Set IDs. The format of this databox is "From: To: By". Default is 1.

Parameter	Description
Output File Name	Displays the File Selection subform.
Write Nodes	"Write Nodes" is off by default. If enabled, then the MSC.Nastran nodes (GRIDs) is included in the text file in the Basic Coordinate System.
Select Groups	Select the Structural Groups to export. This widget is available when the Write Nodes toggle is set.
Apply	APPLY creates a text file of MSC.Nastran FORCE/MOMENT entries. If more than one Result Quantity is selected, a LOAD entry is also generated for each Result Case. If nodes are selected, then the appropriate GRID entries are written to the text file.

Select File Subform

The user selects the name of the text file for exportation from the following.



BDF Fragments

The Export of BDF Fragments allows you to select certain data from FlightLoads to be exported to a text file in MSC.Nastran Bulk Data File format. These "fragments" of BDF files allow access to commonly needed portions of models without requiring you to create a subcase and job.

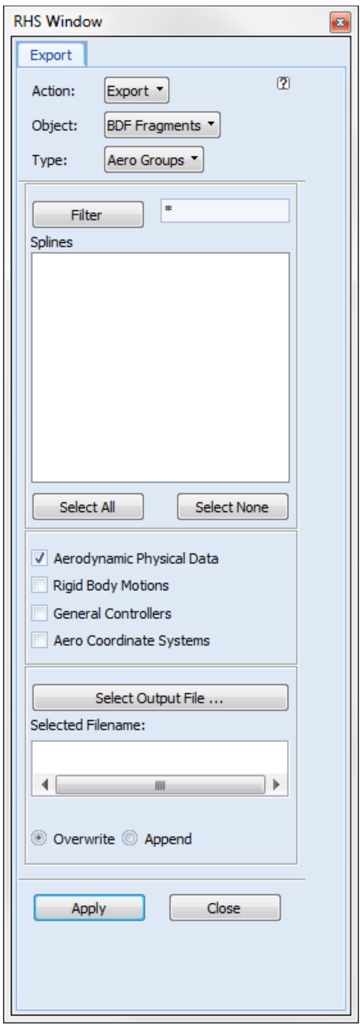
This form is exercised by naming a file and repetitively selecting instances of the various "Method" types. The file will initially default to "Overwrite" status and, following the first pick, will change to "Append" status.

Thus, repetitive "Apply" actions on selections will append to the named file by default. The default can be overridden by explicit selection.

When the selected item requires that more than one BDF entry be created (for example, SPLINE4 refers to an AELIST and SET1 entry), the dependent entries will automatically be created.

Export/BDF Fragments/Aero Groups

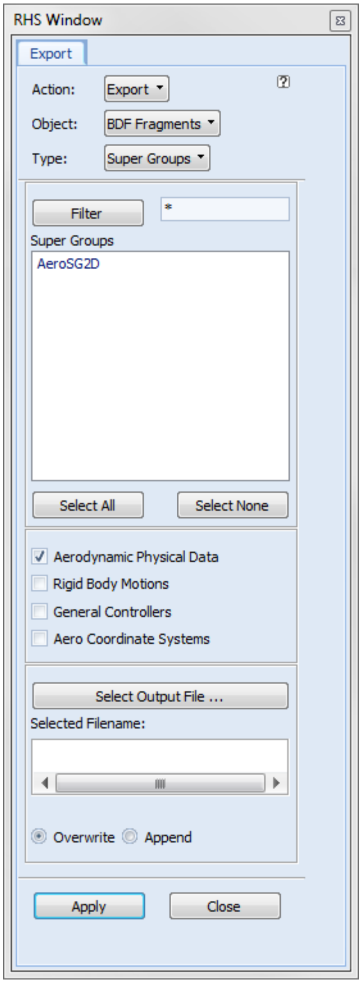
This method allows selection of MSC.FLD aerodynamic components. Selected components are exported as CAEROi and their dependent Bulk Data entries in the BDF fragment file.



Parameter	Description
Select Aero Group(s)	Select the Aero Groups to be exported.
Output File Name	Displays the File Selection subform.
Append	Tells MSC.FLD to either overwrite the selected file if it exists already or append to it.

Export/BDF Fragments/Super Groups

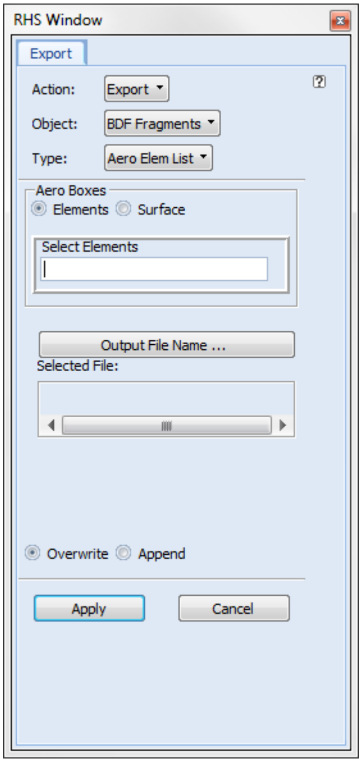
This method allows selection of all an MSC.FLD SuperGroup (the "Aero Groups" that comprise an aerodynamic configuration) and selected SuperGroups become CAEROi and their dependent Bulk Data entries in the BDF fragment file.



Parameter	Description
Select SuperGroup(s)	Select the Aero Groups to be exported.

Export/BDF Fragments/Aero Elem List

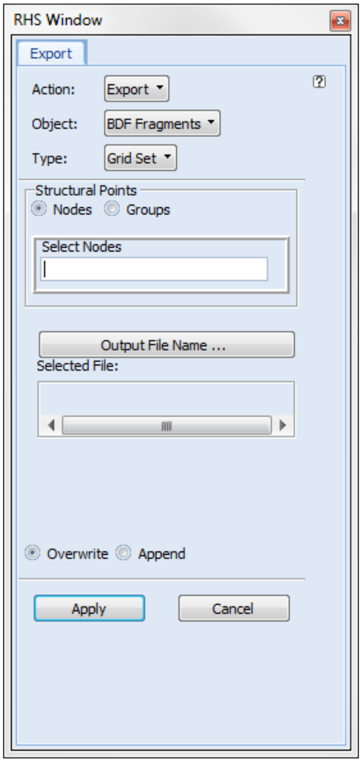
This method allows selection of some number of MSC.FLD aerodynamic elements. Selected elements are then placed on an AELIST entry in the BDF fragment file.



Parameter	Description
Aero Boxes	When the element switch is selected, the user is able to select Aero Elements from the view port or enter them directly into the databox. If the switch is set to Surface, then the select databox is replaced with '**Select Surface**'. This allows the user to select a single Aero Surface for the export.

Export/BDF Fragments/Grid Set

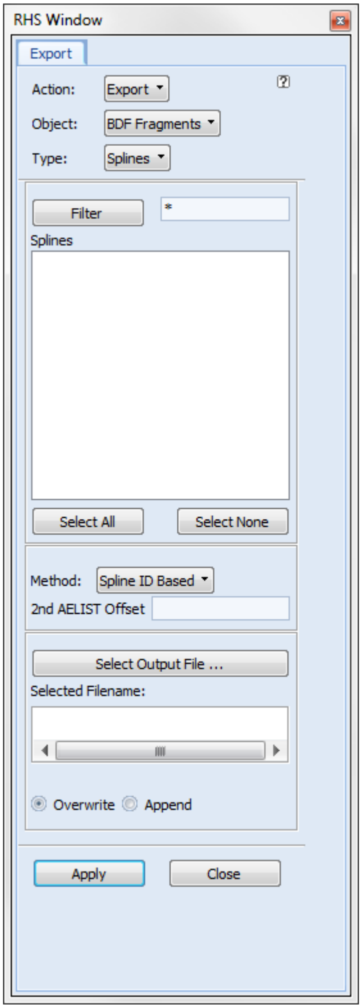
This method allows selection of some number of MSC.FLD structural grids. Selected grids are then placed on a SET1 entry in the BDF fragment file.



Parameter	Description
Structural Points	When the nodes switch is selected, the user is able to select Structural Nodes from the view port or enter them directly into the databox. If the switch is set to Groups, then the select databox is replaced with '**Select Groups**'. This allows the user to select as many structural groups as desired.

Export/BDF Fragments/Splines

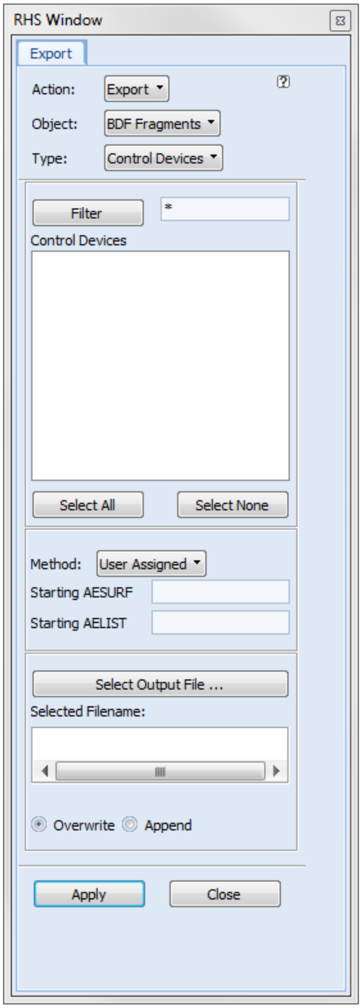
This method allows selection of some number of MSC.FLD splines. Selected splines are then placed on SPLINEi, AELIST and SET1 entries in the BDF fragment file.



Parameter	Description
Select Splines	Select the Splines to be exported.

Export/BDF Fragments/Control Devices

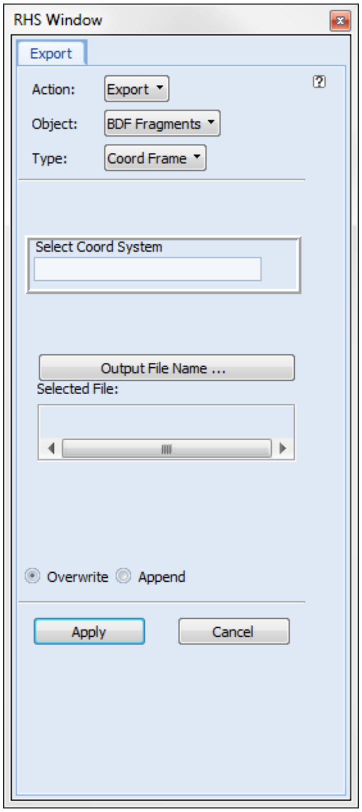
This method allows selection of some number of MSC.FLD control devices. Selected controllers are then placed on AESTAT, AESURF and AEPARM (and dependent entries) in the BDF fragment file.



Parameter	Description
Control Devices	Select the Control Devices to be exported.

Export/BDF Fragments/Coord Frame

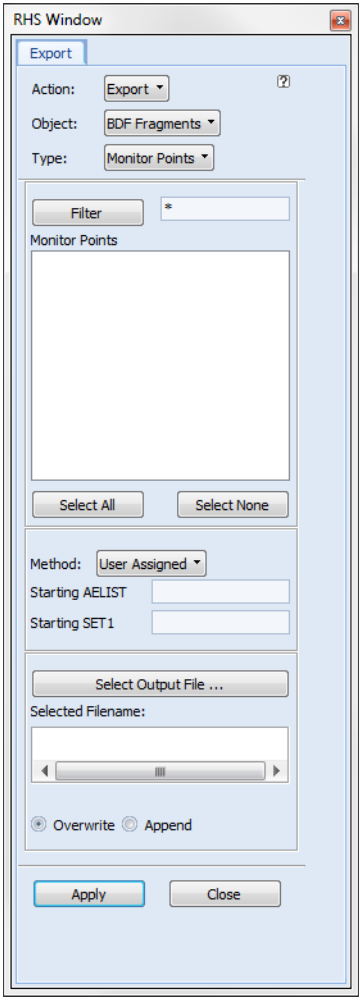
This method allows selection of some number of MSC.FLD coordinate frames. Selected frames are then placed on CORDij entries in the BDF fragment file.



Parameter	Description
Select Coord System	Select the Coordinate System to be exported directly from the viewport or enter it into the databox.

Export/BDF Fragments/Monitor Points

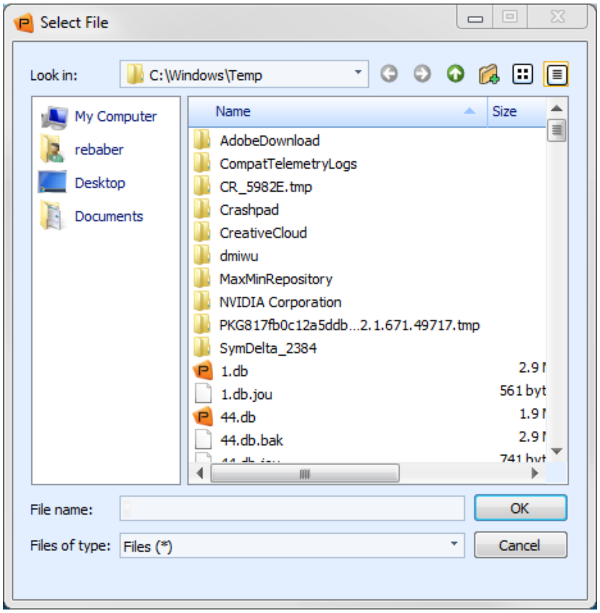
This method allows selection of some number of MSC.FLD monitor points. Selected Monitor Points are then placed on MONPNT1 and their dependent entries in the BDF fragment file.



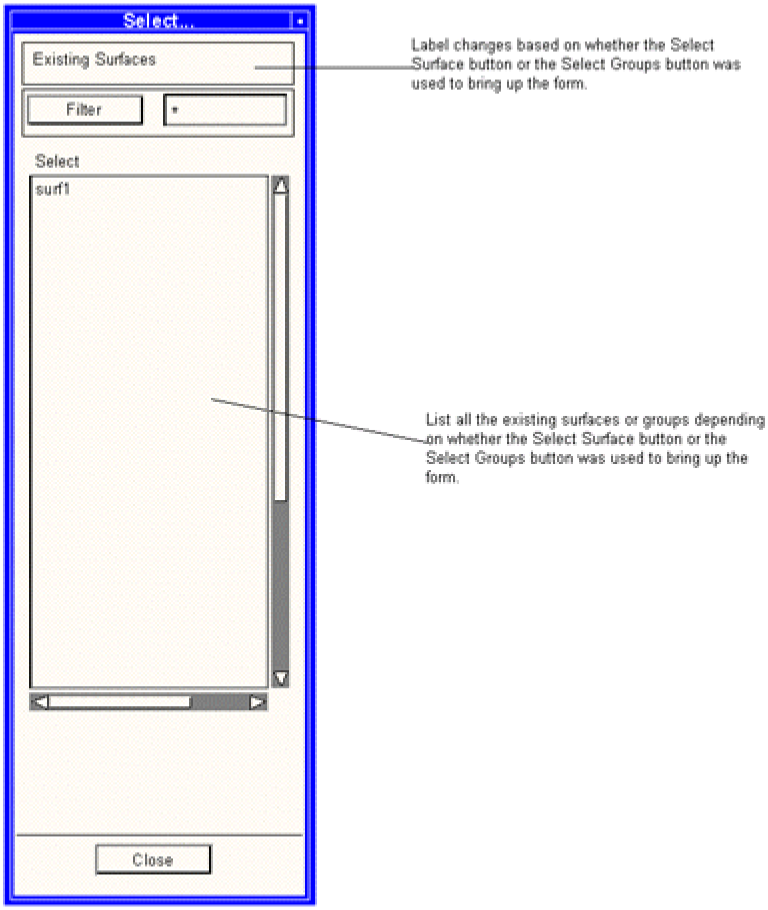
Parameter	Description
Monitor Points	Select the Monitor Points to be exported.

Select File Subform

The user selects the name of the text file that is to be used for the export of the BDF Fragments.



Select Surface/Select Groups Subform


Select...
Existing Surfaces
Filter
*
Select
surf1
Close
Label changes based on whether the Select Surface button or the Select Groups button was used to bring up the form.
List all the existing surfaces or groups depending on whether the Select Surface button or the Select Groups button was used to bring up the form.

A Panel Aerodynamics

Introduction

MSC.FlightLoads and Dynamics Version 1 creates aerodynamic models and produces results that are compatible with the Doublet-Lattice and ZONA51 aerodynamics that are provided in MSC.Nastran. The Doublet Lattice method (DLM) is applicable to subsonic flows while ZONA51 can be considered its supersonic counterpart. These methods are of a of class "Panel Methods" that represent lifting surfaces by flat panels that are nominally parallel to the flow. The Doublet Lattice method has the additional ability to represent bodies as variable radius cylinders, also aligned with the airflow. Other "Panel Methods" can be more general and represent the surfaces as discrete curved surfaces in 3-space.

This Appendix provides a brief description and theoretical discussion of these methods and then offers some guidelines for their use. Much of this development is taken from the *MSC.Nastran User's Guide for Aeroelastic Analysis* (Ref. 12) with corrections and supplementary material provided for this document.

MSC.Nastran aerodynamic analysis, like structural analysis, is based upon a finite element approach. The finite aerodynamic elements are strips or boxes on which there are aerodynamic forces. The aerodynamic elements, even for complex vehicles, must be in regular arrays. In particular, the aerodynamic elements for the lattice methods are arrays of trapezoidal boxes with sides that are parallel to the airflow. These can be described simply by defining properties of the array (panel).

Aerodynamic forces are generated when the flow is disturbed by the flexible vehicle. Theory leads to a matrix that relates the forces acting upon the structure due to the deflections of the structure.

These deflections are the combination of rigid body motions of the vehicle and the structural deformations of the vehicle as it undergoes an applied loading during a maneuver. For the steady flow considered in FlightLoads Static Aeroelastic analysis, the relationship between the deflection and the forces is a function of the aerodynamic model (including any symmetry conditions) and the Mach number of the flow.

Aerodynamic Data Generation

Aerodynamic elements are boxes or segments of bodies that are combined to idealize the vehicle for the computation of aerodynamic forces. These elements, like structural elements, are defined by their geometry and their motions are defined by degrees of freedom at aerodynamic grid points. Requirements of the aerodynamic theory dictate the geometry of the boxes. The Doublet-Lattice (DLM) and ZONA51 methods assume trapezoidal boxes with their edges parallel to the free-stream velocity. By the use of aerodynamic input data, aerodynamic elements and grid points are automatically generated to help ensure that many of the theoretical requirements are met.

Aerodynamic calculations are performed using a Cartesian coordinate system. By the usual convention, the flow is in the positive X-direction, and the X-axis of every aerodynamic element must be parallel to the flow in its undeformed position. (This is an assumption of aerodynamic small disturbance theory.) The structural coordinate systems may be defined independently, since the use of the same system for both may place an undesirable restriction upon the description of the structural model. Any MSC.Nastran Cartesian system may be specified for the aerodynamic coordinates, with the resulting flow defined in the direction of the X-axis. All aerodynamic element and grid point data are transformed to the aerodynamic coordinate system. All the global (displacement) coordinate systems of the aerodynamic grid points will have their T1-directions in the flow direction. Their T3-directions will be normal to the element in the case of boxes, and parallel to the aerodynamic T2- and/or T3-directions in the case of bodies.

The aerodynamic grid points are physically located at the centers of the boxes for the lifting surface theories and at the centers of body elements for the DLM. A second set of grid points, used only for display, is located at the element corners. Grid point numbers are generated based upon the element identification number.

Both sets of grid points are numbered beginning with the user provided ID of the lifting surface. The centroidal grids are numbered from the inboard leading edge box and then incremented by one, first in the chordwise direction and then in the spanwise direction. The corner grid numbering begins at the leading edge inboard corner and again proceeds first chordwise and then spanwise. In terms of the graphical display, the centroidal grids can be thought of as element ID's and corner points as node ID's.

The aerodynamic theories in MSC.Nastran have additional downwash locations that need to be defined here. These points are designated as comprising the j-set of aerodynamic control points. The j-set is not a user set; it is a notational set to identify aerodynamic matrices used in the solution processing. Physically, these are points on the element where the downwash vectors are computed. The location of these points is a function of the aerodynamic method employed:

- For Doublet-Lattice boxes, the downwash point is at the 75% chordwise station and spanwise center of the box. The pressure singularity is computed along the 25% chordwise station.
- For ZONA51 boxes, the downwash point is at the 95% chordwise station and the spanwise center of the box while the pressure in considered constant over the element.
- For Doublet-Lattice interference and slender body elements, the control point and pressure singularity are identically located along the axis of the element and at 50% of its length.

Aerodynamic Theories

MSC.FlightLoads and Dynamics supports three aerodynamic theories:

1. Doublet-Lattice subsonic lifting surface theory (DLM)
2. ZONA51 supersonic lifting surface theory
3. Subsonic wing-body interference theory (DLM with slender bodies)

Each of these methods is described in this section, but they all share a common matrix structure.

Three matrix equations summarize the relationships required to define a set of aerodynamic influence coefficients [see Rodden and Revell (1962)]. These are the basic relationships between the lifting pressure and the dimensionless vertical or normal velocity induced by the inclination of the surface to the airstream; i.e., the downwash (or normalwash),

$$\{w_j\} = [A_{jj}]\{f_j/\bar{q}\} \tag{1-1}$$

the substantial differentiation matrix of the deflections to obtain downwash,

$$\{w_j\} = [D_{jk}]\{u_k\} + [D_{jx}]\{u_x\} + \{w_j^g\} \tag{1-2}$$

and the integration of the pressure to obtain forces and moments,

$$\{P_k\} = [S_{kj}]\{f_j\} \tag{1-3}$$

where:

w_j	downwash
w_j^g	static aerodynamic downwash; it includes, primarily, the static incidence distribution that may arise from an initial angle of attack, camber, or twist
f_j	pressure on lifting element j
$\bar{q}$	flight dynamic pressure
$A_{jj}(m)$	aerodynamic influence coefficient matrix, a function of Mach number (m)
u_k	displacements at aerodynamic grid points
P_k	forces at aerodynamic grid points
D_{jk}	Substantial differentiation matrix for aerodynamic grid deflection (dimensionless)
$[D_{jx}]$	substantial derivative matrix for the extra aerodynamic points
$\{u_x\}$	vector of "extra aerodynamic points" used to describe, e.g., aerodynamic control surface deflections and overall rigid body motions
S_{kj}	integration matrix

The three matrices of (1-1), (1-2), and (1-3) can be combined to give an aerodynamic influence coefficient matrix:

$$[Q_{kk}] = [S_{kj}][A_{jj}]^{-1}[D_{jk}] \tag{1-4}$$

which relates the force at an aerodynamic grid point to the deflection at that grid point and a rigid load matrix:

$$[Q_{kx}] = [S_{kj}][A_{jj}^{-1}][D_{jx}] \tag{1-5}$$

which provides the force at an aerodynamic grid point due to the motion of an aerodynamic extra point.

All methods compute the S_{kj}, A_{jj}, and D_{jk} matrices as a function of Mach number. The D_{jx} matrix is only a function of the model geometry and is therefore calculated only once per configuration.

Doublet-Lattice Subsonic Lifting Surface Theory

The Doublet-Lattice method (DLM) can be used for interfering lifting surfaces in subsonic flow. The theory is presented by Albano and Rodden (1969), Giesing, Kalman, and Rodden (1971), and Rodden, Giesing, and Kalman (1972) and is not reproduced here. The following general remarks summarize the essential features of the method.

The theoretical basis of the DLM is linearized aerodynamic potential theory. The undisturbed flow is uniform and is either steady or varying (gusting) harmonically. All lifting surfaces are assumed to lie nearly parallel to the flow. The DLM is an extension of the steady Vortex-Lattice method to unsteady flow.

Each of the interfering surfaces (or panels) is divided into small trapezoidal lifting elements ("boxes") such that the boxes are arranged in strips parallel to the free stream with surface edges, fold lines, and hinge lines lying on box boundaries. The unknown lifting pressures are assumed to be concentrated uniformly across the one-quarter chord line of each box. There is one control point per box, centered spanwise on the three-quarter chord line of the box, and the surface normalwash boundary condition is satisfied at each of these points.

