Adams 2021.2

Adams Flex User's Guide

HEXAGON

Corporate
MSC Software Corporation
5161 California Ave, Suite 200
University Research Park
Irvine, CA 92617
Telephone: (714) 540-8900
Email: americas.contact@mscsoftware.com

Europe, Middle East, Africa
MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) 89 431 98 70
Email: europe@mscsoftware.com

Japan
MSC Software Japan Ltd.
KANDA SQUARE 16F
2-2-1 Kanda Nishikicho, Chiyoda-ku
Tokyo 101-0054, Japan
Telephone: (81)(3) 6275 0870
Email: MSCJ.Market@mscsoftware.com

Asia-Pacific
MSC Software (S) Pte. Ltd.
100 Beach Road
#16-05 Shaw Tower
Singapore 189702
Telephone: 65-6272-0082
Email: APAC.Contact@mscsoftware.com

Worldwide Web
www.mscsoftware.com

Support
http://www.mscsoftware.com/Contents/Services/Technical-Support/Contact-Technical-Support.aspx

Disclaimer

ADAM:V2021.2:Z:Z:Z:DC-FLEX

Documentation Feedback

At MSC Software, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: documentation-feedback@mscsoftware.com.

Please include the following information with your feedback:

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

You may also provide your feedback about MSC Software documentation by taking a short 5-minute survey at: http://msc-documentation.questionpro.com.

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

Welcome to Adams Flex

About Adams Flex

Adams Flex software is an add-on module to the Adams® 2021.2 suite of software that lets you add flexible bodies to your models to achieve more realistic simulation results. The Adams Flex Help explains how to add flexible bodies to the models you build in Adams. It contains tutorial, overview, and theory information on integrating and managing flexible bodies.

Although you can use Adams Flex with Adams Solver, we recommend that you use Adams View because it provides an easy-to-use interface that is built right into the Adams View interface. Because of Adams View's advantages, the Adams Flex Help focuses on how to use Adams Flex with Adams View. For information on how to use Adams Flex with Adams Solver, see Flexible Body Modeling with Adams Solver.

Adams Flex Help also assumes that you know how to run Adams View or Adams Solver. It also assumes that you have a moderate proficiency level of finite element modeling. To use Adams Flex, you need access to a finite element analysis (FEA) program that interfaces with Adams so you can prepare a finite element (FE) model for use with Adams and create a Modal Neutral File (MNF) to be built into your Adams model.

See the section Adams File Types for more information on Adams Flex file types.

MD DB

In addition to using an MNF, a flexible body can be created directly from an MD DB. This offers better integration with Nastran and avoids generating an MNF. Please see Create an MD DB using MSC Nastran on how to create an MD DB that can be used in Adams.

Benefits of Adams Flex

By integrating flexible bodies into your model, you can:

- Capture inertial and compliance properties during handling and comfort simulations.
- Predict loads with greater accuracy by allowing Adams to account for flexibility during simulations.
- Study deformations.
- Examine the linear system modes of a flexible model when you use Adams Flex with Adams Linear. (For more on using Adams Flex with Adams Linear, see Using Adams Linear to Verify Flexible Bodies.)

Ways to Use Adams Flex

You can use Adams Flex with either Adams View or Adams Solver.

- **Adams View** - Using Adams Flex with Adams View, you can view the deformations of your flexible body, enhance the visual display, and easily specify the content of the flexible body that you want used in simulations. Adams View automates many of the operations for generating flexible body deformations for you.

- **Adams Solver** - You can use Adams Flex directly with Adams Solver, Adams's powerful analysis engine. We've provided Adams Solver statements and commands, as well as templates and tools to make it easier to create the necessary elements of a flexible body in Adams Solver.
- **Template-based products** - The template-based product, such as Adams Car, enhance the Adams Flex Modify dialog box and its functions to meet the needs of the types of models you are working with in that product, such as cars or engines. Refer to the template-based products online help for more information on these enhancements.

Learn about using Adams Flex with Adams Solver.

About Flexible Bodies in Adams Flex

Adams Flex uses an assumed modes method of modeling flexible bodies. This method of representing flexible bodies is called modal flexibility. Modal flexibility assigns a set of mode shapes (eigenvectors) to a flexible body. The flexible body modeling element designates a system state variable to each eigenvector and calculates the relative amplitude of each eigenvector during a time analysis. The principle of linear superposition is then used to combine the mode shapes at each time step to reproduce the total deformation of the flexible body.

This modal method of modeling flexibility can be very useful in problems that are characterized by high elasticity and moderate deflections (deflections less than 10% of a characteristic length of the body). Modal flexibility also provides a means of directly correlating Adams flexibility results with finite element analysis (FEA) results.

Learn more details about flexible bodies and when to use them.

- Where to Use Flexible Bodies
- Where to Use Flexible Bodies with Care

Where to Use Flexible Bodies

You should use flexible bodies wherever you expect component flexibility to affect the dynamic behavior of your model or when you require accurate information about the deformations of a component in your model. If you are a handling analyst who is concerned about the negative effects of component flexibility or a component designer who is concerned about component strength and fatigue life, you can benefit from the ability to very accurately model component flexibility.

Where to Use Flexible Bodies with Care

When you use flexible bodies, you should remember that flexible body deformations are *a linear combination of deformation shapes.* Consequently, you must take special precautions when modeling higher order deformations, such as those that occur when deformations are large, or when attempting to correctly model centrifugal stiffening of rotating systems. You can overcome these limitations by dividing a flexible body into multiple flexible bodies and assembling them in Adams View. This technique is described in Modeling Nonlinear Deformations.

About the Flexible Body Description File (MNF)

You use a description of a flexible body from a finite element analysis (FEA) program called a modal neutral file (MNF) as the foundation of a flexible body in Adams Flex. The information in an MNF includes:

- Geometry (location of nodes and node connectivity)
- Nodal mass and inertia
- Mode shapes
- Generalized mass and stiffness for mode shapes

The MNF is a binary file that combines compact storage with efficiency of data access. The MNF is also platform independent, and you can use it on any platform on which Adams products run. In general, the MNF contains a large amount of data that Adams needs to access frequently. Therefore, you should not access the MNF across a network if you want maximum performance. In addition, you can obtain a platform-dependent MNF, which can improve flexible body animation performance in the Adams products.

To create an MNF, you or an FE expert write the data from an FEA program directly to an MNF or use translators available for the most popular FEA programs to translate the data from the FEA program to an MNF. For information on generating MNFs from an FEA program, see Translating FE Model Data. Note that once you've created an MNF, you can build it into Adams without having to access the FEA program from which it was generated.

About the MD DB

With this technique, products share the same database for data storage and transfer. Users can instruct Nastran to create an MD DB that can be used by Adams to create flexible bodies. The following are highlights of this new feature:

1. Multiple flexible bodies can be stored in one single database.
2. One database file is needed: .MASTER.
3. When multiple bodies are present in the database, the user needs an additional parameter, INDEX, to create a flexible body. The user can browse the contents of a database and get the index of a body through Adams Flex Toolkit.
4. It takes advantage of sophisticated Nastran Database access technology.
5. The database is platform-dependent.

Steps in Modeling Flexible Bodies

The steps that you perform to build a flexible body into Adams View and run Simulations are shown in the figure below.

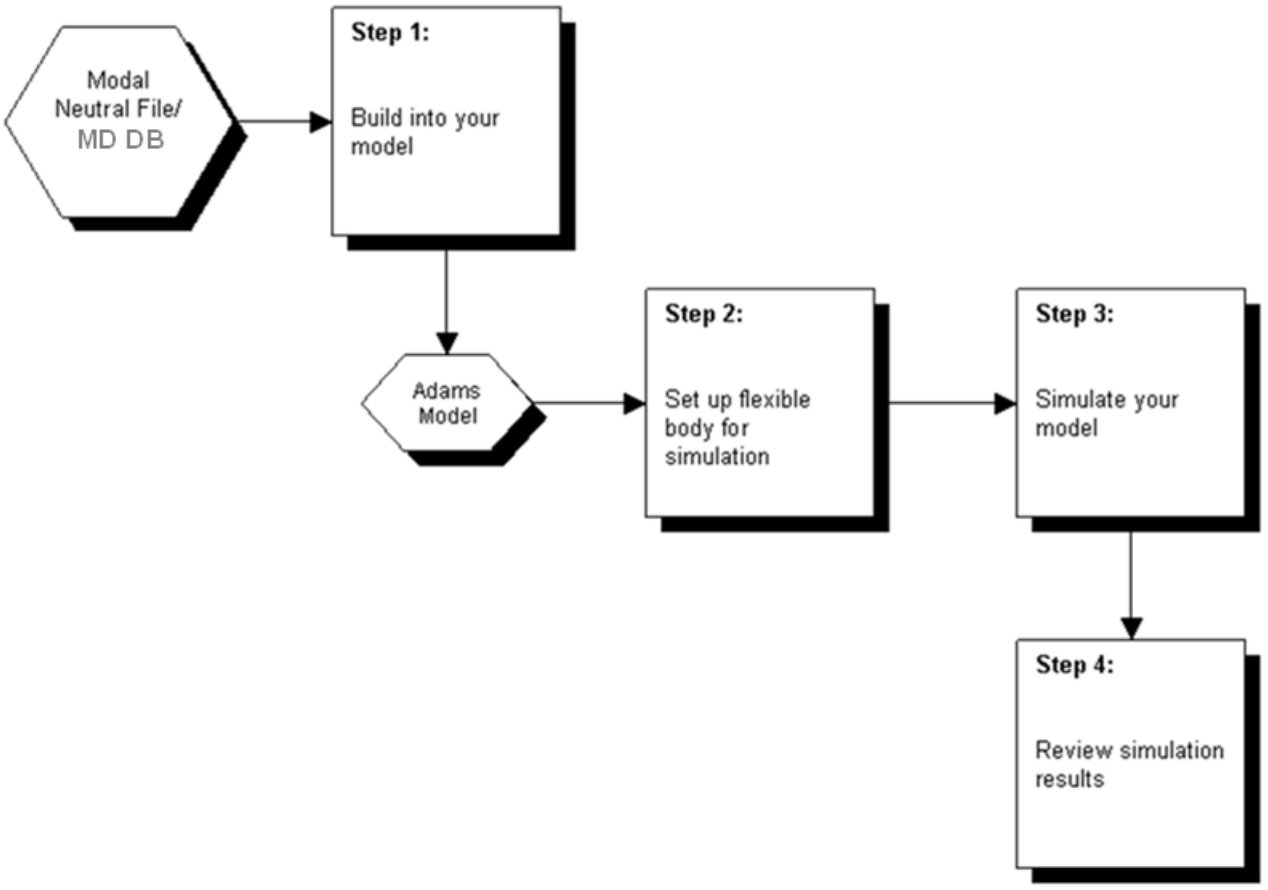


Figure 1 Adams Flex Steps

Note:	When creating flexible bodies using Nastran MD DB files, Adams will create a temporary .**MASTER** file on disk, with a name like **db<number>.MASTER**. Users should not attempt to use this file directly.

Building Flex Body Models

Creating a Flexible Body in Adams View

When you create a flexible body in Adams View, Adams View reads the Modal Neutral File (MNF) or MD DB defining the body and creates the necessary Adams View geometry for displaying the flexible body. It also creates a mesh on the flexible body representing the flexible body elements and nodes. You can simplify the mesh geometry as explained in Substituting Outline Graphics for the Finite Element Mesh.

Adams Flex places the flexible body so the flexible body's local body reference frame (LBRF) is at the origin of the global coordinate system. The LBRF corresponds to the origin of the FE environment in which the body was originally modeled. You can also set the location of the flexible body as you create the body using options in the dialog box or move the body later as you do any element in Adams View using the Object Manipulation Strip on the Main Menu. Learn more about Moving Objects Using the Move Tools.

After you import an MNF or MD DB into Adams View, you can use the **Info** tool to examine pertinent data about the flexible body. (Learn about the Info tool.)

To import an MNF or MD DB into Adams View:

1. Click the **Bodies** tab.
2. From the **Flexible Bodies** container, select **Adams Flex**  .

   The Create a Flexible Body dialog box appears.
3. In the **Flexible Body Name** text box, enter the name you want assigned to the flexible body.
4. First select whether you want to create flexible body from MNF or MD DB from the dropdown box. Tips on Entering File Names in Text Boxes.

- MNF: In the text box, enter the name of the MNF or browse for one.

> **Note:** When you use the Browse command to search for a file, it places the absolute path to the file in the text box. When you save the database or a command file, Adams View uses the absolute path in the saved file. If you are sharing the database or command file with other users, you may want to edit the path that the Browse command places in the text box so it is a relative path or remove it altogether if the MNF file is in the current working directory.

- MD DB: In the **Database File Name** text box, enter the name of the MD DB file name (*.MASTER) or browse for one. Then click the button right to the "Index" textbox. A dialog will popup to show you all the flexible bodies in the database and their information string. Please select the one you want to use. The index of the body you selected will be display in the **index** text box.

5. To set the damping ratio to one other than the default:

- Clear the selection of **default**
- In the **Damping Ratio** text box, either:
  - Enter the critical damping ratio.
  - Enter a function. To get help building the function, next to the **Damping Ratio** text box, select the **More** button  . The Adams View Function Builder appears. For information on using the Function Builder, see the Adams View Function Builder online help.

Learn more about specifying damping ratios, including learning the defaults.

6. Set **Generalized Damping to one of the following**:
   - **Off** - Disables the generalized damping. - Disables the generalized damping.
   - **Full** - Enables the complete generalized damping matrix, including the effects of a resultant damping force.
   - **Internal Only** - Only enables the portion of the generalized damping matrix corresponding to the modal coordinates (that is, ignore the resultant damping force).

   Learn more about specifying damping.
7. Position the flexible body by doing one or more of the following:
   - To set its location, in the **Location** text box, enter the x, y, z coordinate defining the flexible body's location in the default coordinate system.
   - To set its orientation, specify either of these three orientation methods:
     - Orientation
     - Along Axis Orientation
     - In Plane Orientation
   - You can also set its orientation relative to a reference coordinate system by entering the coordinate system in the **Relative to** text box. If you leave it blank, Adams View uses the default coordinate system
8. Select **OK**.

Creating a Flexible Body in Adams View (Classic Interface)

When you create a flexible body in Adams View, Adams View reads the Modal Neutral File (MNF) or MD DB defining the body and creates the necessary Adams View geometry for displaying the flexible body. It also creates a mesh on the flexible body representing the flexible body elements and nodes. You can simplify the mesh geometry as explained in Substituting Outline Graphics for the Finite Element Mesh.

Adams Flex places the flexible body so the flexible body's local body reference frame (LBRF) is at the origin of the global coordinate system. The LBRF corresponds to the origin of the FE environment in which the body was originally modeled. You can also set the location of the flexible body as you create the body using options in the dialog box or move the body later as you do any element in Adams View using the Move Toolstack on the Main toolbox. Learn more about Moving Objects Using the Move Tools (Classic Interface).

After you import an MNF or MD DB into Adams View, you can use the **Info** tool to examine pertinent data about the flexible body. (Learn about the Info tool.)

To import an MNF or MD DB into Adams View:

1. From the **Build** menu, point to **Flexible Bodies**, and then select **Adams Flex**.

   The Create a Flexible Body dialog box appears.
2. In the **Flexible Body Name** text box, enter the name you want assigned to the flexible body.
3. First select whether you want to create flexible body from MNF or MD DB from the dropdown box. Tips on Entering File Names in Text Boxes.

- MNF: In the text box, enter the name of the MNF or browse for one.

> **Note:** When you use the Browse command to search for a file, it places the absolute path to the file in the text box. When you save the database or a command file, Adams View uses the absolute path in the saved file. If you are sharing the database or command file with other users, you may want to edit the path that the Browse command places in the text box so it is a relative path or remove it altogether if the MNF file is in the current working directory.

- MD DB: In the **Database File Name** text box, enter the name of the MD DB file name (*.MASTER) or browse for one. Then click the  button right to the "Index" textbox. A dialog will popup to show you all the flexible bodies in the database and their information string. Please select the one you want to use. The index of the body you selected will be display in the **index** text box.

4. To set the damping ratio to one other than the default:

- Clear the selection of **default**
- In the **Damping Ratio** text box, either:
  - Enter the critical damping ratio.
  - Enter a function. To get help building the function, next to the **Damping Ratio** text box, select the **More** button  . The Adams View Function Builder appears. For information on using the Function Builder, see the Adams View Function Builder online help.

Learn more about specifying damping ratios, including learning the defaults.

5. Set **Generalized Damping to one of the following**:
   - **Off** - Disables the generalized damping. - Disables the generalized damping.
   - **Full** - Enables the complete generalized damping matrix, including the effects of a resultant damping force.

- **Internal Only** - Only enables the portion of the generalized damping matrix corresponding to the modal coordinates (that is, ignore the resultant damping force).

Learn more about specifying damping.

6. Position the flexible body by doing one or more of the following:
   - To set its location, in the **Location** text box, enter the x, y, z coordinate defining the flexible body's location in the default coordinate system.
   - To set its orientation, specify either of these three orientation methods:
     - Orientation
     - Along Axis Orientation
     - In Plane Orientation
   - You can also set its orientation relative to a reference coordinate system by entering the coordinate system in the **Relative to** text box. If you leave it blank, Adams View uses the default coordinate system
7. Select **OK**.

Connecting Flexible Bodies to Your Model

Once you have your flexible body imported into Adams View, you need to make it part of your model by attaching it to rigid bodies or other flexible bodies and applying forces to it as appropriate. You can also replace existing rigid bodies with flexible bodies.

Learn more about how to make flexible bodies essential components of your model:

- About Connecting Flexible Bodies
- Limitations on Applying Elements to Flexible Bodies
- Using Massless Links
- Adding Markers to Flexible Bodies

About Connecting Flexible Bodies

When you initially create a flexible body from a Modal Neutral File (MNF) or an MD DB file (*.MASTER), Adams Flex places the flexible body at the global coordinate system origin. The flexible body is not attached to any other part in your model nor are its movements constrained.

Using any of the features of Adams View, you can connect the flexible body to elements in your model, apply forces to it, and attach points and markers to it. There are some limitations, however, and these are explained in Limitations on Applying Elements to Flexible Bodies.

As you connect flexible and rigid bodies, you will often find it useful to create dummy parts, called massless links, to help you create your model. Also, the connection of some modeling elements is only supported through massless links so Adams View often creates massless links between the bodies to facilitate the creation of the model. For more information, see Using Massless Links.

If you haven't already done so, you may want to go through the first tutorial, Integrating Flexible Bodies into Your Adams Model, of Getting Started Using Adams Flex, to get some ideas for how to connect flexible and rigid bodies.

Limitations on Applying Elements to Flexible Bodies

Currently, there are a few limitations to the elements you can use with flexible bodies. Most of these limitations are based on the version of Adams Solver that you are using, either the FORTRAN or C++ version. Most of the limitations are removed if you use Adams Solver (C++) as the solver executable. Learn more about Solver versions.

Learn more about the limitations:

- Flexible Connector Limitations in Adams Solver (FORTRAN)
- Force Limitations in Adams Solver (FORTRAN)
- Joint Limitations in Adams Solver (FORTRAN)

Table of Supported Forces and Joints

Flexible Connector Limitations in Adams Solver (FORTRAN)

The following Adams View flexible connectors have not been implemented for use with flexible bodies in Adams Solver (FORTRAN):

- Beams
- Bushings
- Field elements
- Torsion spring (Rotational spring-damper elements)

In each case, the flexible connectors can indirectly act on the flexible body through a dummy part, called a massless link. If one of these elements is connected directly to the flexible body, Adams Solver (FORTRAN) automatically introduces a massless link for you. Learn about Using Massless Links.