The code for computing the aerodynamic influence coefficients A_{jj} was taken from Giesing, Kalman, and Rodden (1972b). Any number of arbitrarily shaped interfering surfaces can be analyzed, provided that each is idealized as one or more trapezoidal planes. Aerodynamic symmetry options are available for motions which are symmetric or antisymmetric with respect to one or two orthogonal planes. The user may supply one-half (or one-fourth) of the model and impose the appropriate structural boundary conditions. The full aircraft can also be modeled when the aircraft or its prescribed maneuvers lack symmetry.

ZONA51 Supersonic Lifting Surface Theory

ZONA51 is a supersonic lifting surface theory that accounts for the interference among multiple lifting surfaces. It is an optional feature in MSC.Nastran (available as the Aero II option). It is similar to the Doublet-Lattice method (DLM) in that both are acceleration potential methods that need not account for flow characteristics in any wake. An outline of the development of the acceleration-potential approach for ZONA51 is presented by Liu, James, Chen, and Pototsky (1991), and its outgrowth from the harmonic gradient method (HGM) of Chen and Liu (1985) is described. ZONA51 is a linearized aerodynamic small disturbance theory that assumes all interfering lifting surfaces lie nearly parallel to the flow, which is uniform and either steady or gusting harmonically. As in the DLM, the linearized supersonic theory does not account for any thickness effects of the lifting surfaces.

Also, as in the DLM, each of the interfering surfaces (or panels) is divided into small trapezoidal lifting elements ("boxes") such that the boxes are arranged in strips parallel to the free stream with surface edges, fold lines, and hinge lines lying on box boundaries. The unknown lifting pressures are assumed to be uniform on each box. There is one control point per box, centered spanwise on the 95 percent chord line of the box, and the surface normalwash boundary condition is satisfied at each of these points.

The code for computing the aerodynamic influence coefficients, A_{jj}, was integrated into MSC.Nastran by Zona Technology, Inc., taking full advantage of the extensive similarities with the DLM. Any number of arbitrarily shaped interfering surfaces can be analyzed, provided that each is idealized as one or more trapezoidal planes. Aerodynamic symmetry options are available for motions that are symmetric or antisymmetric with respect to the vehicle centerline. Unlike the DLM, symmetry about the XY-plane is not supported. The user may supply one half of the vehicle model and impose the appropriate structural boundary conditions.

Subsonic Wing-Body Interference Theory

The method of images, along with Slender Body Theory, has been added to the Doublet-Lattice method (DLM) in Giesing, Kalman, and Rodden (1972a, 1972b, and 1972c). The DLM is used to represent the configuration of interfering lifting surfaces, while Slender Body Theory is used to represent the lifting characteristics of each body (i.e., fuselage, nacelle, or external store). The primary wing-body interference is approximated by a system of images of the DLM trailing vortices and doublets within a cylindrical interference body that circumscribes each slender body. The secondary wing-body interference that results from the DLM bound vortices and doublets is accounted for by a line of doublets located on the longitudinal axis of each slender body. The boundary conditions of no flow through the lifting surfaces or through the body (on the average about the periphery) lead to the equations for the lifting pressures on the surfaces and for the longitudinal (and/or lateral) loading on the bodies in terms of the normalwashes on the wing-body combination.

The code for computing the aerodynamic matrices was adapted for MSC.Nastran from Giesing, Kalman, and Rodden (1972b). The adaptation required a matrix formulation of all of the body interference and body loading calculations. These equations are written using the symbols adopted for MSC.Nastran and showing the equivalences to names used in the documentation of Giesing, Kalman, and Rodden (1972b).

The program of Giesing, Kalman, and Rodden (1972b) finds the forces on the lifting boxes and bodies of an idealized airplane in terms of the motions of these elements. The lifting surfaces are divided into boxes. The bodies are divided into elements. There are two types of body elements: slender elements, which are used to simulate a body's own motion, and interference elements, which are used to simulate the interaction with other bodies and boxes. The body elements may have Z (vertical), Y (lateral), or both (ZY) degrees of freedom.

The basic method is the superposition of singularities and their images. There are two basic singularity types: "forces" and modified acceleration potential "doublets." Each "force" singularity is equivalent to a line of doublets in the wake. As discussed, the wing boxes use the "force" type of singularity concentrated along the box quarter chord. The interference elements use the "doublet" type of singularity. The slender body elements use both types.

An extensive set of matrix equations dealing with slender body theory as adapted to the Doublet Lattice Method are described in the MSC.Nastran Aeroelastic Guide, and are not reproduced here.

Aerodynamic Modeling Guidelines

Aerodynamic elements are regions of lifting surfaces or bodies. Since the elements occur in regular streamwise arrays, the Aerodynamic Modeling procedures of Aero Modeling (Ch. 3) are to be used to generate the aerodynamic connection data (CAEROi) bulk data entries that specify these arrays to MSC.Nastran. The bulk data entries that comprise the aerodynamic model are discussed in this section, as well as guidelines for their creation.

For every aerodynamic problem, pertinent basic flight and geometric parameters are specified on the AEROS entry. This entry includes the Aerodynamic Coordinate System (input Options Input in the Aero Modeling module), the Reference Coordinate System (input as the Rigid Body Coord Frame as Global Data within the Aeroelasticity Module), the Reference Chord, Span and Area (also entered as Global Data), and the symmetry options (input as Trim Parameters as part of the subcase create in the Aeroelastic Analysis form of the Aeroelasticity module). Note that FlightLoads always requests the full reference area even though the AEROS entry needs the half model area for half-span model. The necessary translation occurs when the bulk data are created. A rectangular aerodynamic coordinate system must be identified. The flow is in the positive X-direction in this system and parallel to the plane of the aerodynamic elements. The use of symmetry (or antisymmetry) is available to analyze structures that have both stiffness and inertial symmetry, to simulate ground effects, and to simulate wind tunnel wall effects. The planes of symmetry are taken from the aerodynamic coordinate system. The user must ensure that the aerodynamic reference system has the same symmetries. Any consistent set of units can be used for the dimensional quantities.

The types of elements available are shown in Table 1-1. Every CAEROi entry must reference an aerodynamic property (PAEROi) data entry that is used to list additional parameters. Tabulations of numbers or other defining parameters are sometimes required, depending on the selected aerodynamic method, and these are listed on AEFACT entries. These lists include division points (for unequal box sizes) and a variety of other parameter values.

Table 1-1 MSC.Nastran Aerodynamic Elements

	Aerodynamic Theory		
Attribute	Doublet-Lattice Panel	Lifting Body (Interference)	ZONA51 Panel
Bulk Data Entries	CAERO1 PAERO1	CAERO2 PAERO2	CAERO1 PAERO1
Mach Number	Subsonic	Subsonic	Supersonic
Symmetry Options (Aero flow coordinate system)	Two Planes y = 0 z = 0	Two Planes y = 0 z = 0	One Plane y = 0
Interaction	Panels and Bodies in the Same Group		Panels in the Same Group
Interconnection to Structure	Box Centers	Slender Body Centers	Box Centers
Displacement Components Used at Connection Points	3,5	3,5 z-Bodies 2,6 y-Bodies	3,5

Doublet-Lattice and ZONA51 Panels

The configuration is divided into planar trapezoidal panels (macro-elements), each with a constant dihedral and with sides parallel to the airstream direction. These panels are further subdivided into "boxes" (see Figure 1-1), which are similarly configured trapezoids. The following guidelines are not enforced by the program; the user is expected to ensure adherence to these rules.

If a surface lies in (or nearly in) the wake of another surface, then its spanwise divisions should lie along the divisions of the upstream surface. The strips near the intersection of intersecting surfaces should have comparable widths. The aspect ratio of the boxes should be approximate unity; less than three is acceptable in the subsonic case, and of order one is desirable in the supersonic case. Boxes should be concentrated near wing edges and hinge lines or any other place where downwash is discontinuous and pressures have large gradients. [Note that concentrating boxes near hinge lines is a requirement of Potential Theory (which neglects viscous effects). Not increasing the concentration of boxes near hinge lines lowers the calculated control surface effectiveness and leads to closer agreement with experimental data.] The chord lengths of adjacent boxes in the streamwise direction should change gradually.

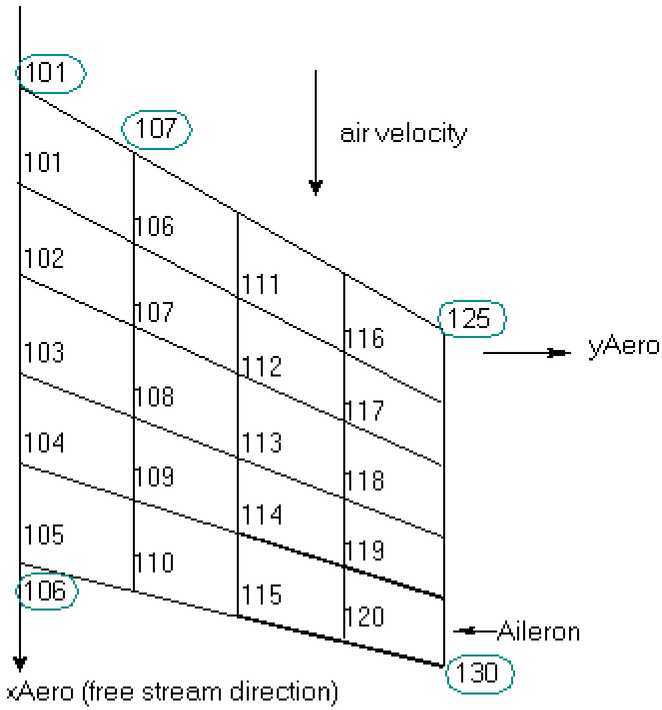


Figure 1-1 An Aerodynamic Doublet-Lattice/ZONA51 Panel Subdivided into Boxes. The number in the boxes are the aerodynamic grid IDS. The circled numbers are selected IDs for the aerodynamic mesh points.

Aerodynamic panels are assigned to interference groups. All panels within a group have aerodynamic interaction. The purpose of the groups is to reduce the computational effort for aerodynamic matrices when it is known that aerodynamic interference is important within the group but otherwise is negligible or to allow the analyst to investigate the effects of aerodynamic interference.

Each panel is described by a CAERO1 Bulk Data entry. A property entry PAERO1 is used to identify associated interference bodies in the subsonic case. A body should be identified as a member of the group if the panel is within one diameter of the surface of the body. The box divisions along the span are determined either by specifying the number of equal boxes NSPAN, or by identifying (by LSPAN), the AEFACT data entry that contains a list of division points in terms of fractions of the span. A similar arrangement is used to specify divisions in the chordwise direction by choosing NCHORD or LCHORD. The locations of the two leading edge points are specified in the coordinate system (CP) defined by the user (including basic). The sides (chords) are in the airstream direction (i.e. parallel to the X-axis of the aerodynamic coordinate system specified on the AEROS Bulk Data entry). Every panel must be assigned to an interference group (IGID). If all panels interact, then IGID must be the same for all panels.

There is an aerodynamic grid point with its associated degrees of freedom in plunge and pitch for each box within a given panel. These points are located at the center of each box and are automatically numbered and sequenced by the program. The lowest aerodynamic grid point number for a given panel is automatically assigned the same number as specified for the panel ID field on the CAERO1 entry starting with the box connected to point 1. The grid point numbers increase in increments of 1 (see the CAERO1 Bulk Data entry description) first in the chordwise direction and then spanwise over all boxes in the panel. There are also corner grid points that are spawned in order to display the aerodynamic mesh. These grid IDs also begin with panel ID at point 1 and are incremented by 1, first in the chordwise and then the spanwise direction over all points on the mesh. (Figure 1-1). Note that to insure uniqueness, panel IDs must be separated enough that mesh IDs do not overlap. For example, subsequent panel IDs associated with the panel given in Figure 1-1 must be > 130. As a matter of practice, the user should always allow more spacing in numbering so that, during mesh convergence studies, each panel can be independently incremented. The local displacement coordinate system has component T1 in the flow direction and component T3 in the direction normal to the panel in the element coordinate system defined on the CAERO1 entry.

Spanwise Convergence Criteria

Guidelines for modeling the spanwise cuts for the mesh can be gained from an investigation that was made of the steady case, i.e., the Vortex-Lattice Method (VLM) to which the DLM reduces at zero frequency. It showed slow convergence as the number of strips was increased. This has been observed in the past by Hough (Ref. 8 and Ref. 9) who showed improved convergence by following a suggestion of Rubbert (Ref. 14) that equally spaced lattices on a surface should be inset from the tip by a fraction of the lattice span. Hough observed that dramatically improves the convergence. It must be stressed that this inset entails a change in the lifting surface geometry and has the undesirable effect of making this geometry a function of the mesh.

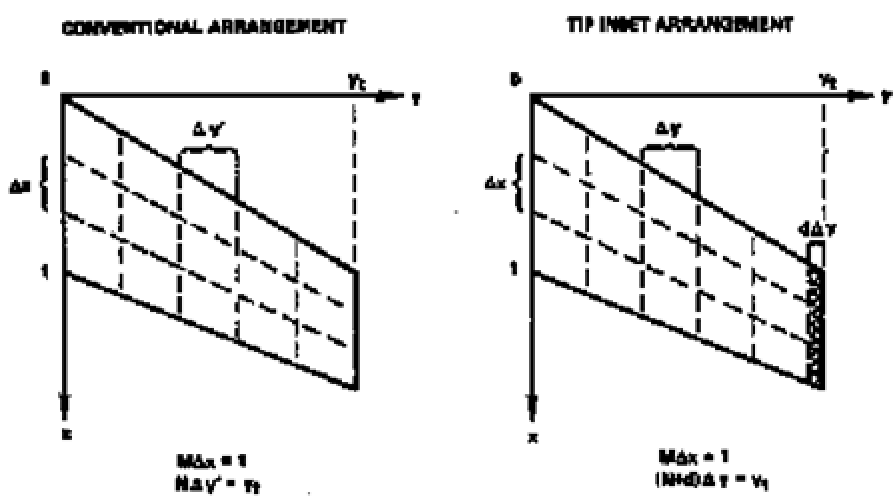


Figure 1-2 Vortex-Lattice Layout

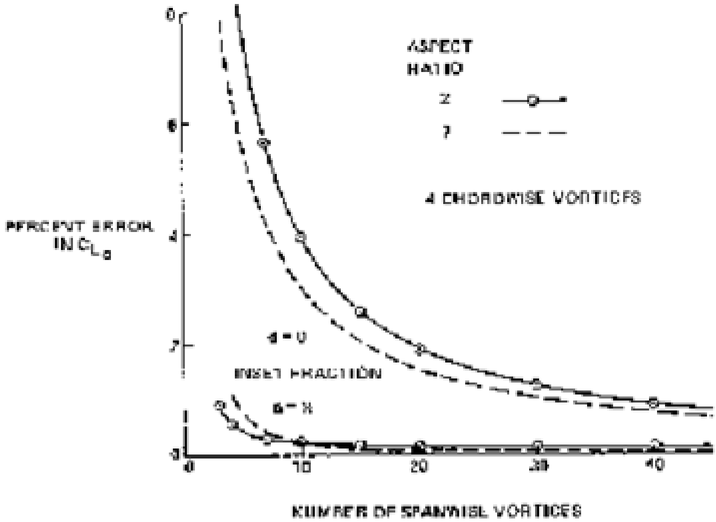


Figure 1-3 Lift-Curve Slope Convergence for Rectangular Wing

With a reasonable number of strips on a surface, say 20 to 30 the error is not large, perhaps 1 to 2% depending on the configuration, and the tip correction may not be worth the effort.

It should be noted that Hough's argument leading to d=1/4 is based on a symmetrical (elliptical) loading in the steady case and with equal spanwise cuts. Limited calculations indicate that it is also valid in the antisymmetrical (in roll) loading, so the tip correction is recommended in general. Users are advised to perform some convergence checks, with and without the tip correction, they may also wish to investigate placing a refined spanwise mesh at the wing tip without the inset.

If users wish to make the tip correction they can reduce the span of the most outboard panel on each surface on the corresponding CAERO1 continuation entry by the factor NSPAN/(NSPAN + d) where NSPAN is the number of strips on the panel and d is the inset fraction (d = 0.25 is recommended).

Slender and Interference Bodies

In subsonic problems, bodies are idealized as "slender" and "interference" elements in combination. The primary purpose of the slender body elements is to account for the forces arising from the motion of the body, whereas the interference body elements are used to account for the interference among all bodies and panels in the same group. This is done by providing a surface through which the boundary condition of no flow is imposed. Bodies are further classified as to the type of motion allowed.

In the aerodynamic coordinate system, Y and Z are perpendicular to the flow. In general, bodies may move in both the Y- and Z-directions. Frequently, a body (e.g., a fuselage) lies on a plane of symmetry and only Z- (or Y-) motion is allowed.

Note, however, that MSC.FlightLoads and Dynamics automatically captures the singularities. Thus a ZY-body on the XZ plane of symmetry in symmetric analysis will act as a Z-body without user interaction; in antisymmetric analysis, the same body will act as a Y-body. Thus, any model may contain Z-bodies, ZY-bodies, and Y-bodies.

One or two planes of symmetry or antisymmetry may be specified. Figure 1-4 and Figure 1-5 shows an idealization with bodies and panels. This example is the one used to illustrate the Doublet-Lattice program in Giesing, Kalman, and Rodden (1972b, pages 19-42). It has a fuselage, a wing, a pylon, and a nacelle.

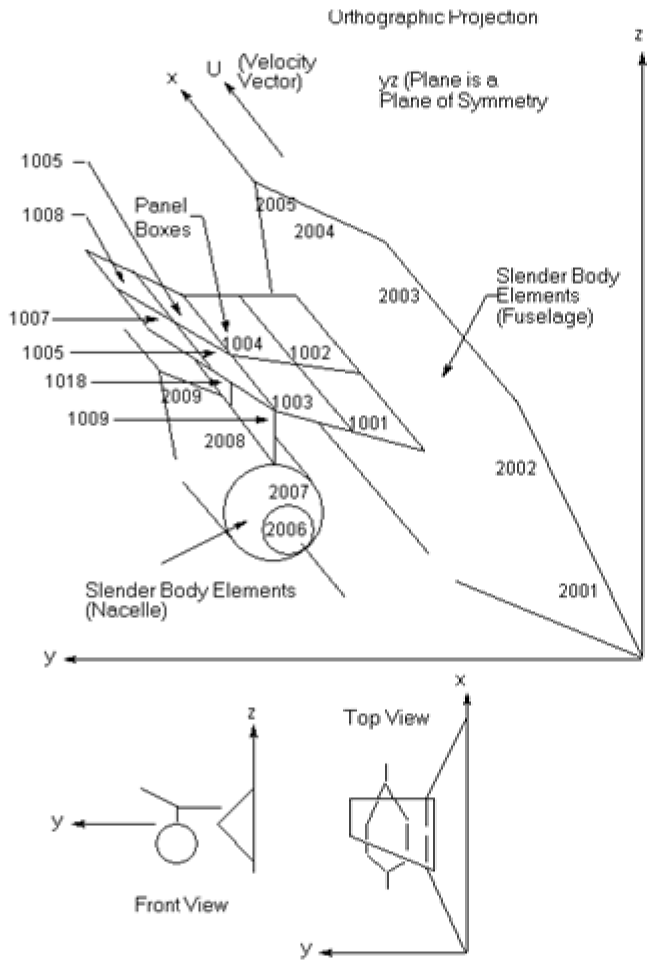


Figure 1-4 Illustration of Boxes and Slender Body Elements. N5KA Bomber Example with Three Panels, Ten Boxes, Two Bodies, Nine Slender Body Elements, and Seven Interference Elements

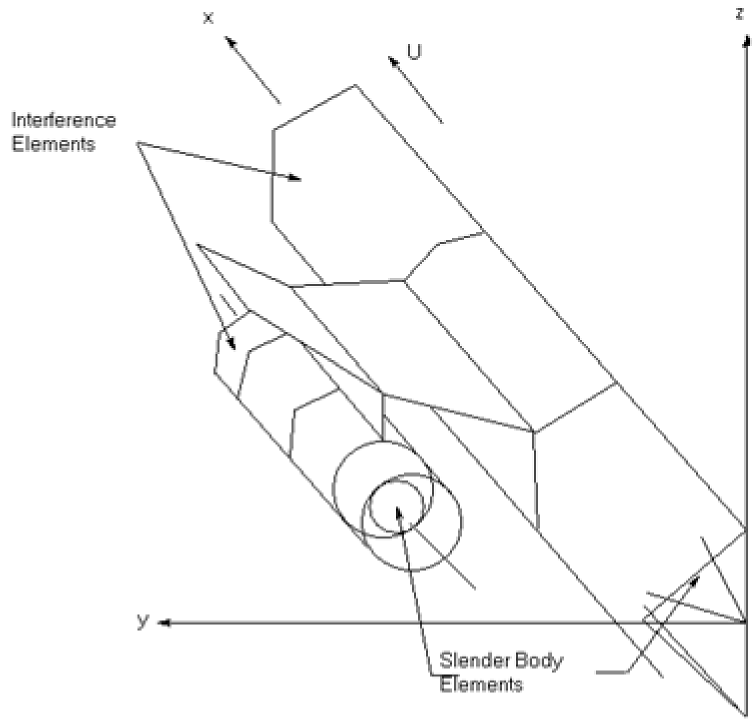


Figure 1-5 Illustration of Interference Elements. N5KA Bomber Example with Three Panels, Ten Boxes, Two Bodies, Nine Slender Body Elements, and Seven Interference Elements

The PAERO1 Bulk Data entry lists the IDs of all the bodies that are associated (i.e., interfere) with a given Doublet-Lattice panel (CAERO1 entry). The CAERO2 entry specifies the geometry and divisions for the slender body and interference elements. The PAERO2 entry provides orientation and cross-section data for the slender body and interference elements as well as the sampling data to account for the residual flow discussed later. The location of the body nose and the length in the flow direction are given. The slender body elements and interference elements are distinct quantities and must be specified separately. At least two slender body elements and one interference element are required for each body. The geometry is given in terms of the element division points, the associated width, and a single height-to-width ratio for the entire body length. The locations of the division points may be given in dimensionless units or, if the lengths are equal, only the number of elements need be specified. The body may be divided along its length unequally to characterize the lift distribution, noting that Slender Body Theory gives a lift proportional to the rate of change of cross-section area. Shorter elements should be chosen at the nose where the area is changing rapidly; longer elements can be used along cylindrical regions where the area is constant and intermediate length elements can be used in transition regions. The semi-widths of the slender body at interference element boundaries can be specified separately and are given in units of length. Usually the slender body semi-width is taken as zero at the nose and is a function of X, while the interference body semi-width is taken to be constant.

The interference elements are intended for use only with panels and/or other bodies, while slender body elements can stand alone. Grid points are generated only for the slender body elements. The first grid point is assigned the ID of the body corresponding to the element at the nose and other grid points are incremented by one. The user should be cautious about the use of associated interference bodies since they increase computational effort significantly.

A brief review of the Method of Images (and its approximations) follows before the implementation of the method in MSC.Nastran is discussed.

The interference elements provide the basis for the internal image system that cancels most of the effects of the trailing vortices from the lifting surfaces. Because of the two-dimensional basis for this approximation (Thompson's Circle Theorem in Hydrodynamics), the body surface has been approximated by a constant elliptical cross-section cylinder called the interference tube. All panels that intersect a body must be attached to the interference tube. Image locations are computed from the semi-width of the interference tube for all lifting surfaces associated with the body. The image is only computed if it lies between the front and aft of the interference element for the associated body.

There is a residual flow "through" the body surface because the image system, being based on two-dimensional considerations, only partially cancels the flow through the body surface. It does not compensate for the effects of the bound vortices on the lifting surfaces or other bodies. Additional unknown "residual" doublets are located along the axis of the body, and, when determined, are added to the known doublet strengths of the slender body elements. The residual flow is calculated by "sampling" the vertical or side velocity components from the net effect of the surface, slender body, and image vortices or doublets. The sampling is performed at various angular positions around the periphery of the elliptical interference tube at the end points of the interference elements. The strengths of the "residual" doublets are then determined to cancel the net velocity.