Force Limitations in Adams Solver (FORTRAN)

Like the rotational spring damper, you cannot directly attach a rotational Single-component force to a flexible body in Adams Solver (FORTRAN). You need to use a masslink to connect the force to the flexible body. Learn about using massless links.

You cannot apply the following forces to a flexible body when the flexible body is the reaction body:

- Six-component general force vector
- Three-component force vector
- Three-component torque vector

Again, you can avoid this limitation by attaching massless links to your flexible body, which makes the massless link the reaction body. It is important, however, to understand the consequences of making the massless link the reaction body. For example, consider a rigid sphere rolling on a flexible surface. Ideally, you would model the contact using a force vector element with the I marker on the bottom of the sphere and a

floating marker on the flexible surface. Because a point of contact on a flexible body must be a set of nodes determined a priori, you cannot define a floating marker on a flexible body. Therefore, the flexible body cannot be the reaction body.

As a workaround, assume that you attach a massless link on some remote location on the flexible surface and define the floating marker on the massless link. Adams Solver would simulate the model without difficulty, but the effect you modeled is now different from the one you intended to model. Rather than modeling a point force sliding over the flexible surface, you modeled a varying point torque acting on the flexible surface at the location of the massless link. For this reason, Adams Solver does not automatically introduce a massless link like it does when attaching an unsupported force to a flexible body.

When the flexible body has a large number of modes (more than 40), the computational cost can be considerable. In these cases, you will find it more expedient to attach a massless link at the location of the flexible body marker so that the function expression depends on part states rather than the higher number of flexible body states.

You should also note that whenever a function expression associated with a force element, including those listed above and single-component forces, refers to a marker on a flexible body, it establishes a functional dependency on all the state variables of the flexible body. For example, if you use a DX function to define the force on the flexible body, you would establish the dependency.

Joint Limitations in Adams Solver (FORTRAN)

You cannot directly connect the following joints to a flexible body when using Adams Solver (FORTRAN):

- Joints, such as the translational joint or an inplane joint primitive, because the point of contact on a flexible body must be well defined. You can eliminate the joint limitation by attaching the joint to a massless link, the same as you would when applying vector forces. You will also, however, have the same consequences. For more information on using massless links with forces, see Force Limitations in Adams Solver (FORTRAN). If you do not use a massless link and attach either a translational or inplane joint primitive directly to your flexible body, Adams Solver (FORTRAN) automatically inserts one for you.
- Joints that are being driven by motion generators. For example, a revolute joint can be attached to a flexible body when no motion generator is attached to it but not when a motion generator is acting on the revolute joint. Again, to circumvent this limitation, Adams Solver (FORTRAN) introduces a massless link between the joint and flexible body.

Supported Forces and Joints

The following tables list the forces and joints that you can apply directly to flexible bodies.

For the features that you cannot directly apply to flexible bodies, see the workarounds listed in Limitations on Applying Elements to Flexible Bodies.

The force:	Is supported in Adams Solver (FORTRAN)	Is supported in Adams Solver (C++)
ACCGRAV	Yes	Yes
BEAM	No	Yes
BUSHING	No	Yes
CONTACT	Yes*	Yes*
FIELD	No	Yes
FRICTION	SPHERICAL, CYLINDRICAL, UNIVERSAL, HOOKE, REVOLUTE	SPHERICAL, CYLIDRICAL, UNIVERSAL, HOOKE, TRANSLATION, REVOLUTE
GFORCE	Yes**	Yes
MFORCE	Yes	Yes
NFORCE	Yes	No
TRANSLATIONAL SFORCE	Yes	Yes
ROTATIONAL SFORCE	No	Yes
TRANSLATIONAL SPRINGDAMPER	Yes	Yes
ROTATIONAL SPRINGDAMPER	No	Yes
VFORCE	Yes**	Yes
VTORQUE	Yes**	Yes

* For Adams Solver (FORTRAN), only point-to-plane and point-to-curve contacts are supported, where the point is on the flexible body. Adams Solver (FORTRAN) can only treat one point per CONTACT statement. Adams Solver (C++) can treat multiple points per CONTACT statement. From Adams MD R3, general Flexible to Flexible, Flexible to Rigid Contact is supported in Adams Solver (C++).

** For Adams Solver (FORTRAN) the floating marker cannot be on a flexible body, and the reaction force cannot act on a flexible body. There are no restrictions for Adams Solver (C++).

Joints Supported

The joint:	Is supported in Adams Solver (FORTRAN)	Is supported in Adams Solver (C++)
CONVEL	No	Yes
CYLINDRICAL	No	Yes
FIXED	Yes	Yes
HOOKE	Yes	Yes
PLANAR	No	Yes
RACKPIN	No	Yes
REVOLUTE	Yes	Yes
SCREW	No	Yes
SPHERICAL	Yes	Yes
TRANSLATIONAL	No	Yes
UNIVERSAL	Yes	Yes
GEAR	No	Yes
COUPLER	No	Yes
CVCV	No	No
MOTION	No	Yes
PTCV	No	No
UCON	No	No
JPRIM	ATPOINT, ORIENTATION, PARALLEL_AXES, PERPENDICULAR	ATPOINT, ORIENTATION, PARALLEL_AXES, PERPENDICULAR, INLINE, INPLANE

Using Massless Links

A massless link, also called a dummy part, is a link with zero or an insignificant amount of both mass and inertia. A massless link is not actually part of your model but facilitates the creation of a model so it behaves like your physical mechanism. It allows you to indirectly connect forces and joints to your flexible body, which are not supported.

You can either:

- Let Adams Solver create a massless link for you
- Create one yourself

Automatic Massless Link Creation in Adams Solver (FORTRAN)

When Adams Solver (FORTRAN) detects either:

- An unsupported force or joint on a flexible body (see Limitations on Applying Elements to Flexible Bodies)
- A marker on the flexible body, which is not coincident with a node (see Configuration 2: Position and Attachment Nodes are Not Coincident)

It automatically introduces a massless link between the unsupported element and the flexible body. The massless link is attached to the flexible body with a fixed joint. Neither the kinematic results of the massless link or the reaction forces at the joint will be available.

Creating Massless Links Manually

To create a massless link, you can simply remove the geometry of a link since, by default, Adams View determines the mass of a part using its geometry. If you want to keep the link geometry in your model display to help remind you that there is a massless link in your model, you may want to edit the link to set its mass and inertia properties to zero.

Note:	A massless link cannot contribute Degrees of freedom to your model. Therefore, a massless link should always be connected to another mass-bearing element using a fixed joint.

To create a massless link:

1. Create a link between parts, as desired.
2. Do one of the following:
   - Delete the link's geometry.
   - Display the link's Modify Body dialog box. Set **Category** to **Mass Properties** and set **Define Mass by** to **User Input**. In the **Mass** and **Moments of inertia** text boxes, enter **0**.
3. Select **OK**.

Adding Markers to Flexible Bodies

You can add markers to your flexible body in the same way you add them to parts so that you can apply operating loads, attach other Adams elements, or measure a response. Adding markers to flexible bodies requires specifying both the location of the marker, as well as the nodes to which it is attached. This is a fundamental difference between specifying a marker on a flexible body and specifying a marker on a part. Moreover, the position of a marker on a flexible body is treated independently from the nodes to which it is attached. For example, the position of the marker does not need to be coincident with a node.

Joint forces and applied loads are applied at the marker position and then distributed onto the flexible body at the attachment points you specify. How the loads are distributed depends on the marker configuration. For example, the offset behaves like rigid lever on a marker which is not coincident with its attachment node. That is, a force applied to the marker is transferred to the node as a force-moment pair.

Because the location of a marker is independent of how it is attached to the flexible body, there are three general marker configurations, which are described below. Of the three configurations, Adams Solver (FORTRAN) only directly supports markers whose position and attachment node are coincident. Adams Solver (FORTRAN) indirectly supports markers that are not coincident with their attachment node by automatically introducing a massless link (see Using Massless Links). Adams Solver (FORTRAN) does not support markers attached to multiple nodes. All marker configurations are directly supported in Adams Solver (C++). (Learn more about Adams Solver versions.)

Because Adams Solver (FORTRAN) is the default solver for Adams View, when you create a marker in Adams View, the marker is always positioned at a node. To obtain one of the other alternative configurations using Adams View, first create a marker whose position is coincident with its attachment point and then modify it.

- Configuration 1: Position and Attachment Node are Coincident
- Configuration 2: Position and Attachment Node are not Coincident
- Configuration 3: Attached to Multiple Nodes

Configuration 1: Position and Attachment Node are Coincident

This is the default marker configuration that Adams View creates and the only one that Adams Solver (FORTRAN) supports. This configuration is suitable for creating measures and applying concentrated point loads at the attachment node.

To create a marker whose position and attachment node are coincident:

1. Click the **Bodies** tab. From the **Construction** container, select **Marker tool**  .

   (Classic Interface) From the Geometric Modeling tool stack of the Main toolbox, select the **Marker** tool  .

2. In the marker construction container, select **Add to Part**.
3. Following the instructions in the status bar, select the flexible body to which the marker will be attached. Then, click the node closest to the position you want to place node.

   Adams View positions the marker at the closest node and attaches the marker to the same node.

Configuration 2: Position and Attachment Node are not Coincident

It is often necessary to add a marker to a flexible body where there is no node. For example, you may need to connect two flexible bodies whose nodes are not coincident or apply a force at a location offset from a node.

A similar marker configuration which is compatible with the Adams Solver (FORTRAN) can be obtained using massless links. The massless link is attached to the flexible body at the desired attachment node using a fix joint. Subsequently, a marker is attached to the massless link at the desired position. For more information about massless links, see Using Massless Links.

To create a marker whose position is offset from its attachment node:

1. First create a marker which is positioned at the node you wish to attach it to following the steps in Configuration 1: Position and Attachment Node are Coincident.

   Now modify the marker.

2. Right-click the marker you just created, and then select **Modify**.

   The Marker Modify dialog box appears.

3. In the **Location** text box, enter the desired position of the marker. You can either type in the coordinates directly; or right-click the **Location** text box, select **Pick Location**, and then click the desired location with the mouse.

4. Select **OK**.

   Adams View issues a warning indicating that the model is only compatible with the Adams Solver (C++). In addition, a white leader line from the attachment node to the marker icon appears.

To create a marker whose position is offset from its attachment node using massless links:

1. Create a massless link near where you want to position the marker. For more information about massless links, see Using Massless Links.

2. Fix the massless link to the desired attachment point by first selecting, **Connectors** tab → **Joints** container → **Fixed Joint** tool 

   (Classic Interface) Fix the massless link to the desired attachment point by first selecting the **Fixed Joint** tool  from the Joint tool stack on the Main toolbox.

3. In the construction container, select the method **2 Body 1 Location**.

4. Following the instructions in the status bar, select the massless link you just created and then the flexible body to which you want to attach it. Finally, select the node to which you want to attach the massless link.

   Adams View creates a fix joint between the massless link and flexible body.

5. Create a marker on the massless link at the desired position by first selecting, **Bodies** tab → **Construction** container → **Marker tool**  .

   (Classic Interface) Create a marker on the massless link at the desired position by first selecting the **Marker** tool  from the **Geometric Modeling** tool stack on the Main toolbox.

6. In the marker construction container, select **Add to Part**.

7. Following the instructions in the status bar, select the massless link, and then select the desired location.

Adams View creates a marker on the massless link. The behavior of this marker is identical to one which was added directly to the flexible body but offset from its attachment node.

Configuration 3: Attached to Multiple Nodes

Point forces, applied force acting at single point on a flexible body, are idealized representations of concentrated distributed forces. Point forces typically give rise to large stress gradients that may not be present in the physical structure. A more meaningful result is often obtained by distributing the force over a small patch on the flexible body. This can be accomplished by introducing a rigid element spider web in the finite element discretization or introducing a marker that is attached to multiple nodes. A force applied to the marker is then distributed to the attachment nodes.

To create a marker attached to multiple nodes:

1. First create a marker which is positioned at one of the nodes you wish to attach it to following the steps in Configuration 1: Position and Attachment Node are Coincident.

   Now modify the marker.

2. Right-click the marker you just created, and then select **Modify.**

   The Marker Modify dialog box appears.

3. In the **Location** text box, enter the desired position of the marker.

   You can either type in the coordinates directly; or right-click the Location text box, select Pick Location, and then click the desired location with the mouse.

4. In the text box to the right of **Snap**, enter a comma-separated list of attachment nodes, or right-click the text box, select **Pick FlexBody Node**, and then click the desired attachment nodes with the mouse.

5. Select **OK**.

   Adams View issues a warning indicating that the model is only compatible with Adams Solver (C++). In addition, white leader lines from all the attachment nodes to the marker icon appear.

Replacing Existing Bodies with Flexible Bodies

It is easy to replace a rigid body with a flexible body to more realistically model your mechanism. Because it is easy to replace rigid bodies with other flexible bodies, you can start gradually and slowly add complexity to your model as you need it. You may want to go through the tutorial, Exchanging a Rigid Body with a Flexible Body, or Getting Started Using Adams Flex, to step through the process of replacing a rigid body with a flexible body.

In addition to replacing a rigid body with a flexible body, you can replace an existing flexible body with a new flexible body. This may be useful if you want to modify the design of your flexible component.

Learn more about replacing rigid and flexible bodies:

Overview of Replacing Rigid and Flexible Bodies

- About Replacing Bodies with Flexible Bodies
- Overall Procedure for Replacing Rigid Bodies with Flexible Bodies

- Overall Procedure for Replacing Flexible Bodies with Flexible Bodies
- Marker and Node Table
- Attributes Inherited
- Substitution Limitations
- Handling of Expressions and Design Variables

Detailed Procedures

- Selecting the Existing and Replacement Bodies
- Aligning Replacement Body
- Transferring Connections
- Replacing the Existing Body
- Sorting Objects in Marker and Node Table

About Replacing Bodies with Flexible Bodies

To replace a rigid or flexible body with a flexible body, you start by specifying:

- The rigid or flexible body to be substituted and then the replacement flexible body. When replacing rigid bodies, the replacement flexible bodies can either be an existing flexible body or a Modal Neutral File (MNF) or MD DB. When replacing flexible bodies, the replacement flexible bodies must be specified as an MNF or MD DB.
- The position and orientation of the new flexible body. You specify the position and orientation of the replacement flexible body using one of three convenient alignment tools.

Adams Flex then identifies the connections on the existing body to other objects in the model, such as motions, forces, and joints. It displays the markers on the existing body through which the connections are made. Because the connections to flexible bodies are through markers, you will transfer the markers to the flexible body, thereby transferring the connections.

- If the existing body is a rigid body, Adams Flex searches for the markers on the rigid body, and transfers these markers to the nodes on the replacement flexible body that are closest to the markers. If a marker is used to define joints or forces, Adams Flex searches for interface nodes. For all other markers, it searches the entire set of nodes.
- If the existing body is another flexible body, the markers are already attached to nodes. Therefore, Adams Flex searches for the node closest to the attached node on the existing body. If the marker is attached to multiple nodes, Adams Flex searches for the closest node for each connection. Again, if a marker is used to define joints or forces, Adams Flex searches for interface nodes.

Adams Flex displays a Marker and Node Table to let you make any changes, such as change the node to which a marker will be moved.

Adams Flex moves all markers to the replacement flexible body even if the markers have no connections. This is because you may have User-written subroutines or Measures that reference the markers. By transferring all markers to flexible bodies, you ensure that all your subroutines and measures work. You can delete the markers if you want.

Because Adams Flex moves all markers, it also moves the center of mass (CM) marker. Once the marker is on the flexible body, however, it no longer indicates the CM nor does it stay coincident with the CM during a simulation. To eliminate confusion, Adams Flex renames the CM marker from the substituted rigid body to original_rigid_cm.

Note:	Adams Flex only transfers markers and any connections they define to the flexible body. It does not transfer string variables and other children of the rigid body.

Overall Procedure for Replacing Rigid Bodies with Flexible Bodies

To replace a rigid body with a flexible body, perform the following steps:

- Select the rigid body to be replaced
- Position and orient the replacement body
- Transfer the connections
- Replace the rigid body

Each step is briefly outlined below with links to more complete procedures.

Selecting the Bodies to be Replaced

You can replace a rigid body with a flexible body that is already in your model or you can specify an MNF or MD DB and Adams Flex creates a new flexible body. The following explains how to specify a flexible body already in your model. Learn more about selecting the body to be replaced.

To select the bodies:

1. On the screen, select the rigid body to be replaced.
2. Click the **Bodies** tab. From the **Flexible Bodies** container, select **Rigid to Flex**  .

   (Classic Interface) From the **Build** menu, point to **Flexible Bodies**, and then select **Rigid to Flex**.
3. Under the **Alignment** tab, select **Flex Body**, and then select a flexible body that has already been imported into your model.

Position and Orient Replacement Body

You use the positioning options in the **Alignment** tab of the Swap rigid/flexible body dialog box to move the flexible body so that it is aligned with the existing body that it is replacing. Three alignment tools are available:

- Aligning center of mass of bodies
- Aligning using the Precision Move dialog box
- Aligning by defining three points

You can use one or more of the tools to position and orient the flexible body consistently with the rigid body. Once you've used these tools, you can use the Flip about buttons to rotate the body 180° into position.

Learn about the tools.

Transferring the Connections

You use the **Connections** tab on the Swap rigid body dialog box to display and transfer the connections on the rigid body to the replacement flexible body. The Connections tab displays a Marker and Node table listing the connections and how they will be transferred. It also provides options for positioning the markers relative to the attachment nodes on the replacement flexible body. You can also launch a Node Finder Dialog Box to search for a different node.

To transfer the connections:

1. Review the list of marker connections and how Adams Flex will transfer them to the flexible body.
2. To change the node to which the marker will be transferred, enter the new node ID in the **Node ID** text box, and then select **Apply**. Learn more.
3. If the attachment nodes are offset from the marker locations, you have three options:
   - Do nothing, and simply select **OK** as explained in the next procedure. In which case, the Adams Solver (C++) creates a marker with an offset, while Adams Solver (FORTRAN) automatically introduces a massless part and fixed joint pair. You are not responsible for creating dummy parts.
   - Move the marker to node location. Learn about this method.
   - Maintain any parameterization that was set up through Adams View. Learn about this method.

Replacing the Rigid Body

Once you've set the alignment and connections, you can replace the existing body with the replacement flexible body. Learn about the complete procedure.

To replace the body:

- Select **OK**.

Note:	Adams Flex only transfers markers and any connections they define to the flexible body. It does not transfer string variables and other children of the rigid body.

Overall Procedure for Replacing Flexible Bodies with Flexible Bodies

To replace a flexible body with a flexible body, perform the following steps:

- Select the flexible bodies to be replaced
- Align them
- Transfer the connections
- Replace the flexible body

Each step is briefly outlined below with links to more complete procedures.

Selecting the Bodies to be Replaced

To replace a flexible body with a flexible body, you must create the replacement flexible body by specifying an MNF an MD DB. Learn more about selecting the body to be replaced.

To select the flexible bodies:

1. Click the **Bodies** tab. From the **Flexible Bodies** container, select **Flex to Flex**  .

   (Classic Interface) From the **Build** menu, point to **Flexible Bodies**, and then select **Flex to Flex**.
2. Under the **Alignment** tab, in the **Flexible Body** text box, enter an existing flexible body in your model.
3. First select whether you want to use MNF file or MD DB file from the dropdown box.
4. Enter the MNF file or MD DB file name in the text box or browsing for one.
5. If you select MD DB file, click  button right to the "Index in DB" text box to bring out a list of flexible bodies in the DB file. Then select one, the index of the selected body will be input in the "Index in DB" textbox.