The calculation of the velocity field induced by the residual doublets requires knowledge of the geometry of the cross section of the slender body at the end points of the interference elements. However, experience shows that the residual flow is small compared to the slender body flow field so that the residual flow need not be represented accurately. This permits the further approximation of simply using the geometry of the constant cross-section interference tube in the calculation of the velocities induced by the residual doublets.

A discussion of two related problems follows:

The requirement for a constant cross-section interference tube may require moving the stabilizer (or wing); see Giesing, Kalman, and Rodden (1972a, §2.5.8). MSC.Nastran can accommodate the requirements by specifying a stabilizer coordinate system, two sets of GRID points for the (same) stabilizer root and its fuselage connection, and MPCs constraining the motions of the two sets of GRIDs to be the same. In this way, the structure can be modeled faithfully although the aerodynamic model is only approximate. This is illustrated in the example "FSW Airplane with Bodies" (HA144F) in Chapter 7 of the *MSC.Nastran Aeroelastic User's Guide*.

The idealization of a jet engine installation as a slender body results in a mass flow ratio through the engine of zero, since there is no flow through the body. Idealizing the engine as a ring-wing results in a mass flow ratio of unity, since all the flow goes through the tube. A typical mass flow ratio is 0.7, so a ring-wing representation is more appropriate.

B Splines

Introduction

In the context of MSC.FlightLoads and Dynamics, Splines provide an interpolation capability that couples the disjoint structural and aerodynamic models in order to enable the static aeroelastic analysis. The aeroelastic splines are used for two distinct purposes: as a force interpolator to compute a **structurally equivalent** force distribution on the structure given a force distribution on the aerodynamic mesh and as a displacement interpolator to compute a set of aerodynamic displacements given a set of structural displacements. The force interpolation is represented mathematically as:

$$[F_s] = [G_{sa}][F_a] \tag{2-1}$$

and the displacement interpolation as:

$$[U_a] = [G_{as}][U_s] \tag{2-2}$$

Where ***G*** is the spline matrix, ***F*** and ***U*** refer to forces and displacements, respectively and the ***s*** and ***a*** subscripts refer to structure and aerodynamics, respectively.

The two splines given in the above relationships are used when making the force and displacement interpolations. However, virtual work principals can be applied to relate the two splines as being the transform of one another:

$$[G_{sa}] = [G_{as}]^T \tag{2-3}$$

That is, the same set of aerodynamic and structural degrees of freedom are coupled for both interpolation functions. While this relationship is valid, this usage assumption is not necessary and can be limiting for static aeroelastic applications where the set of structural DOFs that is appropriate for load application may not be the same set that is appropriate to represent the important deflections for the aeroelastic correction. Therefore, as shown with the **Type** option of Aero-Structure Coupling (Ch. 5), each spline can be either General (same spline used for Force and Displacement), Displacement or Force.

Splining methods for aeroelastic analyses available in FlightLoads include the Harder-Desmarais Infinite Plate Spline (SPLINE1 or SPLINE4 with METH=IPS), the Infinite Beam Spline (SPLINE2),the Thin Plate Spline (SPLINE1 or SPLINE4 with METH=TPS) and the Finite Plate Spline (SPLINE1 or SPLINE4 with METH=FPS). A fifth method that employs an MPC-like interpolator (SPLINE3) is available in MSC.Nastran, but is not supported in FlightLoads. The SPLINE3 allows the user to build an interconnection between select aerodynamic DOFs and select structural DOFs and is not discussed further here.

The IPS, FPS and the linear spline assume that the aerodynamic and structural points for a single interpolation matrix lie on or can be projected to the same plane. They relate structural displacements normal to that plane to aerodynamic displacements normal to the plane and to an aerodynamic slope (rotation about a single axis lying in the plane). The TPS is a three dimensional extension of the existing IPS spline. The FPS uses a virtual planar finite element mesh to interpolate between the two meshes.

As stated in (2-1) and (2-2), the two basic relationships that must be developed are the displacement transformation and the force transformation. In general, the structural displacements are the usual six global displacement degrees of freedom and the forces are the usual three forces and three moments. The aerodynamic degrees of freedom depend on the aerodynamic method, but must include displacements normal to a local surface and rotations about an axis lying in the osculatory plane since these are the degrees of freedom used in the aerodynamic methods of Panel Aerodynamics (App. A). The corresponding aerodynamic forces are a normal force and a local pitching moment.

Each set of structural points and aerodynamic points may be related via a pair of unique spline transformations of the form of (2-1) and (2-2). The total transformation matrices for all the aerodynamic and structural DOFs are then assembled from the individual spline matrices. In FlightLoads, the structural points are taken as the independent degrees of freedom in the spline relationships, so the same structural point may appear in more than one spline relation. However, each aerodynamic point may appear in only one.

The force transformation must be computed such that the resultant structural loads are statically equivalent to the aerodynamic loads:

$$\sum_{i=1}^{3ns} [TBG]_i [F_s]_i = \sum_{j=1}^{3na} [TBA]_j [F_a]_j \tag{2-4}$$

where $[TBG]$ is the transformation from global to basic coordinates. For moments, the following condition must be satisfied:

$$\sum_{i=1}^{3ns} [r]_i \times [TBG]_i [F_s]_i = \sum_{j=1}^{3na} [r]_j \times [TBA]_j [F_a]_j \tag{2-5}$$

where the $[r]_i$, $[r]_j$ are, respectively, the vectors between the (arbitrary) moment center and the structural and aerodynamic mesh points in the basic coordinate system. These two requirements are imposed on the individual spline matrices on a component-by-component basis, thus ensuring that the relationship will hold for the assembled spline transformation.

Each of the spline methods yields a relationship:

$$\{U(x,y,z)\} = [R]\{a\} + [\bar{A}]\{P\} = [C][P] \tag{2-6}$$

where $[R]\{a\}$ are the weighted coefficients of the interpolant (usually determined by boundary conditions on the function, e.g., equilibrium) and $[\bar{A}]\{P\}$ are the coefficient matrix and the applied load respectively. The coefficient matrix is a function only of geometry and the form of the interpolant. The evaluation uses the structural geometry alone in (2-6) to evaluate the coefficients:

$$\{P_s\} = [C_{ss}]^{-1}\{U_s(x,y,z)\} \tag{2-7}$$

and then uses (2-6) again, with both geometries to evaluate the displacement function at the aerodynamic points given the solution of (2-7) (which are loads at the structural grids) for point displacements at the structural grids.

$$\{U_a(x,y,z)\} = [R_a]\{a\} + [\bar{A}_{as}]\{P_s\} \tag{2-8}$$

In other words, to create the spline transformation matrix, (2-6) is evaluated for point loads at the structural points to form basis vectors at the aerodynamic points that are the columns of the displacement transformation of (2-1).

Theoretical Development

Infinite Plate Spline and the Linear Spline

The theoretical developments for the Infinite Plate Spline and the Linear Spline are given in some detail (in the *MSC.Nastran Aeroelastic User's Guide*) and are not repeated here.

Thin Plate Spline

The thin plate spline (TPS) is a generalization of the IPS to three dimensions. The derivation is entirely analogous with the IPS with the addition of the third coordinate. The superimposed fundamental solutions (see Eq. 2-30 of Ref. 12) remain

$$w(x, y, z) = \sum[A_i + B_i r_i^2 + (P_i / 16\pi D) r_i^2 \ln r_i^2] \qquad (2\text{-}9)$$

but now $r_i^2 = (x - x_i)^2 + (y - y_i)^2 + (z - z_i)^2$. The boundary conditions at infinity now require the addition of the moment in the third axis:

$$\sum B_i = 0 \qquad (2\text{-}10)$$

$$\sum P_i = 0 \qquad (2\text{-}11)$$

$$\sum x_i P_i = 0 \qquad (2\text{-}12)$$

$$\sum y_i P_i = 0 \qquad (2\text{-}13)$$

$$\sum z_i P_i = 0 \qquad (2\text{-}14)$$

A solution to the general spline problem, formed by superimposing solutions of (2-9), is given by

$$w(x, y, z) = a_0 + a_1 x + a_2 y + a_3 z + \sum_{i=1}^{N} K_i(x,y,z) P_i \qquad (2\text{-}15)$$

$$K_i(x,y,z) = (1/16\pi D) r_i^2 \ln r_i^2$$

$$r_i^2 = (x - x_i)^2 + (y - y_i)^2 + (z - z_i)^2$$

and P_i is the concentrated load at (x_i, y_i, z_i)

The N+4 unknowns $(a_0, a_1, a_2, a_3, P_i; i = 1, N)$ are determined from the 4 equilibrium equations (2-11) through (2-14) and the N equations

(2-16)

$$w_j(x_j, y_j, z_j) = a_0 + a_1 x_j + a_2 y_j + a_3 z_j + \sum_{i=1}^{N} K_{i,j} P_i \; ; j = 1, N$$

(2-17)

$$K_{i,j} = K_i(x_j, y_j, z_j) = (1/16\pi D) r_{i,j}^{\;2} \ln r_{i,j}^{\;2}$$

(2-18)

$$r_{i,j}^{\;2} = (x_j - x_i)^2 + (y_j - y_i)^2 + (z_j - z_i)^2$$

The coefficient matrix can then be assembled that permits solution for the vector of $a_i$and $P_{i.}$ The interpolation to any point (x,y,z) is then achieved by evaluating $w(x,y,z)$ from (2-15), at the desired points. This gives an overall equation relating the **M** dependent aerodynamic points to a displacement pattern at the **N** independent structural points:

$$\{w\}_a = \begin{bmatrix} 1 & x_1 & y_1 & z_1 & K_{1,1} & K_{1,2} & \dots & K_{1,N} \\ 1 & x_2 & y_2 & z_2 & K_{2,1} & K_{2,2} & \dots & K_{2,N} \\ \dots & \dots & \dots & \dots & \dots & \dots & \dots & \dots \\ 1 & x_M & y_M & z_M & K_{M,1} & K_{M,2} & \dots & K_{M,N} \end{bmatrix} [C]^{-1} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ w_1 \\ w_2 \\ \dots \\ w_N \end{Bmatrix} = [K][C]^{-1} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ w_1 \\ w_2 \\ \dots \\ w_N \end{Bmatrix}$$

(2-19)

In the derivation, we have not indicated the meaning of the scalar P_i, but we can now take them to be a set of forces in one coordinate direction. In that case, provides an interpolation between forces in one direction and displacements in that direction with equilibrium preservation. We can apply that transform for each translational direction to build a three dimensional interpolant. With this observation, we now can assemble the local spline matrix that relates the aerodynamic degrees of freedom to the structural degrees of freedom for the DOF's that are participating in the particular application of the TPS:

$$\{u_a\}|_{tran} = \begin{Bmatrix} u_1 \\ \dots \\ u_M \\ v_1 \\ \dots \\ v_M \\ w_1 \\ \dots \\ w_M \end{Bmatrix}_a = [T^t{}_{ap}] \begin{bmatrix} [K]\left[C^{-1}\right] & 0 & 0 \\ 0 & [K]\left[C^{-1}\right] & 0 \\ 0 & 0 & [K]\left[C^{-1}\right] \end{bmatrix} \begin{bmatrix} P_I & 0 & 0 \\ 0 & P_I & 0 \\ 0 & 0 & P_I \end{bmatrix}$$

$$[T_{ps}]\{u_g\} = \left[G^t{}_{as}\right]\{u_g\}$$

(2-20)

$$\{u_a\}\big|_{rot} = \begin{Bmatrix} \theta_{x_1} \\ \dots \\ \theta_{x_M} \\ \theta_{y_1} \\ \dots \\ \theta_{y_M} \\ \theta_{z_1} \\ \dots \\ \theta_{z_M} \end{Bmatrix}_a = [T^r{}_{ap}] \begin{bmatrix} [\partial K][C^{-1}] & 0 & 0 \\ 0 & [\partial K][C^{-1}] & 0 \\ 0 & 0 & [\partial K][C^{-1}] \end{bmatrix} \begin{bmatrix} P_I & 0 & 0 \\ 0 & P_I & 0 \\ 0 & 0 & P_I \end{bmatrix}$$

$$[T_{ps}]\{u_g\} = [G^r{}_{as}]\{u_g\}$$

(2-21)

$$[P_I] = \begin{bmatrix} 0 & 0 & \dots & 0 \\ 0 & 0 & \dots & 0 \\ 0 & 0 & \dots & 0 \\ 1 & 0 & \dots & 0 \\ 0 & 1 & \dots & 0 \\ \dots & \dots & \dots & \dots \\ 0 & 0 & \dots & 1 \end{bmatrix}_{3+N \times N}$$

and $[T^t{}_{ap}]$,$[T^r{}_{ap}]$ are the three assembled 3x3 transformation matrices that relate the aerodynamic coordinate system to the spline coordinate system. Similarly, $[T_{ps}]$ is the assembled 3x3 transformation matrices that relate the structural displacements to the spline component directions. Just as in the 2-D case, the structural rotations are not involved in this spline formulation.

Most of the difference between this 3-D version and its 2-D counterpart is in the formation of the $[T^t{}_{ap}]$,$[T^r{}_{ap}]$, $[T_{ps}]$ matrices to account for separate treatment of each displacement component instead of a single planar interpolation surface.

Singularity Conditions: In the 2-D case, the spline interpolant is singular if the independent (structural) points are all collinear. In the 3-D case, the interpolant is singular if the independent points are all coplanar. If the TPS interpolant is found to be singular, the code automatically reverts to a 2-D interpolant with the spline plane defined by the plane of the coplanar structural grid points. In the case of a singular 2-D case, a fatal error is issued, since we do not attempt to revert to a beam spline.

Two Dimensional Finite Plate Spline

The finite plate spline (FPS) is a method that uses a mesh of elemental quadrilateral or triangular plates to compute the interpolation function. It is similar to the IPS and beam spline methods, in that the interpolant is based on structural behavior, but is different in that the equations are a discretized approximation of a finite structural component. A finite plate approximation has the advantage of being able to more closely approximate the boundary conditions at the edge of the interpolation region: boundary conditions at infinity are replaced by an FE (finite element) approximation of the independent degrees of freedom extrapolated to the plate edges. This methodology has been shown (see Ref. 2) to limit the "potato chip" effect observed in extrapolation using the IPS.

The FPS is complicated by the need to establish the virtual surface and, on that surface, a virtual mesh. For the 2D applications of Aerodynamic Panel Methods, the virtual mesh for the interpolation surface is simply the planar region defined by the CAEROi entry. A simple *n* x *m* mesh of points can be used to subdivide the region into finite elements. The geometries of these planes are such that each FE is almost certainly going to have acceptable geometry. Furthermore, since only normal forces are mapped, the structural points can be projected onto the virtual surface without any complications (noting that the aerodynamic points are on the interpolation surface).

Consider a planar trapezoidal surface that lies in a local *x-y* plane and that is divided into a series of finite elements as shown in Figure 2-1. We want to use this FE plate to interpolate between a set of *n* structural points and *m* aerodynamic points, which are not necessarily coincident with the N virtual mesh points.

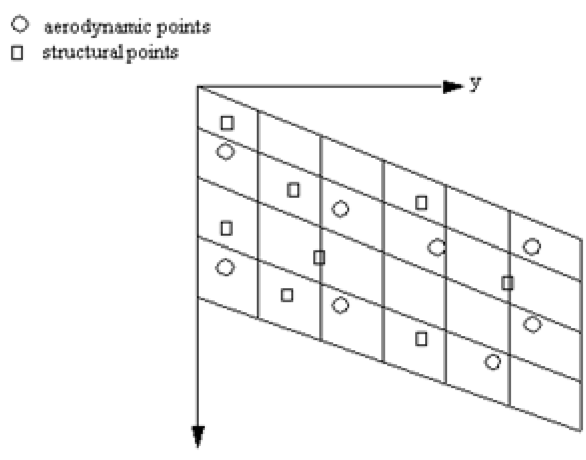


Figure 2-1 Two Dimensional Finite Surface Spline

Consider a 4-noded quadrilateral element in which the normal displacement, $w = w(x, y)$, and the rotations, $\theta = \theta(x, y)$, about the x axis and $\phi = \phi(x, y)$, about the y axis are given by:

$$\{r\} = \lceil \Omega \rfloor \{u^e\} \tag{2-22}$$

$$\{r\} = \begin{Bmatrix} w \\ \theta \\ \phi \end{Bmatrix}; \ \lceil \Omega \rfloor = \begin{Bmatrix} \lfloor \omega \rfloor \\ \lfloor \omega_x \rfloor \\ \lfloor \omega_y \rfloor \end{Bmatrix}; \ \{u^e\} = \lfloor w_1 \ \theta_1 \ \phi_1 \ldots \phi_4 \rfloor^T \tag{2-23}$$

and where the angles satisfy the relations

$$\theta = \frac{dw}{dy}; \quad \phi = \frac{dw}{dx} \tag{2-24}$$

The shape functions, $\lfloor \omega \rfloor$, are a *1x12* row matrix used to interpolate the displacement field within the element in terms of the nodal displacements, $\{u^e\}$. Experience documented in Ref. 2 suggests that a C^1continuous shape function is the preferred choice with the angular rotations given by (2-24) rather than by independent shape functions. For each element in the entire virtual FE mesh, a boolean connectivity matrix, $[B]$, is developed to relate the element nodal displacements to the overall FE mesh displacements:

$$\{u^e\}_i = \lceil B \rfloor_i \{u\} \tag{2-25}$$

Given Equations (2-25) and (2-22), the displacement at each of the structural points and each of the aerodynamic points may be related to the virtual mesh displacements as:

$$\lceil \psi_s \rfloor = \begin{bmatrix} \lceil \Omega \rfloor^s_1 \ \lceil B \rfloor_1 \\ \lceil \Omega \rfloor^s_2 \ \lceil B \rfloor_2 \\ \ldots \quad \ldots \\ \lceil \Omega \rfloor^s_n \ \lceil B \rfloor_n \end{bmatrix} \tag{2-26}$$

and

$$\lceil \psi_a \rfloor = \begin{bmatrix} \lceil \Omega \rfloor^a_1 \ \lceil B \rfloor_1 \\ \lceil \Omega \rfloor^a_2 \ \lceil B \rfloor_2 \\ \ldots \quad \ldots \\ \lceil \Omega \rfloor^a_m \ \lceil B \rfloor_m \end{bmatrix} \tag{2-27}$$

and the structural point displacements and aerodynamic point displacements can be expressed as functions of the virtual FE mesh displacements:

$$\{u_s\} = \lceil \psi_s \rfloor \{u\} \tag{2-28}$$

$$\{u_a\} = [\psi_a]\{u\} \tag{2-29}$$

Since the virtual surface described by the FE mesh is required to pass through the set of independent (structural) points, a penalty method can be used to express the equilibrium state of the virtual surface:

$$[K]\{u\} + [\alpha][\psi_s]^T([\psi_s]\{u\} - \{u_s\}) = 0 \tag{2-30}$$

where $[\alpha]$ represents an invertible, diagonal weighting matrix to scale the elements of $[K]$ and $[\psi_s]^T[\psi_s]$. Using (2-30) to solve for the virtual mesh displacements, $\{u\}$ yields:

$$\{u\} = \left\{[\alpha]^{-1}[K] - [\psi_s]^T[\psi_s]\right\}^{-1}[\psi_s]\{u_s\} = [A]^{-1}[\psi_s]\{u_s\} \tag{2-31}$$

and substituting into (2-29), the desired splining relationship can be found directly:

$$[G_{as}] = [\psi_a]\left\{[\alpha]^{-1}[K] + [\psi_s]^T[\psi_s]\right\}^{-1}[\psi_s]^T \tag{2-32}$$

The virtual surface stiffness properties are such that our requirements for equilibrium preservation are satisfied and virtual work principles allow us to use the transpose of (2-32) as the force transform. Spring attachments are available in this method by adding flexibilities to the diagonals of the matrix $[A]$ in (2-31). Three such flexibilities are possible: k_w, k_θ and k_ϕ. but a single value has been used for all three in the actual implementation.

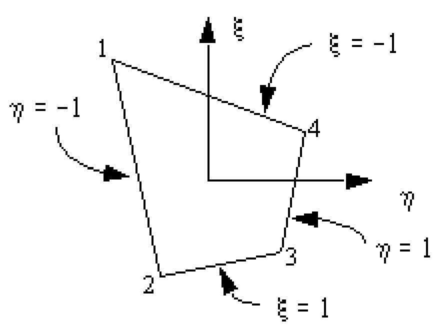


Figure 2-2 Element Coordinate System for FPS

Experience based on the implementation of Ref. 2, has shown that reasonable behavior for 2D aeroelastic applications is obtained if the virtual mesh is constructed on the plane of the CAEROi within the superscribing quadrilateral that contains all the aerodynamic points and the projected structural points. This represents an improvement over the IPS without requiring the complexity of the triangular degenerate case.

Spring Attachments and Interpolation Smoothing

Each of the splining methods lends itself to a smoothing parameter or parameters that can loosen the requirement that the interpolation "surface" pass through the structural points. This feature is useful in that, for certain selections of interpolating structural grid points, forcing the spline function to pass through all of the points causes oscillation in the interpolant (e.g., when a number of closely space grid points are attached to the spline function) or when the set of structural grid points overspecifies the behavior of the spline and causes a singularity (e.g., when more than two grids are attached to the same axial station of a beam spline or when both translations and rotations are applied at the same axial station of a beam spline).

Relaxing the equality criterion at a point can be thought of as a translational and/or torsional spring that attaches the structural point to the interpolation surface. Mathematically, this is presented in Ref. 2 and repeated here for completeness.

Each of the interpolation functions can be represented as:

$$\{U(x,y,z)\} = [R]\{a\} + [\bar{A}]\{P\} = [C][P] \tag{2-33}$$

where $\{U(x,y,z)\}$ represents the spline surface deflection under a set of point loads. If the structural points are connected to the spline surface by springs, the structural displacements will differ from the spline surface deformation by the deformation of the spring. The spring deformation results in forces:

$$\{P(x,y,z)\} = [K_s](\{U_g\} - \{U_a\}) \tag{2-34}$$

where the diagonal matrix of spring stiffnesses, $[K_s]$. These stiffnesses are nonzero (since, if they were zero, the structural grid point would be disconnected from the spline surface and would be omitted from the set), so an inverse exists. This results in a modification to the influence coefficient matrix in (2-33):

$$\{U(x,y,z)\} = [R]\{a\} + ([\bar{A}] + [K_s]^{-1})\{P\} \tag{2-35}$$

Thus, all that is required with any of these interpolation methods to introduce smoothing is to add a "spring flexibility" to the diagonals of the influence coefficient matrix.

Guidelines

The interpolation from the structural to aerodynamic degrees of freedom is based upon the theory of splines (Figure 2-3). High aspect ratio wings, bodies, or other.

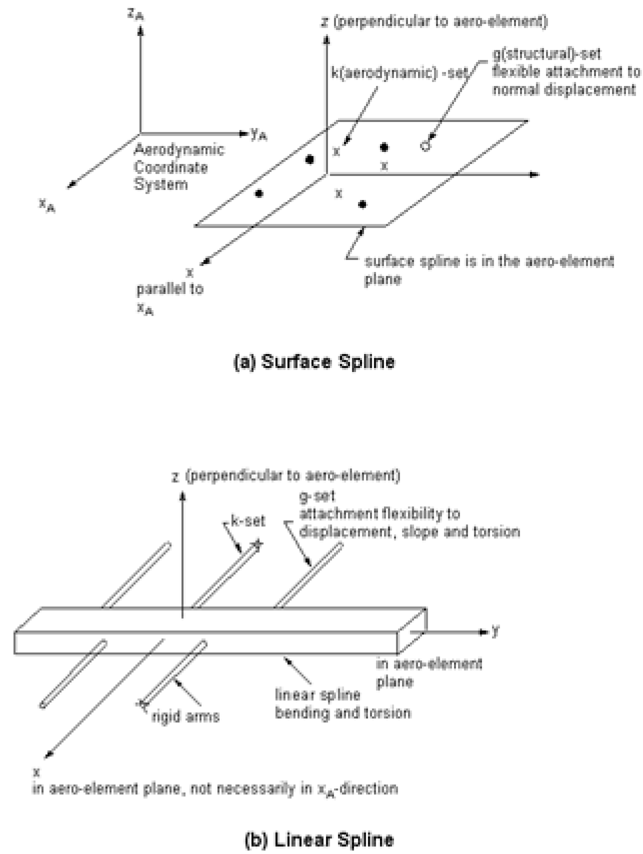


Figure 2-3 Splines and Their Coordinate Systems

beam-like structures should use linear splines (SPLINE2). Low aspect ratio wings, where the structural grid points are distributed over an area, should use surface splines (SPLINE1). Several splines can be used to interpolate to the boxes on a panel or elements on a body; however, each aerodynamic box or element can be referenced by only one spline. Any box or body element not referenced by a spline will be "fixed" and have no motion, and forces on these boxes or elements will not be applied to the structure. A linear relationship (like an MPC) may be specified for any aerodynamic point using the SPLINE3 entry. This is particularly useful for control surface rotations.

For all types of splines, the user must specify the structural degrees of freedom and the aerodynamic points involved. The degrees of freedom utilized at the grid points include only the normal displacements for surface splines. For linear splines, the normal displacement is always used and, by user option, torsional rotations and/or slopes may be included.

The SPLINE1 data entry defines a surface spline This can interpolate for any "rectangular" subarray of boxes on a panel. For example, one spline can be used for the inboard end of a panel and another for the outboard end. The interpolated aerodynamic degrees of freedom (k-set) are specified by naming the lowest and highest aerodynamic grid point numbers in the area to be splined. A parameter DZ is used to allow smoothing of the spline fit. If $DZ = 0$ (the recommended value), the spline will pass through all deflected grid points. If $DZ > 0$, then the spline (a plate) is attached to the deflected grid points via springs, which produce a smoother interpolation that does not necessarily pass through any of the points. The flexibility of the springs is proportional to DZ.

Three different methods of surface splines can be invoked with the METH field on the SPLINE1 entry: IPS, TPS and FPS (infinite, thin and finite plate spline, respectively). For the flat plate aerodynamics contained in FLDS, there is little benefit from selecting the TPS. If the TPS is selected and all the grids are coplanar, the algorithm automatically reverts to IPS in any case. The FPS is a recent addition to the arsenal of spline methods in MSC.Nastran and is believed to provide improved performance relative to the two infinite splines since it is better able to handle the deformation patterns at the edge of the aerodynamic panel. It is therefore recommended that the FPS be selected, but that the other two methods are available as a fallback if FPS performance is not satisfactory.

The USAGE flag on the SPLINE1 entry identifies whether the spline is to be used to transform forces, displacements or both (FORCE, DISP or BOTH). The BOTH option is often acceptable, but the other options provide the ability to tailor the splining to the application. Since the force transformation imposes loads on the structure, it is reasonable to select grid points that can withstand this loading without severe deformations. The displacement transformation requires an accurate representation of the overall deformation pattern, so it is quite conceivable that a different set of grids from the force transform would be the most appropriate.

Final inputs on the continuation of the SPLINE1 entry allow the user to define the mesh density in the chordwise and spanwise directions of the underlying aerodynamic panel. The default values should be adequate, but substituting large numbers provides added refinement in making the transformations.

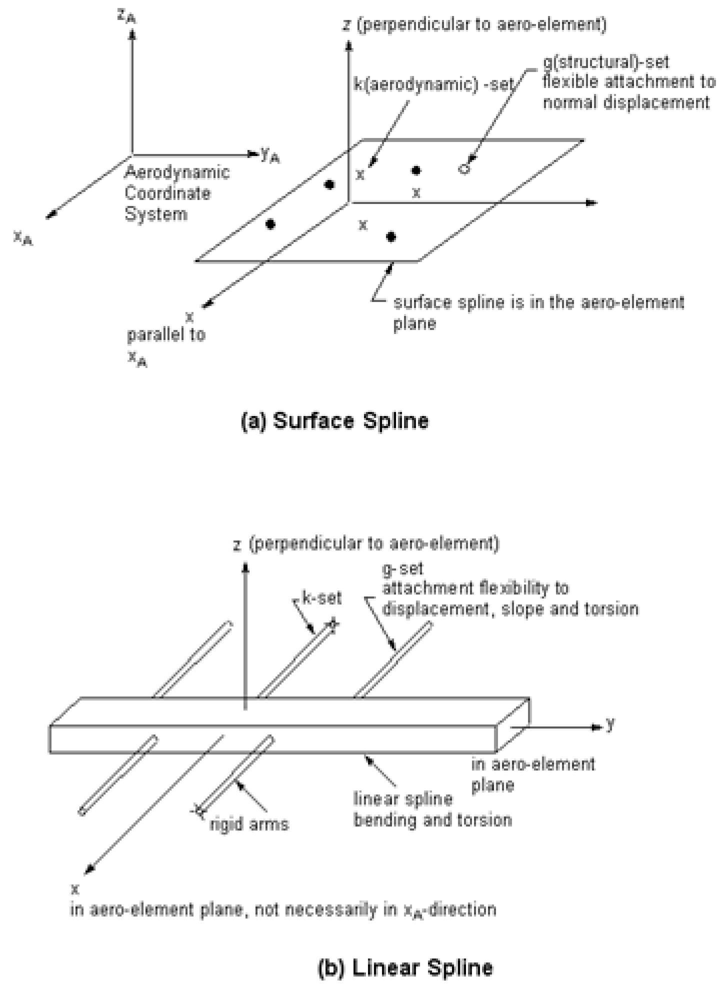


Figure 2-4 Splines and Their Coordinate Systems

The SPLINE2 data entry defines a linear spline. As can be seen from Figure 2-3, this is a generalization of a simple beam spline to allow for interpolation over an area. It corresponds to the frequently used assumption of the "elastic axis" in which the structure is assumed to twist about the axis such that the airfoil chord perpendicular to the axis behaves as if it were rigid. The portion of a panel to be interpolated and the set of structural points are determined in the similar manner as with SPLINE1. However, a coordinate system must also be supplied to determine the axis of the spline (which is the elastic axis of the virtual beam); a coordinate system with its Y-axis collinear with the spline axis is required. That coordinate system should be somewhere near the true elastic axis and approximately aligned with it (e.g., swept if the elastic axis is swept). Since the spline has torsion and bending flexibility, the user may specify the ratio DTOR of flexibilities for a wing as a representative value of $(EI)/(GJ)$; the default value for this ratio is 1.0. The attachment flexibilities, D_x, $d_{\theta x}$, and $D_{\theta y}$ allow for smoothing, but usually all values are taken to be zero; when the attachment flexibilities are taken to be zero, the spline passes through all of the connected grid points and the value of the ratio DTOR has no effect. In the case where the structural model does not have one or both slopes defined, the convention $DTHX = -1.0$ and/or $DTHY = -1.0$ is used. When used with bodies, there is no torsion and the spline axis is along the body so that a user input coordinate system is not required.

There are special cases with splines where attachment flexibility is either required or should not be used. The following special cases should be noted:

1. Two or more grid points, when projected onto the plane of the element (or the axis of a body), may have the same location. To avoid a singular interpolation matrix, a positive attachment flexibility must be used (or better yet, only one grid point selected at that location).
2. With linear splines, three deflections with the same spline Y-coordinate over determine the interpolated deflections since the perpendicular arms are rigid. A positive DZ is needed to make the interpolation matrix nonsingular.
3. With linear splines, two slopes (or twists) at the same Y-coordinate lead to a singular interpolation matrix. Use $DTHX > 0$ (or $DTHY > 0$) to allow interpolation.
4. For some modeling techniques, i.e., those which use only displacement degrees of freedom, the rotations of the structural model are constrained to zero to avoid matrix singularities. If a linear spline is used, the rotational constraints should not be enforced to these zero values. When used for panels, negative values of DTHX will disconnect the slope, and negative values of DTHY will disconnect the twist. For bodies, DTHY constrains the slopes since there is no twist degree of freedom for body interpolation.

The USAGE flag on the SPLINE2 entry performs the same function as on the SPLINE1.

The SPLINE4 bulk data entry is very similar to the SPLINE1 entry. The difference is that the SPLINE4 entry invokes a list of aerodynamic elements that are to be included in the spline while the SPLINE1 entry selects the first and last aerodynamic box to be splined and allows the spline algorithm to determine the intermediate box ID's.

The SPLINE5 bulk data entry is very similar to the SPLINE2 entry. The difference is the use of a list of aerodynamic elements rather than using "first box - last box" logic.

Spline Metrics

This final subsection for this appendix provides a short description of a resource that is available in MSC.Nastran for evaluating the quality of the splines. It entails activating special prints within the MSC.Nastran that provide summary information for each of the splines. Although this feature is not available directly from the FlightLoads GUI, it may be of utility for especially problematic cases and therefore is documented here.

The special prints are invoked by setting a system cell at the top of the MSC.Nastran input file that says either:

NASTRAN SPLINE_METRICS

This can be done from the FlightLoads GUI by using the Direct Text Input feature, and inserting the text into the File Management portion of the input.

With this input, summary information on the spline is created and printed in the .f06 output from the run. An example of this output is given in Figure 2-5.

S P L I N E   M E T R I C S

SPLINE	METHOD	AERO POINTS	STRUCT POINTS		MAXIMUM	AERO GRID POINT	STRUCTURAL GRID POINT	MINIMUM	AERO GRID POINT	STRUCTURAL GRID POINT
100	IPS	24	20	TRANSLATION COEFF	.90036	116	23	-.14039	107	36
				ROTATION COEFF	1.3039	112	36	-1.2725	117	6
				TRANSLATION SUM	1.0000	121		1.00000	122	

Figure 2-5 Spline Metrics Example

The output first lists the spline id and its type followed by the number of aero points and structural points that are being used in the spline. This is followed by two sets of tables that provide information on the maximum value and the minimum value for the translational and rotational coefficients in the spline matrix. It is difficult to assign physical meaning to the rotational coefficients, but the translational coefficients can be thought as a percentage of force that is being transformed for the aero/structural grid point listed. If the two sets of grids were identical in number and location, the max translation coefficient would be 1.0 and the min would be 0.0. If the meshes are not coincident, an estimate of the average coefficient value is given by the ratio of the number of aero points to the number of structural points. The maximum and minimum values would be somewhat above and below this number and judgment is required to determine if the values indicate problems with the spline. If the maximum value differs from the average value by over an order of magnitude, it may be beneficial to refine the spline in the area of the offending aero/structure pair. (Note that "refinement" may require removing one or more "nearly coincident" structural points -- a common cause of large couples.) A similar comment applies if the minimum value has an change of sign from the average and is greater in magnitude by over a factor of ten.

The matrix print shown above also prints the maximum and minimum sums of the spline matrix. Theoretically, each translation sum should be 1.0 and each rotational sum should be numerically 0.0 to indicate equilibrium. Recent experience has indicated that this is always the case for nonsingular splines.

C Results Interface via XDB

XDB Output

If PARAM,POST,0 is requested in SOL 144, a large number of new outputs become available. First, the aerodynamic mesh (for Doublet Lattice Slender Body geometries) is stored so that the unit solutions and trim solutions can be visualized in Patran using either the NASTRAN preference or FlightLoads. Without MSC.FLD, you must import both the model and the results. Using MSC.FLD, you will typically have the model already available in Patran. When the XDB is attached to PATRAN for both model and results data, the aerodynamic model will be imported and placed in a GROUP named "Aero Model CONFIG=<aeconfig>" where <aeconfig> is the AECONFIG name assigned in Case Control. The structural model is imported as before, except it will be placed in the group entitled "Aeroelastic Structural Model."

In the Results module of Patran, the trim subcase data and unit solution data are available to be visualized on both the structural mesh and the aerodynamic mesh. These data consist of both displacements and several force components. The data are divided into TRIM results (subcases labelled SC<*subcase index*>) and "Flexible Increments (labelled FI <*condition and parameter identifier*>)" that represent the unit solution results. Each of these "Result Cases" (in the PATRAN parlance) is further divided into several "Results Types" that are the rigid, elastic, inertial, and total loads and the displacements. For Flexible Increments, both restrained and unrestrained data are available. For TRIM subcases, only restrained data are used to create the linear combination that is the TRIM result.

The Result Case labels are formed using the following rules:

- SC<*subcase index*><*config*>
  - <*subcase index*> is the subcase index, e.g., SC1 or SC2
  - <*config*> is the AECONFIG name if the result case applies to the aerodynamic mesh and is the word "Structure" if it applies to the structural model.
- FI: <*config*> M=<*mach*>, Sxy=<*sym*>, DC=<*controller*>, Q=<*q*>, Sxz=<*sym*>
  - <*config*> is the AECONFIG name from Case Control if the result case applies to the aerodynamic mesh and is the word "Structure" if it applies to the structural mesh
  - <*mach*> is the Mach number of the aerodynamics used in the unit solution
  - <*sym*> is the XY or XZ symmetry value of the aerodynamic model used in the unit solution
  - <*q*> is the dynamic pressure value of the unit solution
  - <*controller*> is the name (AESTAT/AESURF) of the controller that was perturbed to produce the unit solution.

All the data results are available on both meshes except for the inertial loads and the static applied loads, which are defined only on the structural mesh. Since these forces arise on the structural mesh, and the spline relationships don't allow us to move forces from the structure to the aerodynamic nodes, these data are not defined over the aerodynamic model. Note, however, that the ***elastic increment aeroelastic forces*** associated with the deformations ***caused*** by the inertial forces are defined on both meshes. These data are part of the "elastic" component of the forces.

Table 3-1 shows the Result Types for the Trim Cases and Table 3-2 shows the Results Types for the Flexible Increments Cases. Note that, for Doublet Lattice models, the results are defined at the centroidal "grid" points, which, for display purposes in Patran become "elemental" results defined at the center of each aerodynamic box. Hence, the labels further hint that the results are defined on the aerodynamic mesh by labelling the results as “elemental” or “nodal.” In fact, “nodal” aerodynamic results are supported, but cannot be generated by DLM.

Table 3-1 Result Types for Trim Cases

Mesh	Description	Label
Aero	Rigid Aerodynamic Forces at trim	Aeroelastic Forces, Elemental Rigid Component Aeroelastic Moments, Elemental Rigid Component
Aero	The elastic increment forces at trim due to all controllers. (Including accelerations.)	Aeroelastic Restrained Forces, Elemental Elastic Component Aeroelastic Restrained Moments, Elemental Elastic Component
Aero	The trimmed deformation mapped to the aerodynamic corner point grids.	Displacements, Translational Displacements, Rotational
Structure	Rigid Aerodynamic Forces at trim	Aeroelastic Forces, Nodal Rigid Component Aeroelastic Moments, Nodal Rigid Component
Structure	Elastic Increment Forces at trim due to all controllers including accelerations	Aeroelastic Restrained Forces, Nodal Elastic Component Aeroelastic Restrained Moment, Nodal Elastic Component
Structure	Inertial Forces due to trimmed acceleration	Aeroelastic Restrained Forces, Nodal Inertial Component Aeroelastic Restrained Moments, Nodal Inertial Component
Structure	The trimmed deformations	Displacements, Translational Displacements, Rotational
Structure	The total applied load including static applied loads, elastic corrections and inertial loads. (Requires OLOAD=<n,ALL>)	Applies Load, Translational Applied Load, Rotational

Table 3-2 Result Types for Flexible Increment Cases

Mesh	Description	Label
Aero	Rigid Aerodynamic forces due to perturbation of controller at specified Mach & Dynamic pressure	Aero Unit Forces, Elemental Rigid Component Aero Unit Moments, Elemental Rigid Component
Aero	The elastic increment forces due to the restrained deformations caused by unit perturbation at M, q, symmetry	Aero Unit Restrained Forces, Elemental Elastic Component Aero Unit Restrained Moments, Elemental Elastic Component
Aero	The elastic increment forces due to the unrestrained deformations caused by unit perturbation at M, $\bar{q}$, symmetry	Aero Unit Unrestrained Forces, Elemental Elastic Component Aero Unit Unrestrained Moments, Elemental Elastic Component
Aero	The restrained or unrestrained deformation due to unit perturbation at M, $\bar{q}$, symmetry	Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Translational Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Rotational
Structure	Rigid Aerodynamic forces due to perturbation of controller at specified M, $\bar{q}$, symmetry	Aero Unit Forces, Nodal Rigid Component Aero Unit Moments, Nodal Rigid Component
Structure	The elastic increment forces due to the restrained or unrestrained deformations caused by unit perturbation of controller at M, $\bar{q}$, symmetry	Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Translational Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Rotational
Structure	The inertial forces arising due to the unit perturbation of controller at M, $\bar{q}$, symmetry	Aero Unit Forces, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Inertial Component Aero Unit Moments, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Inertial Component
Structure	The restrained or unrestrained deformation due to unit perturbation of controller at M, $\bar{q}$, symmetry	Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Translational Aero Unit Displacements, $\left\{\begin{matrix}\text{Restrained}\\\text{Unrestrained}\end{matrix}\right\}$, Rotational

D Aerodynamic and Aeroelastic Databases

Aerodynamic and Aeroelastic Databases

While MSC.Nastran has a very sophisticated automated restart capability, in aeroelastic analysis, it is often the case that the aerodynamic data are updated relatively infrequently relative to the structural data. Also, once the computationally intensive unit solutions are obtained, they can be reused in TRIM analysis very rapidly (nearly interactively) to generate distributed trimmed loads.

While MSC.Nastran's NDDL, FMS and DMAP have always allowed the reuse of these data, it was not a trivial matter to define the collection of datablocks that would constitute the minimal archival set for reuse that also had the appropriate functionality (supply all the requisite pieces). Further, to reuse the data blocks (as opposed to automated restart), the DMAP needs to "look" for the data blocks and, if already present, explicitly branch around the "creation" step.

In MSC.Nastran V70.7 (as part of the FlightLoads development), the NDDL and DMAPs associated with SOL 144 were updated to create two collections of data: an aerodynamic database and an aeroelastic database. These collections are implemented in such a way as to simplify the user input requirements, remain compatible with automated restart and allow data "reuse" using the DBLOCATE technique.

The modifications made are primarily in DMAP and, in particular, NDDL. A hierarchy of aerodynamic and aeroelastic paths was defined in NDDL that allows the user to archive and reuse the aerodynamic and aeroelastic data (somewhat independently). This is accomplished by created two new LOCATION values in the NDDL (and DMAP): the ADB and the AEDB. Aerodynamic data (completely independent of structural data) is associated with the ADB and aeroelastic data (coupling a particular aerodynamic model with the current structure and/or structural boundary condition) is associated with the AEDB. To direct the aerodynamic data onto a particular DBSET, one simply adds the appropriate INIT and ASSIGN statements and includes in bulk data the definition of the location parameters (PARAM,ADB,*logical_adb_name*).

Paths

A hierarchy of paths has been established for aerodynamic and aeroelastic data. These paths start at a minimal set at the top level and expand in scope as we move lower in the hierarchy. The top level is "geometry," recognizing that the user has ultimate control and may choose any relationship among disparate instances at this level. In other words, two distinct "geometries" may be the same vehicle with differing mesh topologies or it may be completely unrelated data or even duplicate data.

While the NDDL language cannot support PATHs that include other PATHs, the PATHs will be shown as additions to or unions with other PATHs to facilitate understanding of the hierarchy. In NDDL, they are, of necessity, expanded.

For aerodynamic data, the basic path is:

```
path aegeom     aeconfig,symxy,symxz,APRCH,HIGHQUAL $
path aegeomf    modltype + aegeom $
```

representing a (centroid and corner point) mesh topology. Notice that symmetry is included, meaning the mesh is reinstantiated for each symmetry option. This redundancy eliminates a layer in the hierarchy for almost no cost (the number and size of data blocks that would be a function only of AECONFIG is very small) and so has been adopted.

Geometry and Mach dependence:

```
PATH aegm        IMACHNO + aegeom $
```

Geometry, Mach and Reduced Frequency:

```
PATH aegmk       ikbar + aegm $
```

These paths handle all the steady and unsteady datablocks that comprise the ADB. These datablocks are a function only of the aerodynamics. No structural data are involved with one exception that has not been addressed:

Note:	MSC.Nastran aeroelastic analyses all assume that the basic coordinate system of the structure and that of the aerodynamic model are the same. This is in contrast to superelements, in which each superelement has its own basic system that is then related to the overall solution basic at the time the model components are assembled. This limitation has not yet been addressed.

The **aeroelastic** paths are essentially the union of the aerodynamic paths with the appropriate structural path. As a consequence, there are many more aeroelastic paths, but most are dealing with structural sets:

```
PATH aepeid      PEID,AUXMID  + aegeom $
PATH aeaset      a-set + aegeom $
PATH aelset      l-set + aegeom $
```

and a similar set that includes Mach number and/or dynamic pressure:

```
PATH aegsetq     g-set + aegm + iq, iqr $
PATH aeasetq     a-set + aegm + iq, iqr $
PATH aelsetq     l-set + aegm + iq, iqr $
PATH aelsetm     l-set + aegm $
```

and, finally, for unsteady aeroelastic analysis in modal coordinates, the generalized forces must be qualified by the modal reduction qualifiers and the aerodynamic paths:

```
PATH aegenf      aegmk + a-set + modal method + TFL/damping/etc. $
PATH aegenfl     aegeom + generalized boundary condition $
PATH aegenfs     aegeom + superelement generalized boundary $
```

Aerodynamic Database

The aerodynamic database consists of the following data blocks and paths. All the datablocks use the ADB location parameter.

Data Block	Path
ACPT	AEGEOM
AECOMP	AEGEOM
AECSTM	AEGEOM
AERO	AEGEOM
AMSPLINE	AEGEOM

Data Block	Path
CONTROL	AEGEOM
D1JK	AEGEOM
DJX	AEGEOM
FAJE	AEGEOM
HMKT	AEGEOM
TRX	AEGEOM
WGJ	AEGEOM
SRKT	AEGEOM
GDKSK	AEGEOM
GPKSK	AEGEOM
AEBGPDTS	AEGEOMF
AEBOXS	AEGEOMF
AEUSETS	AEGEOMF
D2JK	AEGM
LAJJ	AEGM
QKINTER	AEGM
QKKS	AEGM
QKPRESS	AEGM
QKX	AEGM
UAJJ	AEGM
AJJ0	AEGMK
AJJT	AEGMK
LAJJT	AEGMK
UAJJT	AEGMK
DJK	AEGMK
SKJ	AEGMK

Note: The aerodynamic data are archived without the weighting and correction matrix WKK. These weighting data need to be included in the bulk data on each reuse.

Aeroelastic Database

The aeroelastic database then consists of the following datablocks and paths. All these datablocks use the AEDB location:

Data Block	Path	Data Block	Path	Data Block	Path
GDKA	AEASET				
AEQGDKA	AEASETQ	AEQGDKL	AELSETQ	GPGK	AEPEID
KAAX	AEASETQ	URLR	AELSETQ	GDGK	AEPEID
UUXAX	AELSETQ	ERHM	AELSETQ	SPLINE	AEPEID
UXAX	AELSETQ	EUHM	AELSETQ	PGR	MLL
GPGK0	AEGEOM	HP	AELSETQ		
GDGK0	AEGEOM	HP0	AELSETQ		
PERGX	AELSETQ	IUXLR	AELSETQ		
PERKSX	AELSETQ	KRZX	AELSETQ		
PEUGX	AELSETQ	KSAZX	AELSETQ		
PEUKSX	AELSETQ	RHMCF	AELSETQ		
PIRGX	AELSETQ	RSTAB	AELSETQ		
PIUGX	AELSETQ	RUXLX	AELSETQ		
PRGX	AELSETQ	Z1ZX	AELSETQ		
PRKSX	AELSETQ	ZZX	AELSETQ		
UERKSX	AELSETQ				
UERGX	AELSETQ				
UEUKSX	AELSETQ				
UEUGX	AELSETQ				

Note:	Of necessity, the aeroelastic data includes the weighting matrix corrections.