Aligning the Bodies

You use the positioning options in the **Alignment** tab of the Swap rigid body dialog box to move the flexible body so that it is aligned with the existing flexible body. Three alignment tools are available:

- Aligning center of mass of bodies
- Aligning using the Precision Move dialog box
- Aligning by defining three points

You can use one or more of the tools to position and orient the flexible body consistently with the rigid body. Once you've used these tools, you can use the Flip about buttons to rotate the body 180° into position.

Learn about the tools.

Transferring the Connections

You use the **Connections** tab on the Swap rigid body dialog box to display and transfer the connections on the existing flexible body to the replacement flexible body. The Connections tab displays a Marker and Node table listing the connections and how they will be transferred. It also provides options for positioning the markers relative to the attachment nodes on the replacement flexible body.

1. Review the list of marker connections and how Adams Flex will transfer them to the flexible body.

2. To change the node to which the marker will be transferred, enter the new node ID in the **Node ID** text box, and then select **Apply**. Learn more.
3. If the attachment nodes are offset from the marker locations, you have three options:
   - Do nothing, and simply select **OK** as explained in the next procedure. In which case, the Adams Solver (C++) creates a marker with an offset while Adams Solver (FORTRAN) automatically introduces a massless part and fixed joint pair. You are not responsible for creating dummy parts.
   - Move the marker to node location. Learn about this method.
   - Maintain any parameterization that was set up through Adams View. Learn about this method.

Replacing the Body

Once you've set the alignment and connections, you can replace the existing body with the replacement flexible body.

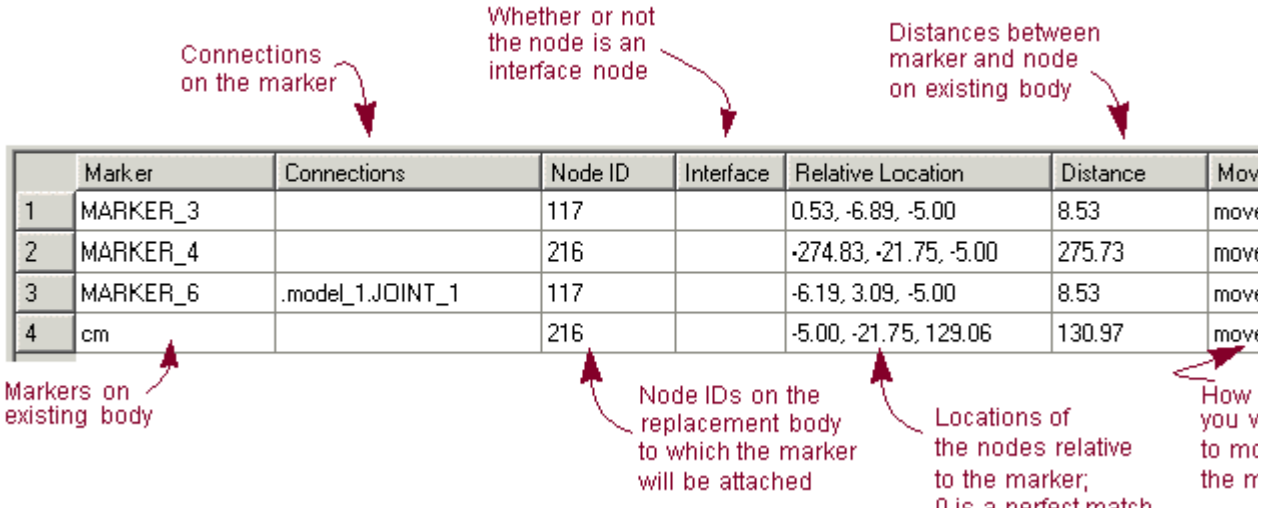
To replace the body:

- Select **OK**.

Marker and Node Table

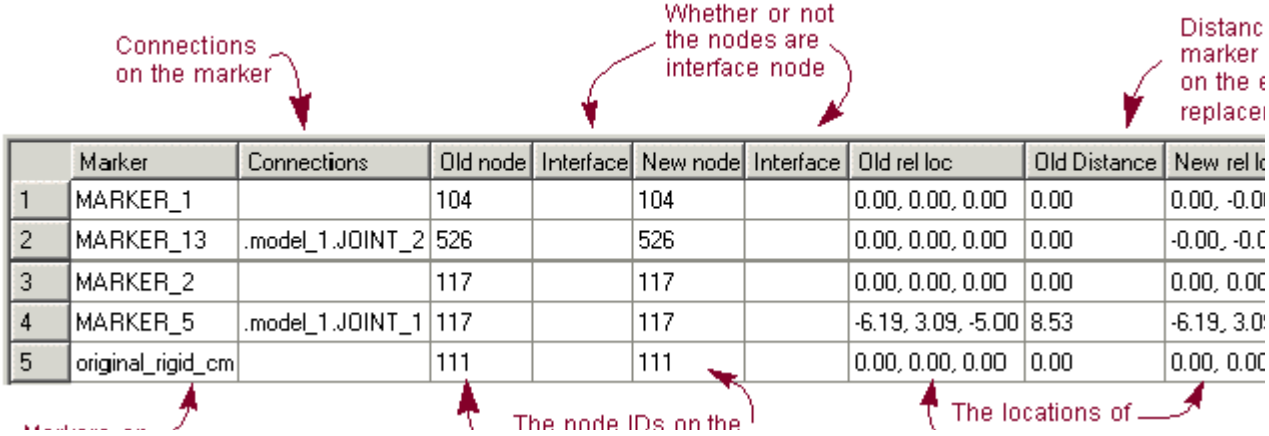
The Connections tab on the Swap a rigid body for another flexible body or Swap a flexible body for another flexible body dialog boxes displays a Marker and Node table. It displays a list of markers on the rigid body that will be transferred to the flexible body. The Connections column identifies the names of constraints or forces that exist on the rigid body that you are replacing. The nodes on the flexible body that are nearest to the markers are listed in the Node ID column.

The following figures show the Marker and Node table as it appears when you are replacing a rigid and flexible body.



	Marker	Connections	Node ID	Interface	Relative Location	Distance	Mov
1	MARKER_3		117		0.53, -6.89, -5.00	8.53	move
2	MARKER_4		216		-274.83, -21.75, -5.00	275.73	move
3	MARKER_6	.model_1.JOINT_1	117		-6.19, 3.09, -5.00	8.53	move
4	cm		216		-5.00, -21.75, 129.06	130.97	move

Marker and Node Table when Replacing Rigid Body



	Marker	Connections	Old node	Interface	New node	Interface	Old rel loc	Old Distance	New rel lo
1	MARKER_1		104		104		0.00, 0.00, 0.00	0.00	0.00, -0.0
2	MARKER_13	.model_1.JOINT_2	526		526		0.00, 0.00, 0.00	0.00	-0.00, -0.0
3	MARKER_2		117		117		0.00, 0.00, 0.00	0.00	0.00, 0.00
4	MARKER_5	.model_1.JOINT_1	117		117		-6.19, 3.09, -5.00	8.53	-6.19, 3.0
5	original_rigid_cm		111		111		0.00, 0.00, 0.00	0.00	0.00, 0.00

Marker and Node Table When Replacing Flexible Body

Attributes Inherited

When you substitute either a rigid or flexible body for a flexible body, the new flexible body takes on the following attributes of the rigid or flexible body that it replaced:

- Color
- Visibility
- Name visibility

- Icon size
- Exact position initial conditions
- Velocity initial conditions

If you replace a flexible body with another flexible body, the new flexible body also receives the attributes of:

- Selected modes
- Invariant selection
- Modal displacement and velocity initial conditions
- Modal exact displacements

Substitution Limitations

Note the following before you substitute rigid and flexible bodies:

- The closer the geometry of the flexible body matches that of the rigid body, the easier it is to substitute the flexible body for the rigid body.
- When you modify a joint so that it connects to a flexible body instead of a rigid body, Adams View moves the markers defining the attachments to the flexible body at the closest node locations. If the markers are being used by other elements in your model, Adams View leaves the markers at their current locations and creates new markers that connect the joints to the flexible body at the node locations.

  As Adams View moves the markers, the joint may appear to be broken. When Adams View simulates the model, its first step will be to repair any broken joints. This may require Adams View to contort the flexible body, and possibly making the simulation fail. You should then move the marker that is not on the flexible body so it coincides with the marker on the flexible body. If this is not possible, bridge the offset between the two markers by creating a massless link that connects the parts (see Using Massless Links) or use an offset marker (see Adding Markers to Flexible Bodies).
- After substituting a body with a (different) flexible body, if one or more analyses which reference the original representation of this body exists in the session, the body will not appear correctly in subsequent animations of those analyses. See Animating After a Flexible Body Swap for details.

Handling of Expressions and Design Variables

When replacing a rigid body with a flexible body, the markers on the rigid body are transferred to the flexible body. By moving the markers, all joints and applied forces on the rigid body will also be moved to the flexible body.

When replacing a rigid body with a flexible body, special treatment is given to the following:

- The center of mass marker on the rigid body is moved to the flexible body. Its position on the flexible body, however, may not be consistent with the flexible body center of mass. The rigid body center of mass marker is also renamed after being transferred to the flexible body.
- Other than the markers, all other children of the rigid body are not transferred to the flexible body. For example, geometry and design points remain on the rigid body.

- When an expression references a rigid part, it automatically will be updated to reference the new flexible body.
- Object type design variables not belonging to the rigid body, but referencing it, will be updated to reference the flexible body. All design variables that are children of the rigid body will not be modified.

Selecting the Existing and Replacement Bodies

You can replace a rigid body with a flexible body that is already in your model or you can specify an MNF or an MD DB to let Adams Flex creates a new flexible body. To replace a flexible body with another flexible body, you must create the replacement flexible body by specifying an MNF or an MD DB.

Learn about:

- Selecting Existing Rigid and Replacement Flexible Bodies
- Selecting Existing and Replacement Flexible Bodies

Selecting Existing Rigid and Replacement Flexible Bodies

To select the bodies:

1. Click the **Bodies** tab. From the **Flexible Bodies**, select **Rigid to Flex**  .

   (Classic Interface) From the **Build** menu, point to **Flexible Bodies**, and then select **Rigid to Flex**.