Inputs and Outputs

To make effective use of the aerodynamic and/or aeroelastic database for reuse, you must have some familiarity with the FMS statements of MSC.Nastran. No new functionality has been introduced, it is merely that a coherent collection of paths and location parameters have been defined to allow basic FMS statements to redirect whole, consistent collections of data to particular DBSETs for reuse. Restart does not require nor is it affected by these modifications.

In particular, you should acquaint yourself with the INIT and ASSIGN statements in the *MSC.Nastran Quick Reference Guide, and some familiarity with the concepts in NDDL would be helpful.*

In the Bulk Data Section, you must define the ADB and AEDB location PARAMETERs (which are just PARAMs, like any other) to have the logical name of the DBSET to which you want the aerodynamic and aeroelastic (respectively) datablocks to be placed. These PARAMs are only needed on the job submittal(s) that create(s) the database.

To create both an aerodynamic and aeroelastic database, for example:

```
INIT, MASTER, LOGICAL=(MASTER(1000MB))
ASSIGN MASTER='adb144.master'
INIT, DBALL, LOGICAL=(DBALL1(1000MB))
ASSIGN DBALL1='adb144.dball'
INIT, myADB, LOGICAL=(myADB(1000MB))
ASSIGN myADB='adb144.adb'
INIT, myAEDB, LOGICAL=(myAEDB(1000MB))
ASSIGN myAEDB='adb144.aedb'
...
BEGIN BULK
param,adb,myadb
param,aedb,myaedb
...
```

In the case of multiple *instances* (an instance is a single data block that is associated with a particular set of qualifiers) of the datablocks on the aerodynamic or aeroelastic database, **all** of them will be located onto the DBSETs associated with the ADB and AEDB parameters.

To reuse an existing database, you need only ASSIGN the MASTER file and DBLOCATE all data blocks and parameters

```
ASSIGN AeroDB='adb144.master'
dblocate where (dbset='myadb' or dbset='myaedb') logical=aerodb
```

In the case of reuse (using DBLOCATE), the DBALL component of the initial database can be deleted (it then appears "offline" to the reuse, but there are no components on DBALL that are required for the reuse). The MASTER and ADB must be saved and DBLOCATEd to reuse the aerodynamic model. The MASTER, ADB and AEDB must be saved and DBLOCATEd to reuse the aeroelastic data.

Note:	For MSC.Nastran to "find" the data blocks on the database, the set of qualifiers (the "path") of the required data blocks in the reuse job must EXACTLY match those on the archived database. Be careful that the AECONFIG name (the only completely arbitrary value in the path) is the same. Also, if the Mach or Dynamic pressure is not found, new computations will occur without warning. This is really a feature since this is usually what you want to do: reuse the existing aerodynamic geometry and compute new rigid data. Further, you can "append" to the database by using the PARAM, ADB or PARAM, AEDB in conjunction with the DBLOCATE to create a new MASTER that includes both the original database and the new datablocks. This extension can be continued indefinitely.

Guidelines and Limitations

Be aware that the aerodynamic datablocks (in particular QKK and AJJ matrices) are fully dense (subsonic) and unsymmetrical. Consequently, the DBSETs of the aerodynamic and aeroelastic database may become very large. Archiving data for reuse is effective only when the tradeoffs between computational expense and data storage requirements are such that you want to store the data. These data are both large and computationally intensive to create, so typically archiving for reuse only makes sense if you **know** you intend to reuse the data.

In MSC.Nastran V70.7, the aeroelastic database is only defined for the static aeroelastic problem. The aerodynamic database, on the other hand, is defined for both the steady and unsteady data for all aerodynamic theories.

Examples

A number of examples of both the aerodynamic database and aeroelastic database are included in the TPL. Because they are database creation and reuse examples, the order in which you execute the samples is important. In the **procs** directory of the MSC.Nastran delivery, there is a procedure **aero_db.com** that can be used to execute the tests in the appropriate order and performing the appropriate clean up between multiple executions. The following bullets list the samples in order of intended execution and describe the feature illustrated by the example:

1. **adb144_1**: Perform a static aeroelastic analysis and save the ADB and AEDB components for reuse.
2. **adb144r1**: Replicate **adb144_1**, reusing the ADB and AEDB components. The original aerodynamic model and spline (aeroelastic) model is described in bulk data, but these data are not used in the run.
3. **adb144r2**: Replicate **adb144_1**, reusing the ADB and AEDB components but, this input deck doesn't include any aerodynamic model. The entire aeroelastic model is retrieved from the archived database. (This is the case in **adb144r1** also, but the original bulk data was left in place to show that its presence doesn't affect the run.)
4. **adb144r3**: Replicate **adb144_1**, reusing the ADB and AEDB components but neither the aerodynamic model nor the spline data are present in the input stream. This illustrates that the full aeroelastic model is retrieved from the ADB/AEDB and the bulk data is not used.

5. **adb144_2**: Create a new aerodynamic and aeroelastic data base by attaching the original from adb144_1, but utilizing a new configuration name and/or new Mach number/dynamic pressures, "append" new data to the database. The MASTER from this run can subsequently be attached and ADB, AEDB datablocks DBLOCATEd and both models will be available for reuse.
6. **dumpadb**: This illustrates a mini-DMAP solution that attaches an aerodynamic database and dumps rigid aerodynamic data to the XDB that allows visualization of the aerodynamic data. No data on the structural mesh is dumped, because the structural geometry/connectivity is not on the ADB or AEDB components, so XDB cannot associate the data to grids/elements.
7. **dumpaedb**: Similar to dumpadb, but includes the aeroelastic unit solution data. In this case, the aeroelastic data on the aerodynamic mesh are dumped to the XDB (in addition to the rigid aerodynamics). No data on the structural mesh is dumped, because the structural geometry/connectivity is not on the ADB or AEDB components, so XDB cannot associate the data to grids/elements.
8. **adb145_1**: This deck generates an unsteady aerodynamic database and also performs flutter analysis at a single Mach and altitude.
9. **adb145r1**: This deck reuses the unsteady aerodynamic database of adb145_1 to perform a complete Mach and altitude sweep for the archived Mach numbers.

E Static Aeroelastic Analysis

Introduction

Panel Aerodynamics (App. A) has discussed the aerodynamic methods contained in FlightLoads while Splines (App. B) has detailed how the aerodynamic and structural models are connected. This appendix completes the description of the computationally intensive portion of MSC.FlightLoads and Dynamics by documenting how these components are used in a static aeroelastic analysis.

As depicted in Aeroelasticity, the static aeroelastic analysis consists of three methods:

- Flexible Increments
- Flexible Trim
- Rigid Trim

After setting up the governing matrix equations, this appendix provides a discussion of these three methods. In this discussion, the term "unit solutions" is used interchangeably with the "Flexible Increments" method.

A very useful by-product of the aeroelastic analysis is the production of stability derivative and hinge moment data. This information is created within MSC.Nastran and is printed within the output (`.f06`) file. Although the information is not available as part of FlightLoads postprocessing, its value motivates the documentation of it here. (The dimensional equivalents can be obtained from the Load Summation tool of the Loads Browser).

Aeroelastic Matrices

MSC.Nastran analysis is based on placing degrees of freedom in sets and, for the purposes of this discussion, the aeroelastic equation can be developed in the f-set. This is the set of equations for the structural model that remains after single and multipoint constraints have been removed. The aerodynamics are splined to these points to produce an overall equation of the form.

$$\begin{bmatrix} \overline{M_{aa}} & M_{ao} \\ M_{oa} & M_{oo} \end{bmatrix} \begin{Bmatrix} \ddot{u}_a \\ \ddot{u}_o \end{Bmatrix} + \begin{bmatrix} \overline{K_{aa}} & K_{ao} \\ K_{oa} & K_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} - \bar{q} \begin{bmatrix} \overline{Q_{aa}} & Q_{ao} \\ Q_{oa} & Q_{oo} \end{bmatrix} \begin{Bmatrix} u_a \\ u_o \end{Bmatrix} = \bar{q} \begin{bmatrix} \overline{Q_{ax}} \\ Q_{ox} \end{bmatrix} \{ u_x \} + \begin{Bmatrix} \overline{P_a} \\ P_o \end{Bmatrix} \tag{5-1}$$

The f-set has been divided into an "analysis" set indicated by the "a" subscript and an omit set with the "o" subscript. The M, K, Q, and P matrices refer to Mass, Stiffness, aerodynamic, and applied loads, respectively while the u terms refer to displacements. The (ŽŽ) superscript indicates acceleration and the x subscript on u refers to the aerodynamic extra-points that have been introduced in (1-2). It should be noted that in FlightLoads, the first term of the u_x vector is the intercept; that is, in addition to the control surfaces and rigid body motions of Analysis, there is a term that represents zero angle of attack forces that could be due to user input twist and/or can be due to user input pressure forces on the aerodynamic elements.

There are two aerodynamic matrices in (5-1), and these are derived from splining the two aerodynamic matrices of (1-4) and (5-5) to the f-set degrees of freedom:

$$[Q_{ff}] = [G^p{}_{fk}][Q_{kk}][G^d{}_{kf}] \tag{5-2}$$

$$[Q_{fx}] = [G^{p}{}_{fk}][Q_{kx}] \tag{5-3}$$

Q_{ff} is the aerodynamic correction matrix in that it provides the forces that are produced by the structural deflections. Q_{fx} is the aerodynamic load matrix and provides the distributed load increment for each of the aerodynamic extra-points.

The spline matrices of (5-2) and (5-3) contain a p superscript to denote the force transform or a d superscript to denote a force transform.

As a final note on (5-1), it is noted that splining to omitted (o-set) degrees of freedom is not allowed so that $Q_{ao} = Q_{oa} = Q_{oo} = Q_{ox} = 0$. (5-1) can now be reduced to the a-set to give:

$$[M_{aa}]\{\ddot{u}_a\} + [K_{aa}]\{u_a\} - \bar{q}[Q_{aa}]\{u_a\} = \bar{q}[Q_{ax}]\{u_x\} + \{P_a\} \tag{5-4}$$

$$[M_{aa}] = [\overline{M_{aa}}] + [M_{ao}][G_{oa}] + [G_{oa}]^T[M_{oa}] + [G_{oa}]^T[M_{oo}][G_{oa}] \tag{5-5}$$

$$[K_{aa}] = [\overline{K_{aa}}] + [K_{ao}][G_{oa}] \tag{5-6}$$

$$[G^{x}{}_{ak}] = [\overline{G^{x}{}_{ak}}] + [G_{oa}]^T[G^{x}{}_{ok}] \tag{5-7}$$

$$[G^{x}{}_{ka}] = [\overline{G^{x}{}_{ka}}] + [G^{x}{}_{ko}][G_{oa}] \tag{5-8}$$

where x can be either "p" or "d" for the force or displacement spline, respectively.

$$[Q_{aa}] = [G^{p}{}_{ka}][S_{kj}][A_{jj}]^{-1}[D_{jk}][G^{d}{}_{ka}] \tag{5-9}$$

$$\{P_a\} = \{\overline{P_a}\} + [G_{oa}]^T\{P_o\} \tag{5-10}$$

$$[Q_{ax}] = [G^{p}{}_{ka}][Q_{kx}] \tag{5-11}$$

The structural displacements are recovered using standard recovery procedures; noting that

$$\{u^{o}{}_{o}\} = [K_{oo}]^{-1}[G^{p}_{ok}][Q_{kx}]\{u_x\} + [K_{oo}]^{-1}\{P_o\} \tag{5-12}$$

In the typical case for which there are free structural accelerations, divides the a-set into a r-set and an l-set. The r-set (reference set) contains user defined degrees of freedom equal in number to the number of rigid body motions permitted for the vehicle while the l-set (leftover set) contains the remaining degrees of freedom. There is a relationship between the aerodynamic extra points and the accelerations in the r-set that can be written as:

$$\{\ddot{u}_r\} = [T_{rR}][T_{Rx}]\{u_x\} \tag{5-13}$$

where the subscript R denotes the aerodynamic reference coordinate system and the $[T_{Rx}]$ matrix is a Boolean matrix that selects the aerodynamic reference point accelerations from the vector of trim parameters.

Under quasi-static analysis, the total free accelerations can be related to the rigid body accelerations as:

$$\{\ddot{u}_l\} = \left[D_{lr}\right]\{\ddot{u}_r\} \tag{5-14}$$

where the $[D_{lr}]$ matrix is a transformation based on the geometry of the structural model, but which can be derived from partitions of the stiffness matrix:

$$\left[D_{lr}\right] = \left[K_{ll}\right]^{-1}\left[K_{lr}\right] \tag{5-15}$$

Aeroelastic Solutions

The equations of the preceding section can now be solved once we have introduced two new concepts. The first concept is "restrained" vs. "unrestrained" and refers to how the displacements of supported degrees of freedom (u_r in the previous section) are treated. The Restrained approach sets these terms to zero and then solves for the remaining variables. This provides considerable simplification in the equations to be solved and is equivalent to the type of analysis that would be performed in a flight simulator. The Unrestrained approach applies a mean axis constraint (see Unrestrained Analysis below) to solve for u_r that is orthogonal to the rigid body motions of the vehicle. This corresponds to the equations of a free-flying vehicle and is therefore equivalent to the type of analysis that would be performed in a flight test. MSC.Nastran uses the restrained formulation for the majority of its calculations, but also does unrestrained calculations for the computation of stability derivatives and flexible load increments.

The second concept is that of Unit Loads and Solutions. The static aeroelastic capability implemented in the first release of FlightLoads is linear so that the total solutions can be regarded as the summation of unit solutions that are obtained from applying a unit value to a particular aerodynamic extra point (component of the u_x vector.). FlightLoads exploits this property in arriving at the overall solutions, in creating data through the Flexible Increment Method of Aeroelasticity and in subsequently presenting the data to the user through Results Browser. This section makes a distinction between Unit Loads/Solutions that are due to inertia loads and those that are due to other rigid body motions, such as angle of attack or control surface motion. The GUI of main body of this report does not distinguish between these two types of loads, but does distinguish between flexible, rigid and total loads (where "rigid" is the aerodynamic load applied to the structure, "flexible" is the incremental load due to the aeroelastic deformation and "total" is the sum of the two).

Restrained Analysis (Displacements Relative to $\{u_r\} = 0$)

We can rewrite (5-4) to enable solution of the free-free equations by combining (5-4) with (5-13) and using (5-14) to relate the (rigid body) l-set accelerations to the r-set accelerations (under the usual assumption of quasi-static analysis) to obtain:

$$\begin{bmatrix} K_{ll} & K_{lr} \\ K_{rl} & K_{rr} \end{bmatrix} \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} - \bar{q} \begin{bmatrix} Q_{ll} & Q_{lr} \\ Q_{rl} & Q_{rr} \end{bmatrix} \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} = \left\{ \bar{q} \begin{bmatrix} Q_{lx} \\ Q_{rx} \end{bmatrix} - \begin{bmatrix} M_{ll} & M_{lr} \\ M_{rl} & M_{rr} \end{bmatrix} \begin{bmatrix} D_{lr} \\ I_{rr} \end{bmatrix} \begin{bmatrix} T_{rR} \end{bmatrix} \begin{bmatrix} T_{Rx} \end{bmatrix} \right\} \{u_x\} + \begin{Bmatrix} P_l \\ P_r \end{Bmatrix} \tag{5-16}$$

Following the derivation of Eq. 2-69 of Ref. 12, we premultiply the first equation by $\begin{bmatrix} D_{lr} \end{bmatrix}^T$ and add it to the second to obtain:

$$\begin{bmatrix} K_{ll} & K_{lr} \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} - \bar{q} \begin{bmatrix} Q_{ll} & Q_{lr} \\ Q^z{}_{rl} & Q^z{}_{rr} \end{bmatrix} \begin{Bmatrix} u_l \\ u_r \end{Bmatrix} = \left\{ \bar{q} \begin{bmatrix} Q_{lx} \\ Q^z{}_{rx} \end{bmatrix} - \begin{bmatrix} M_{ll} D_{lr} + M_{lr} \\ m_r \end{bmatrix} \begin{bmatrix} T_{rR} \end{bmatrix} \begin{bmatrix} T_{Rx} \end{bmatrix} \right\} \{u_x\} + \begin{Bmatrix} P_l \\ P^z{}_r \end{Bmatrix} \tag{5-17}$$

in which the superscript “z” has been used to denote the r-set reductions:

$$-\bar{q}\begin{bmatrix} Q^z{}_{rl} \end{bmatrix} = -\bar{q}(D^T{}_{lr} Q_{ll} + Q_{rl}) \equiv [KAZL] \tag{5-18}$$

$$-\bar{q}\begin{bmatrix} Q^z{}_{rr} \end{bmatrix} = -\bar{q}(D^T{}_{lr} Q_{lr} + Q_{rr}) \equiv [KAZR] \tag{5-19}$$

$$-\bar{q}\begin{bmatrix} Q^z{}_{rx} \end{bmatrix} = -\bar{q}(D^T{}_{lr} Q_{lx} + Q_{rx}) \equiv [KSAZX] \tag{5-20}$$

$$\begin{bmatrix} P^z{}_r \end{bmatrix} = D^T{}_{lr} P_l + P_r \tag{5-21}$$

we have also applied the relationships that

$$D^T{}_{lr} K_{ll} + K_{rl} = D^T{}_{lr} K_{lr} + K_{rr} \equiv 0$$

by the definition of the rigid body transformation matrix and:

$$m_r \equiv D^T{}_{lr} M_{ll} D_{lr} + D^T{}_{lr} M_{lr} + D_{lr} M_{rl} + M_{rr}$$

This latter term is the definition of the "reduced mass matrix." Solving the first equation for $\{u_l\}$ in terms of $\{u_x\}$ (noting that $\{u_r\} \equiv 0$):

$$\{u_l\} = \left\{ \begin{bmatrix} K_{ll} \end{bmatrix} - \bar{q} \begin{bmatrix} Q_{ll} \end{bmatrix} \right\}^{-1} \left[\left\{ \bar{q} \begin{bmatrix} Q_{lx} \end{bmatrix} - \left(\begin{bmatrix} M_{ll} \end{bmatrix} \begin{bmatrix} D_{lr} \end{bmatrix} + \begin{bmatrix} M_{lr} \end{bmatrix} \right) \begin{bmatrix} T_{rR} \end{bmatrix} \begin{bmatrix} T_{Rx} \end{bmatrix} \right\} \{u_x\} + \{P_l\} \right] \tag{5-22}$$

Looking at (5-22), we can rewrite it in terms of three component “unit solutions”:

$$\{u^x{}_{lx}\}^r = \left\{ \begin{bmatrix} K_{ll} \end{bmatrix} - \bar{q} \begin{bmatrix} Q_{ll} \end{bmatrix} \right\}^{-1} \left\{ \bar{q} \begin{bmatrix} Q_{lx} \end{bmatrix} \right\} \equiv [RUXLX] \tag{5-23}$$

$$\{u^x{}_{lr}\}^i = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\left\{\left(\left[M_{ll}\right]\left[D_{lr}\right] + \left[M_{lr}\right]\right)\right\} \equiv [IUXLR] \quad (5\text{-}24)$$

$$\{u^x{}_l\}^p = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\{P_l\} \equiv [PUL] \quad (5\text{-}25)$$

where the r, i and p superscripts denote rigid forces, inertial forces and static forces, respectively. Due to the Boolean matrix selection of the acceleration parameters and the fact that the rigid aerodynamics produces no forces for unit accelerations, the rigid and inertial unit solution vectors are either zero or nonzero for any parameter. This union of elastic unit solutions is denoted:

$$\{u^x{}_{lx}\} = \{u^x{}_{lx}\}^r - \{u^x{}_{lr}\}^i\left[T_{rR}\right]\left[T_{Rx}\right] \quad (5\text{-}26)$$

Using these relations leads to a simplified form of (5-22):

$$\{u_l\} = \{u^x{}_{lx}\}\{u_x\} + \{u^x{}_l\}^p \quad (5\text{-}27)$$

In this form, we can see that the first term is the deflection due to the aeroelastically corrected rigid aerodynamic loads and inertial forces and the second is the deflection due to the aeroelastically corrected statically applied load.

Substituting (5-27) into the second row of , results in the following relation that is the equivalent of Eq. 2-74 of Ref. 12 (ZZX * UX = PZ).

$$\left\{\left[m_r\right]\left[T_{rR}\right]\left[T_{Rx}\right] - \bar{q}\left[Q^z{}_{rl}\right]\{u^x{}_{lx}\} - \bar{q}\left[Q^z{}_{rx}\right]\right\}\{u_x\} = \left[P^z{}_r\right] + \bar{q}\left[Q^z{}_{rl}\right]\{u^x{}_l\}^p \quad (5\text{-}28)$$

There is one equation for every supported degree of freedom and the unknowns are the free parameters in $\{u_x\}$. Note that the right hand side is zero unless there are statically applied loads.

Restrained Stability Derivatives from the Unit Solutions

By inspection of (5-28), we can see that the rigid (splined) stability derivatives in the reference coordinate system are:

$$\begin{Bmatrix} C_x \\ C_y \\ C_z \\ C_{mx} \\ C_{my} \\ C_{mz} \end{Bmatrix}_{rigid} = [N]\left[T_{rR}\right]^T\left[D_{lr}\right]^T\left[Q_{lx}\right] \quad (5\text{-}29)$$

where $[N]$ matrix is the normalization matrix that is derived from the user input reference areas and lengths as:

(5-30)

$$[N] = \begin{bmatrix} 1/S & 0 & 0 & 0 & 0 & 0 \\ 0 & 1/S & 0 & 0 & 0 & 0 \\ 0 & 0 & 1/S & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/Sb_{ref} & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/Sc_{ref} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/Sb_{ref} \end{bmatrix}$$

Similarly, the elastically corrected stability derivatives are:

(5-31)

$$\begin{Bmatrix} C_x \\ C_y \\ C_z \\ C_{mx} \\ C_{my} \\ C_{mz} \end{Bmatrix}_{rest} = \begin{Bmatrix} C_x \\ C_y \\ C_z \\ C_{mx} \\ C_{my} \\ C_{mz} \end{Bmatrix}_{rigid} - [N] \left[T_{rR}\right]^T \left\{ \left[Q^z{}_{rl}\right] \{u^x{}_{lx}\} + \bar{q} \left[Q^z{}_{rx}\right] \right\}$$

Unrestrained Analysis and Stability Derivatives (Displacements Relative to Mean Axis)

In addition to the restrained analysis (trim and stability derivative calculation), MSC.Nastran computes the unrestrained stability derivatives: force increments due to unit perturbations about the mean axes. To introduce the mean axes, an orthogonality criterion is applied such that the rigid body motions are orthogonal to the elastic deflections:

(5-32)

$$\left[D^T{}_{lr} \; I_{rr}\right] \begin{bmatrix} M_{ll} & M_{lr} \\ M_{rl} & M_{rr} \end{bmatrix} \begin{bmatrix} \bar{u}_l \\ \bar{u}_r \end{bmatrix} = 0$$

where we introduce the overbar to denote deflections in the mean axis system. These equations are added to those of to yield:

(5-33)

$$\begin{bmatrix} K_{ll} & K_{lr} \\ M^z{}_{rl} & M^z{}_{rr} \\ 0 & 0 \end{bmatrix} \begin{Bmatrix} \bar{u}_l \\ \bar{u}_r \end{Bmatrix} - \bar{q} \begin{bmatrix} Q_{ll} & Q_{lr} \\ 0 & 0 \\ Q^z{}_{rl} & Q^z{}_{rr} \end{bmatrix} \begin{Bmatrix} \bar{u}_l \\ \bar{u}_r \end{Bmatrix} = \left\{ \bar{q} \begin{bmatrix} Q_{lx} \\ 0 \\ Q^z{}_{rx} \end{bmatrix} - \begin{bmatrix} M_{ll} D_{lr} + M_{lr} \\ 0 \\ m_r \end{bmatrix} \left[T_{rR}\right] \left[T_{Rx}\right] \right\} \{\bar{u}_x\} + \begin{Bmatrix} P_l \\ 0 \\ P^z{}_r \end{Bmatrix}$$

where we again use the “z” superscript:

$$\left[M^z{}_{rl}\right] = D^T{}_{lr} M_{ll} + M_{rl} \equiv [MZRL] \tag{5-34}$$

$$\left[M^{z}_{rr}\right] = D^{T}_{lr}M_{lr} + M_{rr} \equiv [MZRR] \tag{5-35}$$

Again, solving the first equation for $\{\bar{u}_l\}$ in term of $\{\bar{u}_x\}$ (noting that $\{\bar{u}_r\} \neq 0$):

$$\{\bar{u}_l\} = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\left[\left\{\bar{q}\left[Q_{lx}\right]\right.\right.$$
$$\left.\left. -\left(\left[M_{ll}\right]\left[D_{lr}\right] + \left[M_{lr}\right]\right)\left[T_{rR}\right]\left[T_{Rx}\right]\right\}\{\bar{u}_x\} - \left(\left[K_{lr}\right] - \bar{q}\left[Q_{lr}\right]\right)\{\bar{u}_r\} + \{P_l\}\right] \tag{5-36}$$

Looking at (5-36), we can rewrite it in terms of the three component "unit solutions" of the restrained case and an additional term due to the rigid body displacements:

$$\{\overline{u^x_{lx}}\}^r = \{u^x_{lx}\}^r = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\left\{\bar{q}\left[Q_{lx}\right]\right\} \equiv [RUXLX] \tag{5-37}$$

$$\{\overline{u^x_{lr}}\}^i = \{u^x_{lr}\}^i = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\left\{\left(\left[M_{ll}\right]\left[D_{lr}\right] + \left[M_{lr}\right]\right)\right\} \equiv [IUXLR] \tag{5-38}$$

$$\{\overline{u^x_l}\}^p = \{u^x_l\}^p = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\{P_l\} \equiv [PUL] \tag{5-39}$$

$$\{\overline{u^r_{lr}}\} = \left\{\left[K_{ll}\right] - \bar{q}\left[Q_{ll}\right]\right\}^{-1}\left\{\left(\left[K_{lr}\right] - \bar{q}\left[Q_{lr}\right]\right)\right\} \equiv [URLR] \tag{5-40}$$

where the r, i and p superscripts again denote rigid forces, inertial forces and static forces, respectively.