   If a rigid body was not selected on the screen, the Database Navigator appears and lets you select the rigid body to be replaced. Learn about the Database Navigator.

   The Swap a rigid body for another flexible body dialog box appears.
2. Under the Alignment tab, select how you will specify the replacement flexible body:
   - **Flex Body**, and then select a flexible body that has already been imported into your model. Tips on Entering Object Names in Text Boxes.
   - **MNF File**, and then select the name of the MNF. Tips on Entering File Names in Text Boxes.
   - **MD DB**, select the name of the MD DB, then select the index of the flexible body in DB.

Selecting Existing and Replacement Flexible Bodies

To replace a flexible body with another flexible body, you must create the replacement flexible body by specifying an MNF.

To select the flexible bodies:

1. Click the **Bodies** tab. From the **Flexible Bodies**, select **Flex to Flex**  .

   (Classic Interface) From the **Build** menu, point to **Flexible Bodies**, and then select **Flex to Flex**.

The Swap a flexible body for another flexible body dialog box appears.

2. Under the **Alignment** tab, in the **Flexible Body** text box, enter an existing flexible body in your model.
3. First select **MNF** or **MD DB** in the dropdown box. Then specify the file name in the text box. If you selected an MD DB, you need to specify the Index as well.

Aligning Replacement Body

When replacing rigid or flexible bodies with a replacement flexible body, you have the following options available in the Alignment tab of the Swap a rigid body for another flexible body or Swap a flexible body for another flexible body dialog box for aligning the replacement body with the existing body:

- Aligning center of mass of bodies
- Aligning using the Precision Move dialog box
- Aligning by defining three points

Tip:	To facilitate the alignment, you can select either of the following from the bottom of the Alignment tab:

- Select **View Parts only** to display only the existing body and its replacement.
- Select **View Topology** to display a representation of the existing body and its connections to other parts. Learn about Graphically Viewing Model Topology.

Aligning Center of Mass of Bodies

The Align Flex Body CM with CM of Current Part button aligns the replacement flexible body with an existing body by comparing the center of mass and inertia tensor of the two bodies. The replacement flexible body is:

- Positioned such that its center of mass (CM) is coincident with the flexible body CM.
- Oriented such that its principal inertia directions are coincident with the body's principal inertia directions.

If the inertia properties of the two bodies are similar, this method closely aligns the flexible body with the existing body. If the bodies are symmetric about a plane, this method may rotate the flexible body 180° from the existing body. In this case, you can use the Flip about buttons to rotate the flexible body 180° back into position.

To align the center of mass of the replacement flexible body with that of the existing body:

- In the Alignment tab, select **Align Flex Body CM with CM of Current Part**

To rotate the flexible body about an axis:

- Select one of the following to rotate the flexible about the corresponding axis:

- **X axis** - Rotate flexible body 180° about its largest principal inertia direction
- **Y axis** - Rotate flexible body 180° about its second largest principal inertia direction
- **Z axis** - Rotate flexible body 180° about its smallest principal inertia direction

Aligning Using Precision Move Dialog Box

The Precision Move dialog box lets you to move bodies by increments or precise coordinates.

To use the Precision Move dialog box to align the bodies:

- In the Alignment tab, select **Launch Precision Move Panel**. Learn about Moving Objects Using the Precision Move Dialog Box.

Aligning by Defining Three Points

This method uses three unique points to position and orient the flexible body.

To define the alignment by specifying three points:

1. In the Alignment tab, select **3 Point Method**.
2. Select a point on the existing body and a point on the flexible body. Adams Flex will position the flexible body so that the point on it is coincident with the point on the rigid body.
3. Now select two more point pairs, one point on each body, to define the x- and y-axes. Adams Flex will orient the flexible body by making points 2 and 3 on the flexible body reside in the plane defined by points 2 and 3 on the rigid body.

Transferring Connections

The following sections explain how to transfer connections from an existing rigid or flexible body to a replacement flexible body:

- Changing the Node to Which to Transfer Marker
- Changing the Location of the Marker

Changing the Node to Which to Transfer Marker

To change the node to which a marker will be transferred:

1. In the Marker and Node table of the Swap a rigid body for another flexible body or Swap a flexible body for another flexible body dialog boxes, select the row containing the marker.
2. In the **Node ID** text box, enter the node ID to which you want to transfer the marker.

Tip:	To find a node, select **Node Finder** to select the Node Finder Dialog Box, and search for all nodes within a specified radius or number of nodes from a given marker.

3. Select **Apply** (located to the right of the Node ID text box).

Changing the Location of the Marker

By default, Adams Flex moves the marker to the attachment node specified in the Marker and Node table. If the distance between the attachment node and the marker is nonzero, you may want to preserve the marker location so you do not break any joints on the flexible body or incorrectly define the point of application of a force. In addition, the marker's position may be parameterized. Moving the marker to the chosen attachment node will destroy the parameterization. To prevent this from adversely affecting the flexible body's connectivity with the rest of the model, Adams Flex provides three choices to treat the marker's position:

- Simply move the markers to the location of the specified nodes.
- Transfer the markers but maintain any parameterization that was set up through an Adams View expression to define the location of the marker. This will keep the marker position at its parameterized position.
- Transfer the marker but keep it in its current position, offset from the node. This is helpful if you have defined a joint location using the marker. It keeps the joint from breaking.

To change the location of a marker:

1. In the Marker and Node table of the Swap a rigid body or Swap a flexible body dialog boxes, select the row containing the marker.
2. Select one of:
   - **Move to node**
   - **Preserve expression**
   - **Preserve location**

Adams Flex changes the Move column to reflect the changes.

Replacing the Existing Body

Once you've set the alignment and connections, you can replace the existing body with the replacement flexible body. You can also select to keep the rigid body in the modeling database. Otherwise, Adams Flex deletes the rigid body.

To replace the body:

1. Select **Copy original part** to keep the existing body in the modeling database for later use.
2. Select **OK**.

Note:	Adams Flex only transfers markers and any connections they define to the flexible body. It does not transfer string variables and other children of the rigid body.

Sorting Objects in Marker and Node Table

To help you view the markers and their connections listed in the Marker and Node table, you can sort them based on the headings of the columns. For example, you can sort the table by the marker that is the most distant from its selected node.

To sort objects in the table:

1. In either of the following dialog boxes, select the Connections tab:
   - Swap a rigid body for another flexible body
   - Swap a flexible body for another flexible body
2. Set the **Sort by** option to one of the following:
   - **Marker** - **Sort alphabetically by marker name**.
   - **Connections** - Sort by those markers with connections.
   - **Node ID** - Sort by the ID of the node.
   - **Interface** - Sort by those nodes that are interface nodes
   - **Distance** - Sort by those markers the most distant from the selected node.
   - **Move** - Sort by the values in the Move column.

Verifying Flexible Bodies

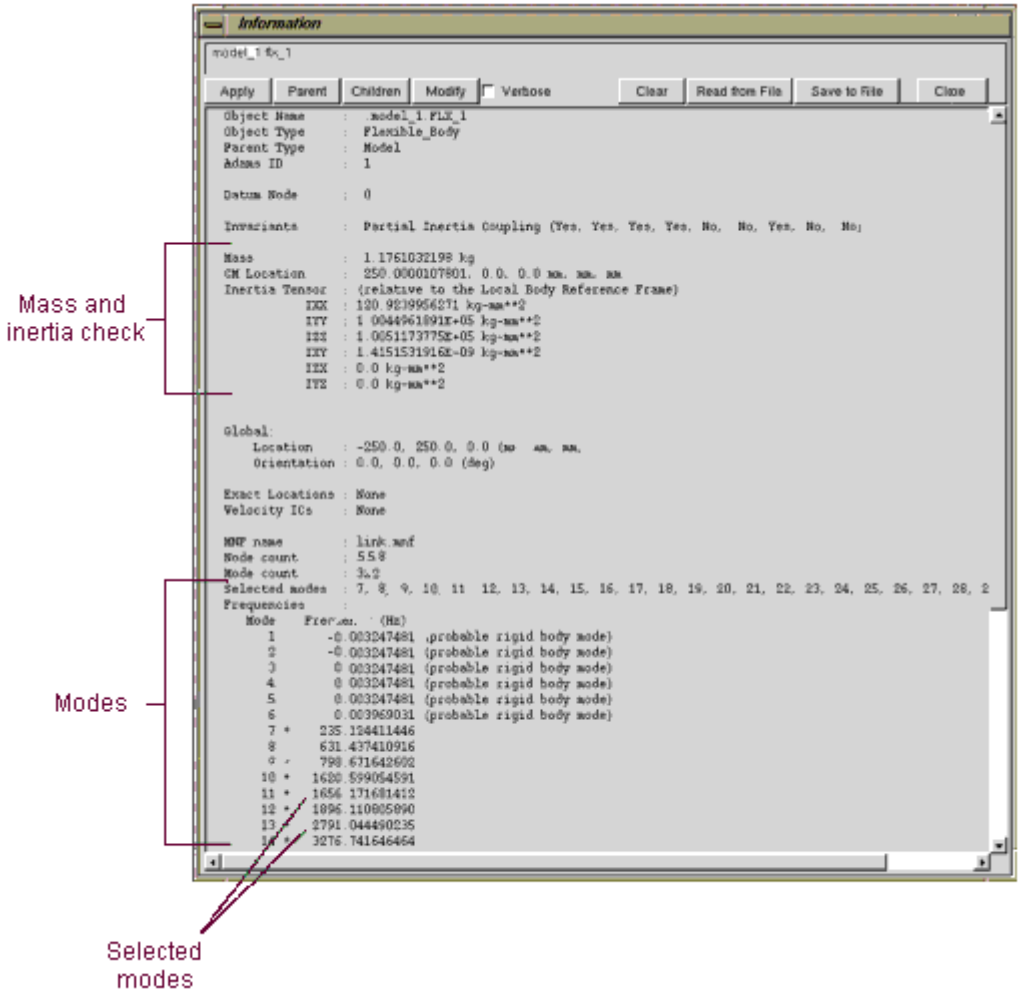
After you've imported a flexible body into Adams View, you might want to verify that the flexible body translated correctly. You can check it two ways:

- Using the Info Tool to Verify Flexible Bodies
- Using Adams Linear to Verify Flexible Bodies

We also provide an Example of Using Adams Linear to Verify Flexible Bodies.