Using these relations leads (using the combination of elastic unit solutions of (5-26)) to a simplified form of (5-36):

$$\{\bar{u}_l\} = \{u^x_{lx}\}\{\bar{u}_x\} - \{\overline{u^r_{lr}}\}\{\bar{u}_r\} + \{u^x_l\}^p \tag{5-41}$$

In this form, we can see that we have added a term to the restrained solution due to the "elastically corrected" rigid body motion of the support points.

Substituting (5-41) into the second and third rows of (5-33), and keeping the accelerations of the support points $\{\ddot{\bar{u}}_r\}$ separate from their $\{\bar{u}_x\}$ counterparts we obtain:

$$\begin{bmatrix} [M^z_{rl}]\{u^x_{lx}\}^r & \left\{-[M^z_{rl}]\{\overline{u^r_{lr}}\} + [M^z_{rr}]\right\} & -[M^z_{rl}]\{u^x_{lr}\}^i \\ \left\{-\bar{q}[Q^z_{rx}] - \bar{q}[Q^z_{rl}]\{u^x_{lx}\}^r\right\} & \left\{\bar{q}[Q^z_{rl}]\{\overline{u^r_{lr}}\} - \bar{q}[Q^z_{rr}]\right\} & \left\{[m_r] + \bar{q}[Q^z_{rl}]\{u^x_{lr}\}^i\right\} \end{bmatrix}$$

$$\begin{Bmatrix} \bar{u}_x \\ \bar{u}_r \\ \{\bar{\ddot{u}}_r\} \end{Bmatrix} = \begin{Bmatrix} -[M^z_{rl}]\{u^x_l\}^p \\ \bar{q}[Q^z_{rl}]\{u^x_l\}^p + [P^z_r] \end{Bmatrix}$$

Using the first equation to solve for $\{\bar{u}_r\}$ in terms of $\{\bar{u}_x\}$ and $\{\bar{\ddot{u}}_r\}$, we obtain:

$$\{\bar{u}_r\} = \{\overline{u^x_{rx}}\}^r\{\bar{u}_x\} - \{\overline{u^x_{rr}}\}^i\{\bar{\ddot{u}}_r\} + \{\overline{u^x_r}\}^p \tag{5-42}$$

$$\{\overline{u^x_{rx}}\}^r = \left\{-[M^z_{rl}]\{\overline{u^r_{lr}}\} + [M^z_{rr}]\right\}^{-1}\left\{-[M^z_{rl}]\{u^x_{lx}\}^r\right\} \equiv [RUXRX] \tag{5-43}$$

$$\{\overline{u^x_{rr}}\}^i = \left\{-[M^z_{rl}]\{\overline{u^r_{lr}}\} + [M^z_{rr}]\right\}^{-1}\left\{-[M^z_{rl}]\{u^x_{lr}\}^i\right\} \equiv [IUXRR] \tag{5-44}$$

and

$$\{\overline{u^x_r}\}^p = \left\{-[M^z_{rl}]\{\overline{u^r_{lr}}\} + [M^z_{rr}]\right\}^{-1}\left\{-[M^z_{rl}]\{u^x_l\}^p\right\} \equiv [PUR] \tag{5-45}$$

Substituting (5-42) into the second equation of leads to trim equation:

$$\begin{aligned} &\left[[m_r] + \bar{q}[Q^z_{rl}]\{u^x_{lr}\}^i - \left\{\bar{q}[Q^z_{rl}]\{\overline{u^r_{lr}}\} - \bar{q}[Q^z_{rr}]\right\}\{\overline{u^x_{rr}}\}^i\right]\{\bar{\ddot{u}}_r\} + \\ &\left[\left\{-\bar{q}[Q^z_{rx}] - \bar{q}[Q^z_{rl}]\{u^x_{lx}\}^r + \left\{\bar{q}[Q^z_{rl}]\{\overline{u^r_{lr}}\} - \bar{q}[Q^z_{rr}]\right\}\{\overline{u^x_{rx}}\}^r\right\}\right]\{\bar{u}_x\} \\ &= \bar{q}[Q^z_{rl}]\{u^x_l\}^p + [P^z_r] - \left\{\bar{q}[Q^z_{rl}]\{\overline{u^r_{lr}}\} - \bar{q}[Q^z_{rr}]\right\}\{u^x_l\}^p \end{aligned} \tag{5-46}$$

which we can solve for $\{\bar{\ddot{u}}_r\}$ in terms of $\{\bar{u}_x\}$ if we ignore the static load (which does not participate in the stability derivative calculation. Then, if we premultiply the resultant accelerations by the total mass matrix m_r, we have the forces resulting from unit perturbation of the trim parameters: the stability derivatives. To solve, we define the intermediate quantities:

$$K^z_{rr} = -\left\{\bar{q}[Q^z_{rl}]\{\overline{u^r_{lr}}\} - \bar{q}[Q^z_{rr}]\right\} \equiv [K2RR] \equiv [KZRR] \tag{5-47}$$

$$M^I_{rr} = \left[m_r\right] + \bar{q}\left[Q^z{}_{rl}\right]\{u^x{}_{lr}\}^i + K^z{}_{rr}\{\overline{u^x{}_{rr}}\}^i \equiv [MIRR] \tag{5-48}$$

$$K^{r1}_{zx} = \left\{-\bar{q}\left[Q^z{}_{rx}\right] - \bar{q}\left[Q^z{}_{rl}\right](\{u^x{}_{lx}\}^r - \{K^z{}_{rr}\})\{\overline{u^x{}_{rx}}\}^r\right\} = [KR1ZX] \tag{5-49}$$

The accelerations of the r-set due to the trim parameters are then

$$\{\bar{\ddot{u}}_r\}^x = \left[M^I{}_{rr}\right]^{-1}\left[K^{r1}_{zx}\right] \tag{5-50}$$

Then, the dimensional unrestrained stability derivatives can be computed as:

$$\left\{\begin{matrix} C_x \\ C_y \\ C_z \\ C_{mx} \\ C_{my} \\ C_{mz} \end{matrix}\right\}_{unresi} = -\left[N\right]\left[T_{rR}\right]^T\left[m_r\right]\left[M^I{}_{rr}\right]^{-1}\left[K^{r1}_{zx}\right] \tag{5-51}$$

Recovery of Unit Solutions to the a-set and to the k-set

To compute the distributed force increments, we need to recover the unit solutions. Each displacement has a component due to the rigid aerodynamic forces and a component due to the accelerations. Using the previous development, we can summarize the l-set and r-set components as shown in Table 5-1.

To obtain the total elastic increment displacements, the sums of the rigid and inertial increments must be obtained using (5-26) for the restrained case and using the combination of (5-41) and (5-42) for the unrestrained case. Note that, in this formulation, the inertial increment is always **subtracted** from the rigid increment.

Table 5-1 Unit Solution Data Recovery on Structural Degrees of Freedom

Component	Due to Rigid Forces	Due to Inertial Forces
Restrained $\{u_r\}$	$= 0$	$= 0$
Restrained $\{u_l\}$	$= [RUXLX]$ (5-23)	$= [IUXLR][RFSOP]$ (5-24)
Unrestrained $\{\bar{\ddot{u}}_r\}$	$= 0$	$= [MIRR]^{-1}[KR1ZX]$ (5-50)
Unrestrained $\{\bar{u}_r\}$	$= [RUXRX]$ (5-43)	$= [IUXRR][MIRR]^{-1}[KR1ZX]$ (5-44) and (5-50)
Unrestrained $\{\bar{u}_l\}$	$= [RUXLX] - [URLR][RUXRX]$ (5-23), (5-43) and (5-40)	$= [URLR][IUXRR][MIRR]^{-1}[KR1ZX]$ (5-44), (5-50) and (5-40)

Recovery of these displacements to the a-set involves only the merging of the r-set and l-set partitions. To give names to these a-set data:

(5-52)

$$[UXAX] = \{u^x{}_{ax}\}_{rest} = \begin{bmatrix} [RUXLX] - [IUXLR][RFSOP] \\ 0 - 0 \end{bmatrix}$$

(5-53)

$$[UUXAX] = \{u^x{}_{ax}\}_{unrest} =$$

$$\begin{bmatrix} \{[RUXLX]-[URLR][RUXRX]\}-\{[URLR][IUXRR][MIRR]^{-1}[KR1ZX]\} \\ \{[RUXRX]\} - \{[IUXRR][MIRR]^{-1}[KR1ZX]\} \end{bmatrix}$$

where the rigid increment and inertial increment are shown separately with the appropriate sign. These displacement fields are the basic data needed to compute the increments in force due to the elastic response of the aircraft.

To obtain the same data in the k-set, the displacement spline relationship can be applied to each of the displacement fields. In the a-set, this is

$$\{u^x{}_{kx}\} = \left[G^d{}_{ka}\right]\{u^x{}_{ax}\} \tag{5-54}$$

where $\{u^x{}_{ax}\}$ represents the appropriate displacement field (rigid or inertial, restrained or unrestrained).

Distributed Force Increments from Unit Solutions

For purposes of aircraft simulation, it is useful to have the distributed forces for each of the aerodynamic extra points that integrate to the stability derivatives. These data can be used to perform incremental bending moment, shear and torque calculations for a particular set of trim parameters ($\{u_x\}$ or $\{\overline{u_x}\}$ as appropriate). The rigid forces are trivially computed from:

$$\left[F_{gx}\right] = \bar{q}\left[G^p{}_{gk}\right]\left[F_{kx}\right] = \bar{q}\left[G^p{}_{gk}\right]\left[Q_{kx}\right] \tag{5-55}$$

To obtain the rigid forces and force increments from the unit solutions one uses the forces due to the unit solution's displacements using the usual relationship:

$$\left[\Delta F_{gx}\right] = \bar{q}\left[G^p{}_{gk}\right]\left[\Delta F_{kx}\right] = \bar{q}\left[G^p{}_{gk}\right]\left[Q_{kk}\right]\{u^x{}_{kx}\} \tag{5-56}$$

where the displacement field(s) due to the perturbations $\{u^x{}_{kx}\}$ are simply the unit solutions of Table 5-1 recovered to the k-set.

Inertial Forces: The basic inertial forces (that is, those from the rigid airplane) are needed, since the terms inTable 5-1 represent only the aeroelastic increments due to the displacements caused by the inertial forces. The structural inertial force increments are computed directly from:

$$\left[\Delta F_{gx}\right]^{inertia} = \left[M_{gg}\right]\{\ddot{u}^x{}_{gx}\} \tag{5-57}$$

where the incremental accelerations $\{\ddot{u}^x{}_{gx}\}$ are computed by recovering the a-set incremental accelerations to the g-set in the usual manner. The a-set incremental accelerations are computed from:

$$\{\ddot{u}^x{}_{ax}\}_{rest} = \left[D_{ar}\right]\left[T_{rR}\right]\left[T_{Rx}\right] \tag{5-58}$$

for the restrained case in which $[D_{ar}]$ is computed by merging $[D_{lr}]$ with an identity matrix over the r-set. For the unrestrained case, the r-set accelerations come from (5-50) and the a-set accelerations are then:

$$\{\ddot{u}^x{}_{ax}\}_{unrest} = \left[D_{ar}\right]\left[M^I{}_{rr}\right]^{-1}\left[K^{r1}_{zx}\right] \tag{5-59}$$

Printed Output

The primary goal of FlightLoads is to bring visibility to the process of the development of flight loads. The main body of this manual has emphasized the Graphical User Interface of FlightLoads, but there are also important pieces of information that are produced by the system that do not have a visual component.

Instead, they are presented as tables within the printed `.f06` output that is produced as part of the MSC.Nastran run. This section documents this output and has been assembled by adapting documentation that is contained in the *MSC Nastran User's Guide for Aeroelasticity,* the Release Guide for Version 70.5 of MSC.Nastran and new material for this document.

Stability Derivatives and Hinge Moment Coefficients

Stability derivatives are invaluable for characterizing the performance and handling of an air vehicle. They allow for communication with Flight Controls engineers in the language they understand. Within the context of FlightLoads, they are also useful for checking the reasonableness of the results and the quality of the splines. Aeroelastic Solutions has derived the stability derivative calculations in pieces. Hinge moment coefficients are a special type of stability derivative that requires special handling and display. This subsection brings this information together and provides an illustration of how the data appear in the printed output.

Seven input matrices provide dimensional r-set force increment data for the computation of rigid, elastic restrained and elastic unrestrained intercepts, stability derivatives and hinge moments. These data are;

- KRZX - restrained dimensional coefficients
- Z1ZX - unrestrained dimensional coefficients
- RSTAB - rigid dimensional coefficient from aero mesh
- KSAZX - rigid dimensional coefficient from structural mesh
- RHMC - Frigid dimensional hinge moments
- ERHM - elastic, restrained dimensional hinge moments
- EUHM - elastic, unrestrained dimensional hinge moments

Note that the hinge moment data are a single value (hinge moment) for each control surface while the others are single values for each support point for each control surface/intercept.

Each of these matrices can be related to the unit solutions approach. For example:

$$KRZX = \bar{q}\left[Q^z{}_{rl}\right]\left(\{u^x{}_{lx}\}^r - \{u^x{}_{lx}\}^i\left[T_{rR}\right]\left[T_{Rx}\right]\right) - \bar{q}\left[Q^z{}_{rx}\right] \quad (5\text{-}60)$$

$$Z1ZX = -\left[T_{rR}\right]^T\left[m_r\right]\left[M^I{}_{rr}\right]^{-1}\left[K^{r1}_{zx}\right] \quad (5\text{-}61)$$

$$RSTAB = \bar{q}[SRKT][QKX] \quad (5\text{-}62)$$

$$KSAZX = \bar{q}[DAR][GPAK][SRKT][QKX] \quad (5\text{-}63)$$

Hinge moment derivatives: These can also be computed easily from the unit solutions' forces once the $[\Delta F_{kx}]$ are available from each of the unit solutions using intermediate data from the application of (5-56).

MSC Nastran computes a HMKT matrix that integrates the hinge moment for each control surface using a k-set sized force distribution as input. Therefore, all the (dimensional) control surface hinge moment increments are immediately available as:

$$\left[\Delta F_{kx}\right] = \bar{q}\left[Q_{kk}\right]\left[G^d{}_{ka}\right]\{u^x{}_{ax}\} \quad (5\text{-}64)$$

$$\left[\Delta H^m_{xx}\right] = \left[\Delta F_{kx}\right]^T \left[H^m_{kx}\right] \tag{5-65}$$

$$[RHMCF] = \bar{q}[QKX]^T[HMKT] \tag{5-66}$$

$$[ERHM] = \bar{q}\left[\left[Q_{kk}\right]\left[G^d{}_{ka}\right][UXAX]\right]^T[HMKT] + [RHMCF] \tag{5-67}$$

$$[EUHM] = \bar{q}\left[\left[Q_{kk}\right]\left[G^d{}_{ka}\right][UUXAX]\right]^T[HMKT] + [RHMCF] \tag{5-68}$$

Figure 5-1 provides typical output for stability derivatives. Four sets of stability derivatives computed about the origin of the aerodynamic reference coordinate system and are generated for the system for each flight condition in that coordinate system:

1. Rigid unsplined
2. Rigid splined
3. Elastic restrained at the SUPORTed degrees of freedom
4. Elastic unrestrained

Before the stability derivatives are tabulated, the transformation from the basic to the reference coordinates is shown. This transformation provides a check on the input of the aerodynamic reference coordinate system for the stability derivatives. The stability derivatives for the rigid and elastic vehicle are shown next. The rigid derivatives are those that are obtained while neglecting elastic deformation of the vehicle. These derivatives are presented in two ways: unsplined and splined coefficients which provide checks on the splining and structural boundary conditions (e.g., single point constraints reacting forces through any actuator can cause apparent load loss). The unsplined coefficients are based on all the boxes in the aerodynamic model and are independent of the spline. usually, the two sets of coefficients are nearly identical unless there is an error in the spline input, such as not including all of the boxes. However, there may be situations where some boxes intentionally may not be connected to the spline, as in the case when no motion of certain boxes is desired. This latter case can be avoided by mapping the forces of those non-moving boxes using a FORCE spline.

```
HALF-SPAN MODEL, STATIC SYMMETRIC LOADING                                                      SUBCASE 1

NON - DIMENSIONAL   S T A B I L I T Y   A N D   C O N T R O L   D E R I V A T I V E   C O E F F I C I E N T S

                    CONFIGURATION = AEROSG2D XY = SYMMETRY = ASYMMETRIC XZ-SYMMETRY = SYMMETRIC

                              MACH = 9.0000E-01                   Q = 4.0000E+01

          TRANSFORMATION FROM BASIC TO REFERENCE COORDINATES:
         { X }        [ -1.0000   0.0000   0.0000  ]  { X }        { 1.5000E+01 }
         { Y }    =   [  0.0000   1.0000   0.0000  ]  { Y }    +   { 0.0000E+00 }
         { Z }REF     [  0.0000   0.0000  -1.0000  ]  { Z }BAS     { 0.0000E+00 }

TRIM VARIABLE        COEFFICIENT          RIGID                           ELASTIC
                                    UNSPLINED        SPLINED        RESTRAINED      UNRESTRAINED
INTERCEPT               CX        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ       -8.420787E-03   -8.420787E-03   -8.463759E-03   -8.508835E-03
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY      -6.008128E-03   -6.008129E-03   -6.031396E-03   -6.063685E-03
                        CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00

ANGLEA                  CX        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ       -5.070976E+00   -5.070976E+00   -5.103214E+00   -5.127251E+00
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY      -2.870932E+00   -2.870932E+00   -2.889138E+00   -2.906503E+00
                        CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00

PITCH                   CX        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ       -1.207429E+01   -1.207429E+01   -1.208659E+01   -1.215826E+01
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY      -9.953999E+00   -9.953999E+00   -9.956323E+00   -1.000715E+01
                        CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00

URDD3                   CX        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ        0.000000E+00    0.000000E+00    3.154314E-03    0.000000E+00
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY       0.000000E+00    0.000000E+00    2.368632E-03    0.000000E+00
                      CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00

URDD5                 CX         0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ        0.000000E+00    0.000000E+00    5.904781E-02    0.000000E+00
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY       0.000000E+00    0.000000E+00    3.950050E-02    0.000000E+00
                        CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00

ELEV                    CX        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CY        0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CZ       -2.461395E-01   -2.461395E-01   -2.537565E-01   -2.519655E-01
                        CMX       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
                        CMY       5.715300E-01    5.715300E-01    5.666561E-01    5.677909E-01
                        CMZ       0.000000E+00    0.000000E+00    0.000000E+00    0.000000E+00
```

Figure 5-1 Stability Derivative Example

Non-dimensional hinge moment coefficient data are output in the format shown in Figure 5-2. Each aerodynamic extra point produces its own row of hinge moment coefficients. The hinge moments are computed directly from the aerodynamic model, rather than from the structural model that is used for the splined stability derivative data. For this reason, there is no comparison of the splined and unsplined hinge moment coefficients as there is for the stability derivatives. There are values for the restrained and unrestrained coefficients.

```
N O N - D I M E N S I O N A L   H I N G E   M O M E N T   D E R I V A T I V E   C O E F F I C I E N T S

   CONTROL SURFACE = ELEV        REFERENCE CHORD LENGTH =  1.000000E+01     REFERENCE AREA =  2.000000E+02

               TRIM VARIABLE             RIGID                          ELASTIC
                                                                 RESTRAINED      UNRESTRAINED
                 INTERCEPT           -7.706106E-05              -7.767072E-05    -7.760999E-05
                 ANGLEA               1.551577E-02               1.510351E-02     1.494753E-02
                 PITCH               -3.062173E-01              -3.066317E-01    -3.070683E-01
                 URDD3                0.000000E+00               2.263822E-05     0.000000E+00
                 URDD5                0.000000E+00               2.893892E-04     0.000000E+00
                 ELEV                 5.966925E-02               5.960630E-02     5.961495E-02
```

Figure 5-2 Hinge Moment Coefficients Example

Note also that since the integration is performed on the aerodynamic mesh, the inertial moment is not included in the hinge moment. However, the aeroelastic load induced by the deformations caused by the inertial forces is included.

HP and HP0 Matrices

The HP and HP0 matrices are the r-set displacements due to the restrained unit trim parameter forces and the aerodynamic intercept force, respectively. The displacements that correspond to these terms are found in (5-43) and (5-44), so that

$$\langle HP0|HP\rangle = \left[m_r\right]^{-1}\left(\{\overline{u^x}_{rx}\}^r - \{\overline{u^x}_{rr}\}^i\left[T_{rR}\right]\left[T_{Rx}\right]\right) \tag{5-69}$$

These terms are printed using a MSC Nastran matrix utility and appear in the output as shown in Figure 5-3.

```
INTERMEDIATE MATRIX ... HP0
                COLUMN      1
 6.451200E-05     -4.062406E-06

       INTERMEDIATE MATRIX ... HP
                COLUMN      1
 3.659248E-02     -1.994000E-03
                COLUMN      2
 9.315132E-02     -6.941710E-03
                COLUMN      3
 6.349168E-03     -1.624086E-04
                COLUMN      4
 1.459012E-02     -8.205780E-03
                COLUMN      5
-3.707928E-04      3.336261E-04
```

Figure 5-3 HP0 and HP Matrix Example

Trim Results and Control Surface Limits

The solution for the trim variables in FlightLoads is almost always carried out by solving the simultaneous equations given by (5-28). MSC.Nastran can also obtain a solution when there are redundant control surfaces and one or more of these control surfaces may be limited to a user specified value of the control surface deflection or hinge moment. These nonlinear solutions are a recent innovation and not considered part of the standard FlightLoads capability. However, the reporting of the trim analysis method in the printed output could make reference to these alternative methods, so they are touched upon here.

There are five possible trim solutions that can occur and these are given in Figure 5-4.

```
A.  TRIM ALGORITHM USED:  LINEAR TRIM SOLUTION WITHOUT REDUNDANT CONTROL SURFACES
B.  TRIM ALGORITHM USED:  LINEAR TRIM SOLUTION WITH REDUNDANT CONTROL SURFACES
C.  TRIM ALGORITHM USED:  SUCCESSFUL NONLINEAR TRIM SOLUTION WITHOUT EXPLICIT CONSIDERATION OF CONTROL SURFACE LIMITS
D.  TRIM ALGORITHM USED:  SUCCESSFUL NONLINEAR TRIM SOLUTION WITH EXPLICIT CONSIDERATION OF CONTROL SURFACE LIMITS
E.  TRIM ALGORITHM USED:  UNSUCCESSFUL NONLINEAR TRIM SOLUTION WITH EXPLICIT CONSIDERATION OF CONTROL SURFACE LIMITS
```

Figure 5-4 Trim Algorithm Selection

Figure 5-5 shows an example of the trim output, including the trim selection method. There are two tables in Figure 5-5. The first gives the aeroelastic trim variables by ID and name and value. The table also lists the variable type "RIGID BODY" or "CONTROL SURFACE" and the trim status. There are four types of trim status: FREE, FIXED, SCHEDULED and LINKED and they indicate that the variable value was free to vary, fixed by the user, scheduled using the CSSCHD entry or linked. (Scheduling has not been discussed in this guide, but is documented in the *MSC Nastran V70.5 Release Guide*.)

The second table provides control surface limit results. Each control surface has an upper and lower limit on its position and hinge moment value. The actual values are also listed and flags are provided to quickly identify any active limit (i.e., a response that is not violated but is within 3% of the prescribed limit marked with an (A)) while any violated constraint (i.e., a response that exceeds its limit by more than 0.3% marked with a (V)). Units of the output are radians for the positions and physically consistent units (e.g., newton-meters) for the hinge moments.

```
A E R O S T A T I C   D A T A   R E C O V E R Y   O U T P U T   T A B L E S

                        MACH = 9.000000E-01              Q = 2.400000E+03

        TRIM ALGORITHM USED: LINEAR TRIM SOLUTION WITHOUT REDUNDANT CONTROL SURFACES.

                                          AEROELASTIC TRIM VARIABLES

                              ID     LABEL              TYPE       TRIM STATUS     VALUE OF UX

                             501     ANGLEA          RIGID BODY         FIXED      3.151200E-03
                             502     PITCH           RIGID BODY         FIXED     -2.511685E-19
                             503     URDD3           RIGID BODY          FREE     -1.382048E+01
                             504     URDD5           RIGID BODY          FREE      2.109222E-01
                             505     CANARD     CONTROL SURFACE         FIXED      4.363320E-01
                             510     ELEV       CONTROL SURFACE         FIXED      0.000000E+00
                             511     SIDES           RIGID BODY         FIXED     -5.592523E-18
                             512     YAW             RIGID BODY         FIXED      1.983739E-17
                             513     ROLL            RIGID BODY         FIXED     -6.398960E-19
                             514     URDD2           RIGID BODY          FREE      3.563613E+00
                             515     URDD4           RIGID BODY          FREE      5.422129E-01
                             516     URDD6           RIGID BODY          FREE     -7.055259E-01
                             517     INBDAIL    CONTROL SURFACE         FIXED      0.000000E+00
                             518     OUTBDAIL   CONTROL SURFACE         FIXED      0.000000E+00
                             519     INBDRUD    CONTROL SURFACE         FIXED      4.363320E-01
                             520     OUTBDRUD   CONTROL SURFACE         FIXED      4.363320E-01

                                 CONTROL SURFACE POSITION AND HINGE MOMENT RESULTS

                         ACTIVE LIMITS ARE FLAGGED WITH AN (A),  VIOLATED LIMITS ARE FLAGGED WITH A (V).

                                     POSITION                                              HINGE MOMENT
  CONTROL SURFACE      LOWER LIMIT       VALUE        UPPER LIMIT       LOWER LIMIT       VALUE        UPPER LIMIT
    CANARD           -1.700000E-01   4.363320E-01   1.700000E-01 V   -1.000000E+04   3.161594E+05   1.000000E+04 V
    ELEV             -1.570796E+00   0.000000E+00   1.570796E+00          N/A       -1.002511E+03        N/A
    INBDAIL          -1.570796E+00   0.000000E+00   1.570796E+00          N/A        1.534836E+02        N/A
    OUTBDAIL         -1.570796E+00   0.000000E+00   1.570796E+00          N/A        1.534625E+02        N/A
    INBDRUD          -5.000000E-01   4.363320E-01   5.000000E-01     -2.000000E+06  -2.151824E+04   2.000000E+06
    OUTBDRUD         -8.760000E-01   4.363320E-01   8.760000E-01          N/A       -2.165500E+03        N/A
```

Figure 5-5 Trim Output and Control Surface Limits

Pressure and Forces on the Aerodynamic Elements

As a final example of printed output, Figure 5-6 depicts pressure and forces that are imposed on the aerodynamic model at trim. The pressures are printed in both dimensional and non-dimensional form and are listed in order of the aerodynamic elements starting from one.

The force output is printed in the format of standard OLOAD output in MSC.Nastran. Six degrees of freedom are printed for each aerodynamic grid even though one or more terms is zero for each grid.

```
EXAMPLE HA144C: HALF SPAN 15-DEG SWEPT UNTAPERED WING  HA144C           MARCH  19, 1999  MSC/NASTRAN  3/18/98   PAGE    26
    CANTILEVERED WIND TUNNEL MOUNT, DOUBLET-LATTICE AERO
    0.041 IN PLATE W/BEVELLED LEADING AND TRAILING EDGES
                              A E R O S T A T I C   D A T A   R E C O V E R Y   O U T P U T   T A B L E S
                                     MACH = 4.500000E-01               Q = 2.000000E+00
                                      AERODYNAMIC PRESSURES ON THE AERODYNAMIC ELEMENTS
                                                     AERODYNAMIC PRES.       AERODYNAMIC
                                   GRID   LABEL        COEFFICIENTS           PRESSURES
                                     1     LS          1.650689E+00         3.301379E+00
                                     2     LS          7.473636E-01         1.494727E+00
                                     3     LS          5.366663E-01         1.073333E+00
                                     4     LS          4.333380E-01         8.666760E-01
                                     5     LS          1.647448E+00         3.294896E+00
                                     6     LS          7.004333E-01         1.400867E+00
                                     7     LS          4.834797E-01         9.669595E-01
                                     8     LS          2.633697E-01         5.267394E-01
                                     9     LS          1.528240E+00         3.056480E+00
                                    10     LS          6.702325E-01         1.340465E+00
                                    11     LS          5.308362E-01         1.061672E+00
                                    12     LS          2.525186E-01         5.050372E-01
                                    13     LS          1.485955E+00         2.971910E+00
                                    14     LS          6.615313E-01         1.323063E+00
                                    15     LS          4.091378E-01         8.182756E-01
                                    16     LS          2.007488E-01         4.014975E-01
                                    17     LS          1.409623E+00         2.819246E+00
                                    18     LS          5.771903E-01         1.154381E+00
                                    19     LS          3.324490E-01         6.648980E-01
                                    20     LS          1.479835E-01         2.959669E-01
    *** LABEL NOTATIONS:  LS = LIFTING SURFACE,   ZIB = Z INTERFERENCE BODY ELEMENT,  ZSB = Z SLENDER BODY ELEMENT,
                                                   YIB = Y INTERFERENCE BODY ELEMENT,  YSB = Y SLENDER BODY ELEMENT.
```

```
EXAMPLE HA144C: HALF SPAN 15-DEG SWEPT UNTAPERED WING  HA144C           MARCH  19, 1999  MSC/NASTRAN  3/18/98   PAGE    27
 CANTILEVERED WIND TUNNEL MOUNT, DOUBLET-LATTICE AERO
 0.041 IN PLATE W/BEVELLED LEADING AND TRAILING EDGES
                         A E R O S T A T I C   D A T A   R E C O V E R Y   O U T P U T   T A B L E S
                                MACH = 4.500000E-01               Q = 2.000000E+00
                                 AERODYNAMIC FORCES ON THE AERODYNAMIC ELEMENTS
GROUP  GRID ID  LABEL        T1                T2                T3                R1                R2                R3
  1      101     LS    0.000000E+00      0.000000E+00      1.573657E+00      0.000000E+00      2.036459E-01      0.000000E+00
  1      102     LS    0.000000E+00      0.000000E+00      7.124863E-01      0.000000E+00      9.220241E-02      0.000000E+00
  1      103     LS    0.000000E+00      0.000000E+00      5.116217E-01      0.000000E+00      6.620865E-02      0.000000E+00
  1      104     LS    0.000000E+00      0.000000E+00      4.131155E-01      0.000000E+00      5.346102E-02      0.000000E+00
  1      105     LS    0.000000E+00      0.000000E+00      1.570567E+00      0.000000E+00      2.032460E-01      0.000000E+00
  1      106     LS    0.000000E+00      0.000000E+00      6.677462E-01      0.000000E+00      8.641262E-02      0.000000E+00
  1      107     LS    0.000000E+00      0.000000E+00      4.609172E-01      0.000000E+00      5.964701E-02      0.000000E+00
  1      108     LS    0.000000E+00      0.000000E+00      2.510791E-01      0.000000E+00      3.249199E-02      0.000000E+00
  1      109     LS    0.000000E+00      0.000000E+00      1.456922E+00      0.000000E+00      1.885393E-01      0.000000E+00
  1      110     LS    0.000000E+00      0.000000E+00      6.389548E-01      0.000000E+00      8.268674E-02      0.000000E+00
  1      111     LS    0.000000E+00      0.000000E+00      5.060637E-01      0.000000E+00      6.548938E-02      0.000000E+00
  1      112     LS    0.000000E+00      0.000000E+00      2.407343E-01      0.000000E+00      3.115328E-02      0.000000E+00
  1      113     LS    0.000000E+00      0.000000E+00      1.416610E+00      0.000000E+00      1.833226E-01      0.000000E+00
  1      114     LS    0.000000E+00      0.000000E+00      6.306595E-01      0.000000E+00      8.161324E-02      0.000000E+00
  1      115     LS    0.000000E+00      0.000000E+00      3.900446E-01      0.000000E+00      5.047544E-02      0.000000E+00
  1      116     LS    0.000000E+00      0.000000E+00      1.913804E-01      0.000000E+00      2.476641E-02      0.000000E+00
  1      117     LS    0.000000E+00      0.000000E+00      1.343843E+00      0.000000E+00      1.739055E-01      0.000000E+00
  1      118     LS    0.000000E+00      0.000000E+00      5.502544E-01      0.000000E+00      7.120807E-02      0.000000E+00
  1      119     LS    0.000000E+00      0.000000E+00      3.169346E-01      0.000000E+00      4.101432E-02      0.000000E+00
  1      120     LS    0.000000E+00      0.000000E+00      1.410775E-01      0.000000E+00      1.825674E-02      0.000000E+00
  1      121     LS    0.000000E+00      0.000000E+00      1.104092E+00      0.000000E+00      1.428800E-01      0.000000E+00
  1      122     LS    0.000000E+00      0.000000E+00      3.981524E-01      0.000000E+00      5.152467E-02      0.000000E+00
```

Figure 5-6   Aerodynamic Pressure and Force Output Example

F Aero Mesh Interface File Format

Introduction

This appendix describes the format of the various entries that make up the Aerodynamic Mesh Interface File (AMIF). This file format allows external meshes to be imported as MSC FLD "3D" aerodynamic meshes. These meshes can then be used to define other aerodynamic data such as Splines, Nonlinear Parametric Loads, Monitor Points, and Control Devices.

Definition of an Aerodynamic Mesh Interface File (AMIF)

To use "external" aerodynamic codes within MSC.FlightLoads and Dynamics the aerodynamic mesh has to be known. The mesh is needed for different purposes:

1. Mesh geometry has to be known to do any aeroelastic analysis.
2. Mesh geometry is needed for splining.
3. Mesh geometry (external node and element IDs) is used to display results and verify the splines.

To serve all these needs, a simple interface file has been defined. There is a minimum set of information needed to do most of this:

1. Aerodynamic grid points
2. Aerodynamic panels (boxes, elements, or whatever you want to call them)

This minimum set of information will be sufficient to make the aerodynamic mesh known to MSC.FlightLoads and Dynamics and is sufficient to do interactive splining and spline verification in MSC FLD. The following tables describe all entries of an AMIF designed to do all functions described above.

General Rules

1. All fields are separated by one or more spaces (not tabs).
2. The entry name must be upper case and must start in the first column.
3. The order is not important.

Entry: Comment

Comment line which will be ignored by the AMIF reader.

1 2 3 4 5 6 7 8 910

1	2	3	4	5	6	7	8
#							

Example:

This file was created by the XYZ Euler Code.

Entry: SGROUP (Required)

Supergroup name for MSC FLD

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
SGROUP	SNAME							

1	2	3	4	5	6	7	8	910
SGROUP	lann_1							

Field	Contents
SNAME	Supergroup name. The length of the name is limited to 8 characters. Please do not use any blanks or hyphens in the name, use underscores instead.

Entry: AGROUP

Aerodynamic group name which becomes a subgroup of the Supergroup.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
AGROUP	AGID	ANAME						

1	2	3	4	5	6	7	8	910
AGROUP	47	Wing_Top						

Field	Contents
AGID	Aerodynamic Group ID
ANAME	Name of the of the aerodynamic group. The group name is limited to 31 characters. Please do not use any blanks or hyphens in the name, use underscores instead. These groups are very useful to separate the aerodynamic model into functional components. It is important to split the model into groups to use the splining later on. It is strongly recommended that you split the model into parts like: Left_Wing_Top, Left_Wing_bot, Fuselage_Left, Fuselage_Right and so on. MSC Patran groups will be created automatically from these groups

Entry: ADOF (Required)

DOFs for the aerodynamic grids.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
ADOF	ADOFS							

1	2	3	4	5	6	7	8	910
ADOF	123							

Field	Contents
ADOFS	The list of degrees of freedom the aerodynamic grids possess. Note that only 125 is currently supported. Nonetheless, this entry is required.

Entry: ACORD

Coordinate frame for aero groups or grids. The coordinate frame definition is based upon the global (MSC Patran) or basic (MSC Nastran) coordinate system.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
ACORD	ACID	XOR	YOR	ZOR	XZA	YZA	ZZA	
XXZ	YXZ	ZXZ						

1	2	3	4	5	6	7	8	910
ACORD	1	0.0	0.0	0.0	0.0	0.0	1.0	
1.0	0.0	0.0						

Field	Contents
ACID	Aero coordinate frame ID
XOR	X-location of the origin
YOR	Y-location of the origin
ZOR	Z-location of the origin
XZA	X-location of a point on the z-axis
YZA	Y-location of a point on the z-axis
ZZA	Z-location of a point on the z-axis
XXZ	X-location of a point in the xz plane
YXZ	Y-location of a point in the xz plane
ZXZ	Z-location of a point in the xz plane

Note: Despite being a free field, this entry must be broken into two records as shown.

Entry: AECORD

Defining the aerodynamic reference coordinate frame. This entry is allowed to appear only once in a AMIF file. If this entry is not in the file the global (MSC Patran) or basic (MSC Nastran) coordinate system will be used as default.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
AECORD	ACID							

1	2	3	4	5	6	7	8	910
AECORD	1							

Field	Contents
ACID	Aero coordinate frame ID which will be used as aerodynamic reference frame.

Entry: AGRID

Aerodynamic grid point.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
AGRID	GID	X	Y	Z	ACID			

1	2	3	4	5	6	7	8	910
AGRID	781	10.456	27.678	1.938	1			

Field	Contents
GID	The ID of the aerodynamic grid point
X	X-location of the grid point
Y	Y-location of the grid point
Z	Z-location of the grid point
ACID	Aero coordinate frame ID which is the reference for the grid location. If the ID is 0 or blank the global (MSC Patran) or basic (MSC Nastran) coordinate system will be used for reference.

Entry: EIDSTART

Starting element ID and element ID offset values.

1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
EIDSTART	SEID	OFFSET						

1	2	3	4	5	6	7	8	910
EIDSTART	2000001	10000						

Field	Contents
SEID	Starting element ID. This value is of type integer.
OFFSET	Offset to be used when updating the starting element ID. This value is of type integer.

Note: These data are used to override the equivalent values in the MSC.FlightLoads and Dynamics Options menu.

Entry: AQUAD

Aerodynamic quadrilateral panel.

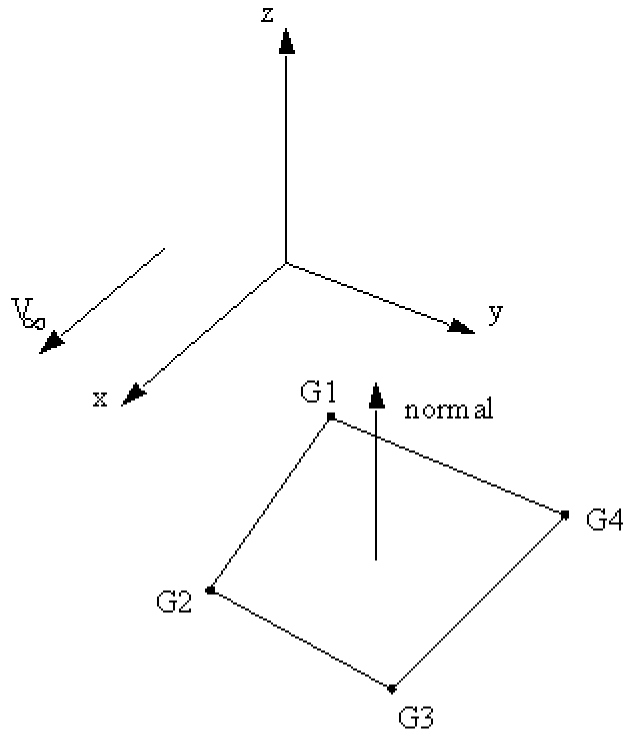
1	2	3	4	5	6	7	8	910
1	2	3	4	5	6	7	8	
AQUAD	APID	G1	G2	G3	G4	AGID		

1	2	3	4	5	6	7	8	910
AQUAD	123	12	13	46	45	47		

Field	Contents
APID	Aerodynamic panel ID
Gi	Aerodynamic grid point IDs of the connection points. Must be unique.
AGID	Aerodynamic Group ID the panel belongs to.

Remark:

The connection of G1 to G4 is not arbitrary. There are two boundary conditions to observe. The panel normal is defined by the connection grids (see sketch below). The normal of the panel as to point at any place from the structure into the fluid. Second the connection of G1 and G2 has to point roughly into the direction of the x-axis of the aerodynamic coordinate system.



Entry: ATRIA

Aerodynamic triangular panel.

1 2 3 4 5 6 7 8 910

1	2	3	4	5	6	7	8
ATRIA	APID	G1	G2	G3	AGID		

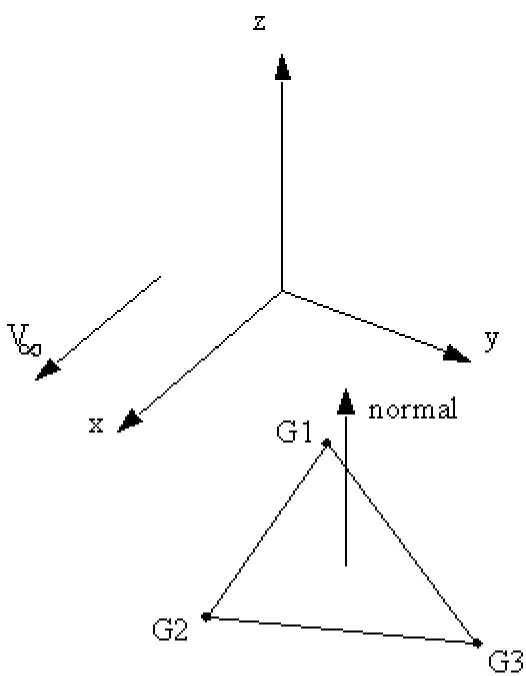
1 2 3 4 5 6 7 8 910

ATRIA	123	12	13	46	1234		

Field	Contents
APID	Aerodynamic panel ID
Gi	Aerodynamic grid point IDs of the connection points. Must be unique.
AGID	Aerodynamic Group ID the panel belongs to.

Remark:

The connection of G1 to G3 is not arbitrary. The panel normal is defined by the connection grids (see sketch below).



AMIF Format and Reading

The AMIF does not use a fixed format but keep in mind that all numbers will be stored in single precision. That is equivalent to I10 for IDs and 6-8 significant places for real numbers. The appearance of the fields is arbitrary. The standard extension of the AMIF is **ami** e.g. lann_1.ami.

When reading the AMIF into MSC FLD the grids and panels may be renumbered since MSC Patran requires unique grid and element IDs. While reading the file the aerodynamic super group will be created and within it the component groups will be created. All panels will be stored into their component group. The aerodynamic grids will be gathered accordingly into the groups. Finally, an attempt will be made to create an underlying surface from the elements/nodes of each group. If the attempt fails, you will receive a warning message that allows you to continue or terminate the import. If you continue, then the only effect will be that the group will NOT have an underlying surface associated with the region. This lack should have little or no impact. However, if you desire surfaces, you may wish to continue and double check the elements. Failure may indicate too warped a surface -- it maybe that, by breaking the group into pieces you can obtain a surface.

Element and Node Numbering - The following rules will be applied to your AMIF.

- If the lowest ID of any AGRID is greater than the largest grid ID in the MSC Patran database, the AMIF numbers will be retained. Otherwise, they will be offset.
- Element numbers are ALWAYS reassigned. However, the MSC FLD Options menu element numbering rules will be applied (contiguous IDs in the group with an integer multiple of the increment to determine the next groups stacking ID). An EIDSTART entry will override the options.

Hint:	While importing the aero grid locations the absolute value of the location components are important. The components are checked against the Global Tolerance. If a component is smaller than the Global Tolerance it is not created. So please make sure the Global Tolerance used is correct for the size of your model.