Using the Info Tool to Verify Flexible Bodies

The Info tool in Adams View lets you check the units and mass of a flexible body. The following shows the information displayed about a flexible body when you select the Info tool. Learn more about using the Info Tool.



Information About a Flexible Body

To display information about a flexible body displayed on the screen:

1. Right-click the body on the screen. You may want to zoom in on the body on the screen to more easily place the cursor over just that object.
2. From the shortcut menu that appears, select **Info**.

To use the Database Navigator to display information about flexible bodies:

1. Double-click the background of the Adams View main window to clear any selections.
2. From the Information Strip, select the **Info** tool  .

The Database Navigator appears.

3. Select the flexible body that you want to display information about. Learn about Showing, Hiding, and Selecting Objects in the Database Navigator.
4. Select **OK**.

Using Adams Linear to Verify Flexible Bodies

You can use Adams Linear, an optional add-on module to Adams View, to help you verify that your flexible body translated correctly to Adams View and to gauge how many modes must be included in Adams View. To purchase Adams Linear, see your MSC Software sales representative.

You usually have two questions when working with flexible body modeling:

- Was the FE model to Adams translation process successful?
- How many modes must be included in the Adams simulation?

Adams Linear is an excellent tool for answering these questions. To answer the questions using Adams Linear, you simply create a model containing only the flexible body and constrain it to ground in a way that is similar to how the flexible body will be constrained in the final model.

You then run Adams Linear and record the natural frequencies. You compare the natural frequencies that Adams Linear found to those that the FEA program predicted for the same set of constraints. Unless the FEA translation process failed in some way (for example, units were misrepresented) or there is an inadequate number of modes, the natural frequencies should match.

You can also use this process to help you disable modes by finding which modes do not contribute significantly to important natural frequencies. You may find it useful to repeat this process for several joint configurations.

Flexible Body Modeling with Adams Solver

The topics in this section explain how to use Adams Flex with Adams Solver:

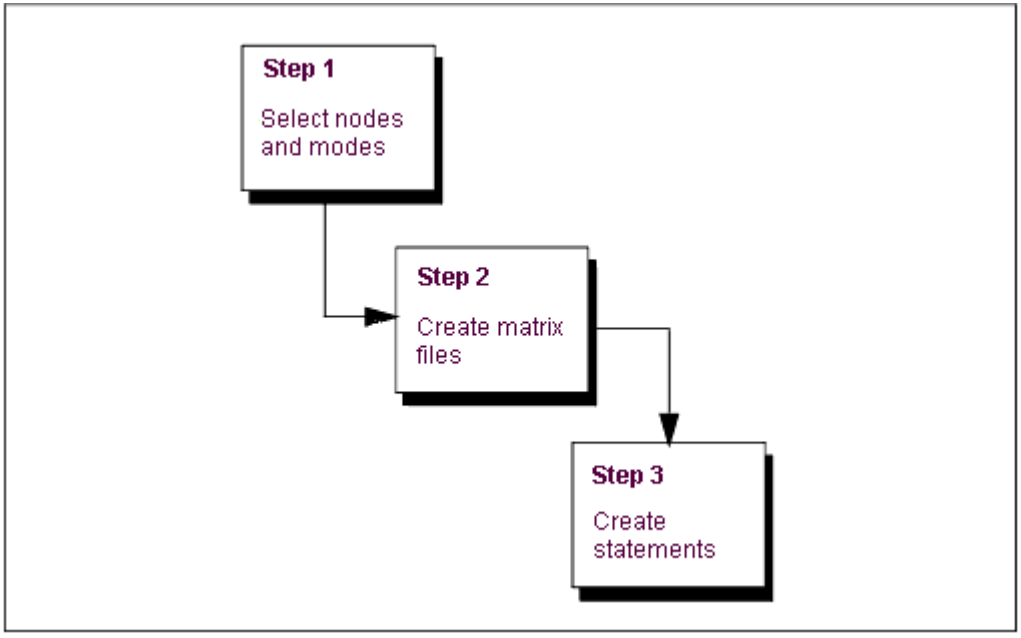
- Steps in Modeling Flexible Bodies with Adams Solver
- Selecting Modes and Nodes
- Creating Matrix Files
- Writing Flexible Body Statements

It would also be helpful to read the information in the other sections of the Adams Flex online help even if you are using Adams Solver with Adams Flex. They contain overview material that pertain to Adams Solver as well as Adams View.

Steps in Modeling Flexible Bodies with Adams Solver

To model a flexible body in Adams Solver, you do the following as shown in the figure below.

Step 1: Select Modes and Nodes	Select the modes that you want included in the dynamic simulation and decide which nodes will have markers attached.
Step 2: Create a Matrix File	Translate an MNF or MD DB created from a finite element analysis (FEA) program to a matrix file. As you do the translation , you specify the selected modes and nodes. You also specify the units for the matrix file. You use the units that you are using in your dataset (.adm file), not in your MNF.
Step 3: Create Statements	Create the following statements in your dataset: ■ FLEX_BODY statements that describe a flexible body and associate it with matrix elements. ■ MARKER statements that create markers at any node locations where you want to attach modeling elements, measure deformations, and more. You must have included these nodes in the matrix file. ■ MATRIX statements that correspond to the matrices in the matrix file.



Steps in Using Adams Solver and Adams Flex

Selecting Modes and Nodes

To increase the efficiency of Simulations, you should only include those modes that are necessary for the motion that you are attempting to model. You should, however, use caution when not including modes. If you do not include a mode, the flexible body cannot take on the particular shape of that mode and may be incorrect.

Adams Solver needs modal information corresponding to any node that has a marker on it. This information is stored in the matrix file. If you do not include a node, you cannot directly place a marker on the flexible body at that location. There is no computational overhead for nodes in a matrix file that do not have associated markers nor for markers that are not used. You should, therefore, also select nodes that may be of interest in the future.

Selecting modes without the visual assistance that Adams View provides can be difficult. You can use the Adams Flex Toolkit to browse a Modal Neutral File (MNF) to determine the modes and nodes that you want included in the matrix file. Learn about Browsing an MNF or an MD DB. Adams View also lets you automatically disable modes based on their strain energy contribution.

Creating Matrix Files

Adams Solver uses a matrix file to describe the properties of a flexible body. The matrix file contains inertia invariants and the portion of the flexible body modes shapes that correspond to marker locations. The basic FLEX_BODY formulations call for 14 matrices but the number can vary depending on:

- If all invariants except 5 and 9 exist in the MNF or MD DB, and mesh coarsening has been performed, the matrices INVAR5 and INVAR9 cannot be created and may not appear in the matrix file.
- If the flexible body has a preload, the file will contain an additional PRELOAD matrix.
- If the flexible body has modal loads defined, the file will contain an additional MODLOAD matrix.
- If the flexible body has a generalized damping matrix specified, the file contains an additional GENDAMP matrix.
- If the flexible body has an edge data specified, the file contains additional EDGE_xx list, where xx denotes an arbitrary edge ID.

As you can see, there can be as few as 12 or as many as 17 matrices, currently.

The 14 standard matrices list the selected modes and nodes, invariants, and pertinent segments of the mode shapes. The 14 matrices are:

- GENSTIFF
- INVAR1
- INVAR2
- INVAR3
- INVAR4
- INVAR5
- INVAR6

- INVAR7
- INVAR8
- INVAR9
- T_MODE
- R_MODE
- SELNOD
- SELMOD

The following shows the first part of a matrix file. The end of the matrix file that is not shown contains matrix data.

From this example, you can see that 12 matrices are defined in this matrix file and their names are listed. The first matrix defined is SELNOD. It is a full matrix with dimensions of 2 rows and 5 columns for a total of 10 elements. Each row represents a node that is referenced by a marker with a connection to the model. Listed is its node number, coordinates and whether it was promoted as an attachment node ("1") or not ("0") during MNF generation. From this example, node 1000 was promoted as an attachment and 1001 was not. The fifth column is when a simplified representation of the flex body is requested, and it may be missing in older version of Adams.

The next matrix defined in this example is SELMOD. It is a full 2 by 26 by 2 matrix for a total of 52 elements. It provides the mode number and its frequency of each mode selected for the simulation.

You generate a matrix file by translating an MNF file. You use the translator available through the Adams Flex Toolkit. The matrix file is in ADAMSMAT or ADAMSMAT2 format, and you should only create it using the translator.