Sample AMI File

```
# AMIF File generated from A502 Flat File: ag0.M0400.a502
#
# Total number of components: 4
# Total number of elements: 160
#
SGROUP ag0
AGROUP 15001 comp_15001
AGROUP 16001 comp_16001
AGROUP 17001 comp_17001
AGROUP 18001 comp_18001
#
# Mesh Points
#
ADOF 123
#
AGRID  15001  21.996000 0.000000 0.000000
AGRID  15002  19.477145 0.000000 0.122751
AGRID  15003  15.213201 0.000000 0.315383
AGRID  15004  9.879925 0.000000 0.439229
AGRID  15005  5.258064 0.000000 0.387279
AGRID  15006  2.365530 0.000000 0.275519
AGRID  15007  0.907077 0.000000 0.178046
AGRID  15008  0.253696 0.000000 0.099594
AGRID  15009  0.000000 0.000000 0.000000
AGRID  15010  28.339230 7.804171 0.000000
AGRID  15011  26.043797 7.804171 0.111863
AGRID  15012  22.158064 7.804171 0.287409
AGRID  15013  17.297849 7.804171 0.400269
AGRID  15014  13.085947 7.804171 0.352928
AGRID  15015  10.449981 7.804171 0.251080
AGRID  15016  9.120892 7.804171 0.162253
AGRID  15017  8.525466 7.804171 0.090760
AGRID  15018  8.294273 7.804171 0.000000
AGRID  15019  33.668299 14.360604 0.000000
AGRID  15020  31.560567 14.360604 0.102716
AGRID  15021  27.992577 14.360604 0.263907
AGRID  15022  23.529789 14.360604 0.367538
AGRID  15023  19.662301 14.360604 0.324068
```

```
AGRID  15024  17.241882 14.360604 0.230549
AGRID  15025  16.021475 14.360604 0.148986
AGRID  15026  15.474738 14.360604 0.083338
AGRID  15027  15.262450 14.360604 0.000000
AGRID  15028  38.282779 20.037868 0.000000
AGRID  15029  36.337580 20.037868 0.094795
AGRID  15030  33.044725 20.037868 0.243557
AGRID  15031  28.926073 20.037868 0.339197
AGRID  15032  25.356816 20.037868 0.299079
AGRID  15033  23.123040 20.037868 0.212771
AGRID  15034  21.996742 20.037868 0.137497
AGRID  15035  21.492164 20.037868 0.076912
AGRID  15036  21.296246 20.037868 0.000000
AGRID  15037  42.442075 25.155112 0.000000
AGRID  15038  40.643375 25.155112 0.087656
AGRID  15039  37.598515 25.155112 0.225214
AGRID  15040  33.790053 25.155112 0.313651
AGRID  15041  30.489607 25.155112 0.276554
AGRID  15042  28.424065 25.155112 0.196746
AGRID  15043  27.382592 25.155112 0.127142
AGRID  15044  26.916016 25.155112 0.071119
AGRID  15045  26.734853 25.155112 0.000000
AGRID  15046  46.380000 30.000000 0.000000
AGRID  15047  44.720002 30.000000 0.080896
AGRID  15048  41.909939 30.000000 0.207847
AGRID  15049  38.395156 30.000000 0.289464
AGRID  15050  35.349216 30.000000 0.255228
AGRID  15051  33.442953 30.000000 0.181575
AGRID  15052  32.481790 30.000000 0.117338
AGRID  15053  32.051193 30.000000 0.065635
AGRID  15054  31.884000 30.000000 0.000000
AGRID  16001  0.000000 0.000000 0.000000
AGRID  16002  0.253696 0.000000 -0.099594
AGRID  16003  0.907077 0.000000 -0.178046
AGRID  16004  2.365530 0.000000 -0.275519
AGRID  16005  5.258064 0.000000 -0.387279
AGRID  16006  9.879925 0.000000 -0.439229
AGRID  16007  15.213201 0.000000 -0.315383
AGRID  16008  19.477145 0.000000 -0.122751
AGRID  16009  21.996000 0.000000 0.000000
AGRID  16010  8.294273 7.804171 0.000000
AGRID  16011  8.525466 7.804171 -0.090760
AGRID  16012  9.120892 7.804171 -0.162253
AGRID  16013  10.449981 7.804171 -0.251080
AGRID  16014  13.085947 7.804171 -0.352928
AGRID  16015  17.297849 7.804171 -0.400269
AGRID  16016  22.158064 7.804171 -0.287409
AGRID  16017  26.043797 7.804171 -0.111863
AGRID  16018  28.339230 7.804171 0.000000
AGRID  16019  15.262450 14.360604 0.000000
AGRID  16020  15.474738 14.360604 -0.083338
AGRID  16021  16.021475 14.360604 -0.148986
AGRID  16022  17.241882 14.360604 -0.230549
AGRID  16023  19.662301 14.360604 -0.324068
AGRID  16024  23.529789 14.360604 -0.367538
AGRID  16025  27.992577 14.360604 -0.263907
AGRID  16026  31.560567 14.360604 -0.102716
AGRID  16027  33.668299 14.360604 0.000000
AGRID  16028  21.296246 20.037868 0.000000
AGRID  16029  21.492164 20.037868 -0.076912
AGRID  16030  21.996742 20.037868 -0.137497
AGRID  16031  23.123040 20.037868 -0.212771
AGRID  16032  25.356816 20.037868 -0.299079
AGRID  16033  28.926073 20.037868 -0.339197
AGRID  16034  33.044725 20.037868 -0.243557
AGRID  16035  36.337580 20.037868 -0.094795
AGRID  16036  38.282779 20.037868 0.000000
AGRID  16037  26.734853 25.155112 0.000000
```

```
AGRID  16038  26.916016 25.155112 -0.071119
AGRID  16039  27.382592 25.155112 -0.127142
AGRID  16040  28.424065 25.155112 -0.196746
AGRID  16041  30.489607 25.155112 -0.276554
AGRID  16042  33.790053 25.155112 -0.313651
AGRID  16043  37.598515 25.155112 -0.225214
AGRID  16044  40.643375 25.155112 -0.087656
AGRID  16045  42.442075 25.155112 0.000000
AGRID  16046  31.884000 30.000000 0.000000
AGRID  16047  32.051193 30.000000 -0.065635
AGRID  16048  32.481790 30.000000 -0.117338
AGRID  16049  33.442953 30.000000 -0.181575
AGRID  16050  35.349216 30.000000 -0.255228
AGRID  16051  38.395156 30.000000 -0.289464
AGRID  16052  41.909939 30.000000 -0.207847
AGRID  16053  44.720002 30.000000 -0.080896
AGRID  16054  46.380000 30.000000 0.000000
AGRID  17001  46.380000 -30.000000 0.000000
AGRID  17002  44.720002 -30.000000 0.080896
AGRID  17003  41.909939 -30.000000 0.207847
AGRID  17004  38.395156 -30.000000 0.289464
AGRID  17005  35.349216 -30.000000 0.255228
AGRID  17006  33.442953 -30.000000 0.181575
AGRID  17007  32.481790 -30.000000 0.117338
AGRID  17008  32.051193 -30.000000 0.065635
AGRID  17009  31.884000 -30.000000 0.000000
AGRID  17010  42.442075 -25.155112 0.000000
AGRID  17011  40.643375 -25.155112 0.087656
AGRID  17012  37.598515 -25.155112 0.225214
AGRID  17013  33.790053 -25.155112 0.313651
AGRID  17014  30.489607 -25.155112 0.276554
AGRID  17015  28.424065 -25.155112 0.196746
AGRID  17016  27.382592 -25.155112 0.127142
AGRID  17017  26.916016 -25.155112 0.071119
AGRID  17018  26.734853 -25.155112 0.000000
AGRID  17019  38.282779 -20.037868 0.000000
AGRID  17020  36.337580 -20.037868 0.094795
AGRID  17021  33.044725 -20.037868 0.243557
AGRID  17022  28.926073 -20.037868 0.339197
AGRID  17023  25.356816 -20.037868 0.299079
AGRID  17024  23.123040 -20.037868 0.212771
AGRID  17025  21.996742 -20.037868 0.137497
AGRID  17026  21.492164 -20.037868 0.076912
AGRID  17027  21.296246 -20.037868 0.000000
AGRID  17028  33.668299 -14.360604 0.000000
AGRID  17029  31.560567 -14.360604 0.102716
AGRID  17030  27.992577 -14.360604 0.263907
AGRID  17031  23.529789 -14.360604 0.367538
AGRID  17032  19.662301 -14.360604 0.324068
AGRID  17033  17.241882 -14.360604 0.230549
AGRID  17034  16.021475 -14.360604 0.148986
AGRID  17035  15.474738 -14.360604 0.083338
AGRID  17036  15.262450 -14.360604 0.000000
AGRID  17037  28.339230 -7.804171 0.000000
AGRID  17038  26.043797 -7.804171 0.111863
AGRID  17039  22.158064 -7.804171 0.287409
AGRID  17040  17.297849 -7.804171 0.400269
AGRID  17041  13.085947 -7.804171 0.352928
AGRID  17042  10.449981 -7.804171 0.251080
AGRID  17043  9.120892 -7.804171 0.162253
AGRID  17044  8.525466 -7.804171 0.090760
AGRID  17045  8.294273 -7.804171 0.000000
AGRID  17046  21.996000 0.000000 0.000000
AGRID  17047  19.477145 0.000000 0.122751
AGRID  17048  15.213201 0.000000 0.315383
AGRID  17049  9.879925 0.000000 0.439229
AGRID  17050  5.258064 0.000000 0.387279
AGRID  17051  2.365530 0.000000 0.275519
```

```
AGRID  17052  0.907077 0.000000 0.178046
AGRID  17053  0.253696 0.000000 0.099594
AGRID  17054  0.000000 0.000000 0.000000
AGRID  18001  31.884000 -30.000000 0.000000
AGRID  18002  32.051193 -30.000000 -0.065635
AGRID  18003  32.481790 -30.000000 -0.117338
AGRID  18004  33.442953 -30.000000 -0.181575
AGRID  18005  35.349216 -30.000000 -0.255228
AGRID  18006  38.395156 -30.000000 -0.289464
AGRID  18007  41.909939 -30.000000 -0.207847
AGRID  18008  44.720002 -30.000000 -0.080896
AGRID  18009  46.380000 -30.000000 0.000000
AGRID  18010  26.734853 -25.155112 0.000000
AGRID  18011  26.916016 -25.155112 -0.071119
AGRID  18012  27.382592 -25.155112 -0.127142
AGRID  18013  28.424065 -25.155112 -0.196746
AGRID  18014  30.489607 -25.155112 -0.276554
AGRID  18015  33.790053 -25.155112 -0.313651
AGRID  18016  37.598515 -25.155112 -0.225214
AGRID  18017  40.643375 -25.155112 -0.087656
AGRID  18018  42.442075 -25.155112 0.000000
AGRID  18019  21.296246 -20.037868 0.000000
AGRID  18020  21.492164 -20.037868 -0.076912
AGRID  18021  21.996742 -20.037868 -0.137497
AGRID  18022  23.123040 -20.037868 -0.212771
AGRID  18023  25.356816 -20.037868 -0.299079
AGRID  18024  28.926073 -20.037868 -0.339197
AGRID  18025  33.044725 -20.037868 -0.243557
AGRID  18026  36.337580 -20.037868 -0.094795
AGRID  18027  38.282779 -20.037868 0.000000
AGRID  18028  15.262450 -14.360604 0.000000
AGRID  18029  15.474738 -14.360604 -0.083338
AGRID  18030  16.021475 -14.360604 -0.148986
AGRID  18031  17.241882 -14.360604 -0.230549
AGRID  18032  19.662301 -14.360604 -0.324068
AGRID  18033  23.529789 -14.360604 -0.367538
AGRID  18034  27.992577 -14.360604 -0.263907
AGRID  18035  31.560567 -14.360604 -0.102716
AGRID  18036  33.668299 -14.360604 0.000000
AGRID  18037  8.294273 -7.804171 0.000000
AGRID  18038  8.525466 -7.804171 -0.090760
AGRID  18039  9.120892 -7.804171 -0.162253
AGRID  18040  10.449981 -7.804171 -0.251080
AGRID  18041  13.085947 -7.804171 -0.352928
AGRID  18042  17.297849 -7.804171 -0.400269
AGRID  18043  22.158064 -7.804171 -0.287409
AGRID  18044  26.043797 -7.804171 -0.111863
AGRID  18045  28.339230 -7.804171 0.000000
AGRID  18046  0.000000 0.000000 0.000000
AGRID  18047  0.253696 0.000000 -0.099594
AGRID  18048  0.907077 0.000000 -0.178046
AGRID  18049  2.365530 0.000000 -0.275519
AGRID  18050  5.258064 0.000000 -0.387279
AGRID  18051  9.879925 0.000000 -0.439229
AGRID  18052  15.213201 0.000000 -0.315383
AGRID  18053  19.477145 0.000000 -0.122751
AGRID  18054  21.996000 0.000000 0.000000
#
# This EIDSTART will perserve the EIDs on import
#
EIDSTART 15001 1000
#
# Connectivity
#
AQUAD  15001 15001 15002 15011 15010  15001
AQUAD  15002 15010 15011 15020 15019  15001
AQUAD  15003 15019 15020 15029 15028  15001
AQUAD  15004 15028 15029 15038 15037  15001
```

```
AQUAD  15005 15037 15038 15047 15046  15001
AQUAD  15006 15002 15003 15012 15011  15001
AQUAD  15007 15011 15012 15021 15020  15001
AQUAD  15008 15020 15021 15030 15029  15001
AQUAD  15009 15029 15030 15039 15038  15001
AQUAD  15010 15038 15039 15048 15047  15001
AQUAD  15011 15003 15004 15013 15012  15001
AQUAD  15012 15012 15013 15022 15021  15001
AQUAD  15013 15021 15022 15031 15030  15001
AQUAD  15014 15030 15031 15040 15039  15001
AQUAD  15015 15039 15040 15049 15048  15001
AQUAD  15016 15004 15005 15014 15013  15001
AQUAD  15017 15013 15014 15023 15022  15001
AQUAD  15018 15022 15023 15032 15031  15001
AQUAD  15019 15031 15032 15041 15040  15001
AQUAD  15020 15040 15041 15050 15049  15001
AQUAD  15021 15005 15006 15015 15014  15001
AQUAD  15022 15014 15015 15024 15023  15001
AQUAD  15023 15023 15024 15033 15032  15001
AQUAD  15024 15032 15033 15042 15041  15001
AQUAD  15025 15041 15042 15051 15050  15001
AQUAD  15026 15006 15007 15016 15015  15001
AQUAD  15027 15015 15016 15025 15024  15001
AQUAD  15028 15024 15025 15034 15033  15001
AQUAD  15029 15033 15034 15043 15042  15001
AQUAD  15030 15042 15043 15052 15051  15001
AQUAD  15031 15007 15008 15017 15016  15001
AQUAD  15032 15016 15017 15026 15025  15001
AQUAD  15033 15025 15026 15035 15034  15001
AQUAD  15034 15034 15035 15044 15043  15001
AQUAD  15035 15043 15044 15053 15052  15001
AQUAD  15036 15008 15009 15018 15017  15001
AQUAD  15037 15017 15018 15027 15026  15001
AQUAD  15038 15026 15027 15036 15035  15001
AQUAD  15039 15035 15036 15045 15044  15001
AQUAD  15040 15044 15045 15054 15053  15001
AQUAD  16001 16001 16002 16011 16010  16001
AQUAD  16002 16010 16011 16020 16019  16001
AQUAD  16003 16019 16020 16029 16028  16001
AQUAD  16004 16028 16029 16038 16037  16001
AQUAD  16005 16037 16038 16047 16046  16001
AQUAD  16006 16002 16003 16012 16011  16001
AQUAD  16007 16011 16012 16021 16020  16001
AQUAD  16008 16020 16021 16030 16029  16001
AQUAD  16009 16029 16030 16039 16038  16001
AQUAD  16010 16038 16039 16048 16047  16001
AQUAD  16011 16003 16004 16013 16012  16001
AQUAD  16012 16012 16013 16022 16021  16001
AQUAD  16013 16021 16022 16031 16030  16001
AQUAD  16014 16030 16031 16040 16039  16001
AQUAD  16015 16039 16040 16049 16048  16001
AQUAD  16016 16004 16005 16014 16013  16001
AQUAD  16017 16013 16014 16023 16022  16001
AQUAD  16018 16022 16023 16032 16031  16001
AQUAD  16019 16031 16032 16041 16040  16001
AQUAD  16020 16040 16041 16050 16049  16001
AQUAD  16021 16005 16006 16015 16014  16001
AQUAD  16022 16014 16015 16024 16023  16001
AQUAD  16023 16023 16024 16033 16032  16001
AQUAD  16024 16032 16033 16042 16041  16001
AQUAD  16025 16041 16042 16051 16050  16001
AQUAD  16026 16006 16007 16016 16015  16001
AQUAD  16027 16015 16016 16025 16024  16001
AQUAD  16028 16024 16025 16034 16033  16001
AQUAD  16029 16033 16034 16043 16042  16001
AQUAD  16030 16042 16043 16052 16051  16001
AQUAD  16031 16007 16008 16017 16016  16001
AQUAD  16032 16016 16017 16026 16025  16001
```

```
AQUAD  16033 16025 16026 16035 16034  16001
AQUAD  16034 16034 16035 16044 16043  16001
AQUAD  16035 16043 16044 16053 16052  16001
AQUAD  16036 16008 16009 16018 16017  16001
AQUAD  16037 16017 16018 16027 16026  16001
AQUAD  16038 16026 16027 16036 16035  16001
AQUAD  16039 16035 16036 16045 16044  16001
AQUAD  16040 16044 16045 16054 16053  16001
AQUAD  17001 17001 17002 17011 17010  17001
AQUAD  17002 17010 17011 17020 17019  17001
AQUAD  17003 17019 17020 17029 17028  17001
AQUAD  17004 17028 17029 17038 17037  17001
AQUAD  17005 17037 17038 17047 17046  17001
AQUAD  17006 17002 17003 17012 17011  17001
AQUAD  17007 17011 17012 17021 17020  17001
AQUAD  17008 17020 17021 17030 17029  17001
AQUAD  17009 17029 17030 17039 17038  17001
AQUAD  17010 17038 17039 17048 17047  17001
AQUAD  17011 17003 17004 17013 17012  17001
AQUAD  17012 17012 17013 17022 17021  17001
AQUAD  17013 17021 17022 17031 17030  17001
AQUAD  17014 17030 17031 17040 17039  17001
AQUAD  17015 17039 17040 17049 17048  17001
AQUAD  17016 17004 17005 17014 17013  17001
AQUAD  17017 17013 17014 17023 17022  17001
AQUAD  17018 17022 17023 17032 17031  17001
AQUAD  17019 17031 17032 17041 17040  17001
AQUAD  17020 17040 17041 17050 17049  17001
AQUAD  17021 17005 17006 17015 17014  17001
AQUAD  17022 17014 17015 17024 17023  17001
AQUAD  17023 17023 17024 17033 17032  17001
AQUAD  17024 17032 17033 17042 17041  17001
AQUAD  17025 17041 17042 17051 17050  17001
AQUAD  17026 17006 17007 17016 17015  17001
AQUAD  17027 17015 17016 17025 17024  17001
AQUAD  17028 17024 17025 17034 17033  17001
AQUAD  17029 17033 17034 17043 17042  17001
AQUAD  17030 17042 17043 17052 17051  17001
AQUAD  17031 17007 17008 17017 17016  17001
AQUAD  17032 17016 17017 17026 17025  17001
AQUAD  17033 17025 17026 17035 17034  17001
AQUAD  17034 17034 17035 17044 17043  17001
AQUAD  17035 17043 17044 17053 17052  17001
AQUAD  17036 17008 17009 17018 17017  17001
AQUAD  17037 17017 17018 17027 17026  17001
AQUAD  17038 17026 17027 17036 17035  17001
AQUAD  17039 17035 17036 17045 17044  17001
AQUAD  17040 17044 17045 17054 17053  17001
AQUAD  18001 18001 18002 18011 18010  18001
AQUAD  18002 18010 18011 18020 18019  18001
AQUAD  18003 18019 18020 18029 18028  18001
AQUAD  18004 18028 18029 18038 18037  18001
AQUAD  18005 18037 18038 18047 18046  18001
AQUAD  18006 18002 18003 18012 18011  18001
AQUAD  18007 18011 18012 18021 18020  18001
AQUAD  18008 18020 18021 18030 18029  18001
AQUAD  18009 18029 18030 18039 18038  18001
AQUAD  18010 18038 18039 18048 18047  18001
AQUAD  18011 18003 18004 18013 18012  18001
AQUAD  18012 18012 18013 18022 18021  18001
AQUAD  18013 18021 18022 18031 18030  18001
AQUAD  18014 18030 18031 18040 18039  18001
AQUAD  18015 18039 18040 18049 18048  18001
AQUAD  18016 18004 18005 18014 18013  18001
AQUAD  18017 18013 18014 18023 18022  18001
AQUAD  18018 18022 18023 18032 18031  18001
AQUAD  18019 18031 18032 18041 18040  18001
AQUAD  18020 18040 18041 18050 18049  18001
```

```
AQUAD  18021 18005 18006 18015 18014  18001
AQUAD  18022 18014 18015 18024 18023  18001
AQUAD  18023 18023 18024 18033 18032  18001
AQUAD  18024 18032 18033 18042 18041  18001
AQUAD  18025 18041 18042 18051 18050  18001
AQUAD  18026 18006 18007 18016 18015  18001
AQUAD  18027 18015 18016 18025 18024  18001
AQUAD  18028 18024 18025 18034 18033  18001
AQUAD  18029 18033 18034 18043 18042  18001
AQUAD  18030 18042 18043 18052 18051  18001
AQUAD  18031 18007 18008 18017 18016  18001
AQUAD  18032 18016 18017 18026 18025  18001
AQUAD  18033 18025 18026 18035 18034  18001
AQUAD  18034 18034 18035 18044 18043  18001
AQUAD  18035 18043 18044 18053 18052  18001
AQUAD  18036 18008 18009 18018 18017  18001
AQUAD  18037 18017 18018 18027 18026  18001
AQUAD  18038 18026 18027 18036 18035  18001
AQUAD  18039 18035 18036 18045 18044  18001
AQUAD  18040 18044 18045 18054 18053  18001
```

1 References

References

1. **Albano, E., and Rodden, W. P. (1969)**. A Doublet-Lattice Method for Calculating Lift Distributions on Oscillating Surfaces in Subsonic Flows. *AIAA J.*, Vol. 7, pp. 279-285, p. 2192.
2. **Appa, Kari**, Finite-Surface Spline, *Journal of Aircraft*, Vol. 26, No. 5, May 1989. pp.495-496
3. **Chen, P. C., and Liu, D. D. (1985)**. A Harmonic Gradient Method for Unsteady Supersonic Flow Calculations. *J. Aircraft*, Vol. 22, pp. 371-379.
4. **Giesing, J. P., Kalman, T. P., and Rodden, W. P. (1971)**. Subsonic Unsteady Aerodynamics for General Configurations; Part I, Vol. I - Direct Application of the Nonplanar Doublet-Lattice Method. Air Force Flight Dynamics Laboratory Report No. AFFDL-TR-71-5, Part I, Vol. I.
5. **Giesing, J. P., Kalman, T., and Rodden, W. P. (1972a)**. Subsonic Unsteady Aerodynamics for General Configurations, Part II, Volume I - Application of the Doublet-Lattice-Method and the Method of Images to Lifting-Surface/Body Interference. Air Force Flight Dynamics Laboratory Report No. AFFDL-TR-71-5, Part II, Vol. I.
6. **Giesing, J. P., Kalman, T. P., and Rodden, W. P. (1972b)**. Subsonic Unsteady Aerodynamics for General Configurations; Part II, Volume II - Computer Program N5KA. Air Force Flight Dynamics Laboratory Report No. AFFDL-TR-71-5, Part II, Vol. II.
7. **Giesing, J. P., Kalman, T. P., and Rodden, W. P. (1972c)**. Subsonic Steady and Oscillatory Aerodynamics for Multiple Interfering Wings and Bodies. *J. Aircraft*, Vol. 9, pp. 693-702.
8. **Hough, G.R.**, "Remarks on Vortex-Lattice Methods", *J. Aircraft*, Vol. 10, No. 5, 1973, pp. 314-317.
9. **Hough, G.R.**, "Lattice Arrangement for Rapid Convergence", Vortex-Lattice Utilization, NASA SP-405, May 17-18, 1976, pp. 325-342.
10. **Liu, D. D., James, D. K., Chen, P. C., and Pototzky, A. S. (1991)**. Further Studies of Harmonic Gradient Method for Supersonic Aeroelastic Applications. *J. Aircraft*, Vol. 28, pp. 598-605.
11. **Rodden, W. P., Giesing, J. P., and Kalman, T. P. (1972)**. Refinement of the Nonplanar Aspects of the Subsonic Doublet-Lattice Lifting Surface Method. *J. Aircraft*, Vol. 9, pp. 69-73.
12. **Rodden, W.P., and Johnson, E. H., (1994)**. *MSC/NASTRAN Aeroelastic Analysis User's Guide*. The MacNeal-Schwendler Corporation.
13. **Rodden, W. P., and Revell, J. D. (1962)**. The Status of Unsteady Aerodynamic Influence Coefficients. Institute of the Aeronautical Sciences, Fairchild Fund Paper No. FF-33.
14. **Rubbert, P. E.**, "Theoretical Characteristics of Arbitrary Wings by a Non-Planar Vortex Lattice Method," D6-9244, 1964, The Boeing Co., Renton, WA.
15. Anon, "MSC.Nastran Quick Reference Guide 2001", MSC Software Corporation, 2001.

Index

MSC FlightLoads and Dynamics User's Guide