For information on creating an MNF, see Translating FE Model Data. For information on creating MD DB, see Create an MD DB using MSC Nastran. For information on using the Adams Flex toolkit, see Translating an MNF or an MD DB into a Matrix File.


```
ADAMSMAT2 link  KILOGRAM  MILLIMETER  SECOND  NEWTON  TTTTFTTTF [C:/examples/mnf/link.mnf@<date>]
12
SELMOD            SELNOD            GENSTIFF          INVAR1
INVAR2            INVAR3            INVAR4            INVAR6
INVAR7            INVAR8            T_MODE            R_MODE
SELNOD    FULL    RORDER      2    5   10   ( 1I8, 3E22.14, 1I4 )
    1000  0.00000000000000E+00  0.00000000000000E+00  0.00000000000000E+00   1
    1001  5.00000000000000E+02  0.00000000000000E+00  0.00000000000000E+00   0
SELMOD    FULL    RORDER     26    2   52   ( 1I8, 1E22.14 )
       7  2.35124411445990E+02
       8  6.31437410915603E+02
       9  7.98671642602431E+02
      10  1.62059905459130E+03
      11  1.65617168141158E+03
      12  1.89611080588974E+03
      13  2.79104449023507E+03
      14  3.27674164646351E+03
      15  3.76317114349769E+03
      16  4.20876557106360E+03
      17  5.03808716009895E+03
      18  5.07827992574175E+03
      19  5.90229357708361E+03
      20  6.17326982227934E+03
      21  6.88941555442633E+03
      22  7.86823590499886E+03
      23  9.00398675924439E+03
      24  9.50922628326121E+03
      25  1.00725069783052E+04
      26  1.13587794345286E+04
      27  1.18659461223119E+04
      28  1.70694446420031E+04
      29  2.38985241406292E+04
      30  2.68184835220328E+04
      31  6.80861711686980E+04
      32  7.88351335227371E+04
INVAR1    FULL    RORDER      1    1    1   ( 1E22.14 )
  1.17610321984740E+00
INVAR2    FULL    RORDER      1    3    3   ( 3E22.14 )
  2.94025817640325E+02  0.00000000000000E+00  0.00000000000000E+00
INVAR3    SPARSE      3   26   60   ( 2I8, E22.14 )
       1       1 -1.03919729765867E-09
       1       2  5.06858777171715E-10
       1       6  2.37571684547078E-09
     .
     .
     .
```

Matrix File

Writing Flexible Body Statements

You need to create several statements in your dataset: FLEX_BODY, MARKER, and MATRIX, as explained in the next sections below. Example dataset contains an example of a dataset containing the necessary statements. You can copy the text of the dataset to a text editor so you can use it as a start to your dataset.

- Creating the FLEX_BODY Statement
- Creating MARKER Statements
- Creating the MATRIX Statement

Example Dataset Containing Flexible Body Statements

```
FLEX_BODY/1
, MATRICES = 1, 2, 3, 4, 5, 6, 7, 8, 9, 10, 11, 12, 13, 14
, INVARIANTS = T:T:T:T:T:T:T:T:F
, CRATIO = USER(.01,100,.1,500,1)
!
MARKER/39
, FLEX_BODY = 1
, NODE_ID = 21
!
MARKER/40
, FLEX_BODY = 1
, NODE_ID = 1
, QP = 1, 0, 0
!
MARKER/41
, FLEX_BODY = 1
, NODE_ID = 211
, QP = 1, 0.5, 0
!
MARKER/42
, FLEX_BODY = 1
, NODE_ID = 231
, QP = 0, 0.5, 0
!
MARKER/43
, FLEX_BODY = 1
, NODE_ID = 116
, QP = 0.5, 0.25, 0
!
!---------------------------- DATA STRUCTURES ----------------------
!
MATRIX/1, FILE = myfile.mtx, NAME = GENSTIFF
MATRIX/2, FILE = myfile.mtx, NAME = INVAR1
MATRIX/3, FILE = myfile.mtx, NAME = INVAR2
MATRIX/4, FILE = myfile.mtx, NAME = INVAR3
MATRIX/5, FILE = myfile.mtx, NAME = INVAR4
MATRIX/6, FILE = myfile.mtx, NAME = INVAR5
MATRIX/7, FILE = myfile.mtx, NAME = INVAR6
MATRIX/8, FILE = myfile.mtx, NAME = INVAR7
MATRIX/9, FILE = myfile.mtx, NAME = INVAR8
MATRIX/10, FILE = myfile.mtx, NAME = INVAR9
MATRIX/11, FILE = myfile.mtx, NAME = T_MODE
MATRIX/12, FILE = myfile.mtx, NAME = R_MODE
MATRIX/13, FILE = myfile.mtx, NAME = SELNOD
MATRIX/14, FILE = myfile.mtx, NAME = SELMOD
!
UNITS/
, FORCE = NEWTON
, MASS = KILOGRAM
, LENGTH = METER
, TIME = SECOND
!
END
```

Creating the FLEX_BODY Statement

The FLEX_BODY statement includes information on a flexible body and its associated matrix file.You must have a FLEX_BODY statement in your dataset. The FLEX_BODY statement also must include a MATRICES argument, damping ratio, and invariants. The MATRICES argument identifies the MATRIX statements in your dataset. For flexible bodies, you always have 14 MATRIX statements, and, therefore, 14 parameters to the MATRICES argument.

Notes:

Including a very high frequency mode in a flexible body analysis poses a challenge to the Adams integrators. This is because the excitation of this mode, at any frequency, can lead to numerical excitation of the mode at its high resonance frequency. While the Adams integrators are capable of tracking this response, this is rarely the analyst's intention and simulation times become unacceptable. You can control this behavior by carefully applying modal damping on a mode-by-mode basis. The CRATIO argument on the FLEX_BODY statement accepts an expression (rather than a SCALAR as in 8.2) and should be used in one of the following ways:

- Default damping is used when you omit the CRATIO argument. The default damping is 1% of critical damping for all modes with a natural frequency below 100, 10% for modes under 1000, and 100% for all modes over 1000.
- To give the same damping ratio to all modes, set CRATIO to a scalar value between 0 and 1. Recall that CRATIO is a ratio of critical damping. To turn off default damping, set CRATIO=0 for zero damping (this was the 8.2 default). Note that since CRATIO is now an expression, it must be followed by a backslash (\) except when it is the last keyword on the statement.
- To vary the damping based on individual modes or run-time results, you can set CRATIO equal to an Adams Solver function expression. In particular, you can use the function expressions FXFREQ and FXMODE to apply different damping levels to different modes.
- To control the modal damping of individual modes using a user-written subroutine, use CRATIO=USER(...). Adams Solver obtains the damping ratios for the selected modes through a call to the DMPSUB subroutine.

Creating MARKER Statements

You use the MARKER statement to create a marker at every node location where you want to create an attachment. The node IDs that you use in the MARKER statement to define a location must match those in the SELNOD section of the matrix file.

- For information on the matrix file, see Creating Matrix Files.
- For information on placing markers at locations where no nodes exist, see the Adding Markers to Flexible Bodies.

Creating the MATRIX Statement

You need a MATRIX statement for each of the 14 matrixes in the matrix file. The format of the MATRIX statement for a flexible body is:

```
MATRIX/n FILE = myfile.mtx, NAME = string
```

The arguments are:

- *n*, which maps to a number in the MATRICES argument in the FLEX_BODY statement.
- FILE=, which is the name of the matrix file.
- NAME=, which specifies the name of a matrix in the file that FILE identifies (for example, GENSTIFF).

Load Case Selection

When you create a non-linear flexible body in Adams View, Adams View reads the Bulk Data File format (BDF) and reads the DLOAD, TLOAD1 and TLOAD2 entries from the Bulk Data section of this file.

DLOAD (Defined in the Bulk Data Section of the BDF file submitted to the Nastran Solver) stands for Dynamic Load Set Selection.

TLOAD1 and TLOAD2 define a time-dependent dynamic load or enforced motion for use in transient response analysis.

Dynamic load sets are selected in the Case Control Section with DLOAD=SID. In case there is no load cases selected, Adams will ignore this definition. TLOAD1 and TLOAD2 loads may be combined only through the use of the DLOAD entry. In Adams, only multiple TLOAD(1/2) bulk data entries can be combined using the DLOAD case control entry and the scale factor 1.0 is used in this case.

The following options are available in the Load Cases tab in the "**Flexible Body Modify**" dialog box for Non-Linear Flexible bodies:

1. **Activate All:** When this button is clicked, all the load cases read from the bulk data section of the input BDF file is selected for activation. We can only activate load cases that satisfy the criteria given below:
   a. Multiple TLOAD(1/2) load cases can be selected, but cannot be combined with any DLOAD load cases.
   b. If DLOAD load cases are selected for activation, we can select at the most one load case for activation.

      On clicking **Activate All** if you get an error message, please click on the **Select** button to individually select load cases for activation that satisfy the above criteria.
2. **Deactivate All:** All load cases read from the bulk data section of the input BDF file are deactivated by default. If there are some load cases activated previously, clicking on the **Deactivate All** button will deactivate all load cases, during the Solver run.
3. **Select:** You can select individual load cases to be activated by clicking on the **Select** button. This will open the "**Load Case Selection**" dialog box.

   If there are load cases read from the input BDF file, these will be displayed in the list on the left.

To Activate load cases

1. Select load cases from the list on the left by clicking on them. Alternatively, clicking on the **Select All** button will select all load cases.
2. Click on the "**Activate→**" button to activate the selected load cases. The selected load cases will be moved from the list on the left to the right.

To Deactivate load cases

1. Select load cases from the list on the right by clicking on them.
2. Click on the "**←Deactivate**" button to move all the selected load cases back in the list on the left.

Notes:	
	1. To deactivate all selected load cases, you can also click on the **Clear All** button. 2. You can enter strings in the "**Filter**" field to look for certain load cases beginning with some characters. For example, if the Filter string is "DL*" or "D*", only DLOAD bulk data entries will be filtered in the list on the left. 3. If an invalid selection of load cases is made for activation, an error message is thrown. 4. Clicking on the **Close** button will close the "**Load Case Selection**" dialog box and mark all the selected load cases for activation in the "**Modify Flexible Body**" dialog box for Non-Linear flexible bodies. Clicking on the **Apply** button in this dialog box will mark these load cases for de-activation and will get written into the input file for the Nastran Solver when you run a simulation.

Setting Up Flexible Bodies for Testing

Testing Setup

Once you've built your flexible body into Adams View, you can set it up for testing or simulation. For example, you can change its modal content to improve the efficiency of the simulation.

Accessing the Flexible Body Modify Dialog Box

To modify a flexible body for simulation, you use the Flexible Body Modify dialog box. Follow the instructions below to learn how to display the Flexible Body Modify dialog box.

To display the modify dialog box for a flexible body on the screen:

1. Right-click the flexible body whose properties you want to modify. You may want to zoom in on the flexible body on the screen to more easily place the cursor over just that body. Learn about View Options.
2. From the shortcut menu that appears, point to the flexible body that you want to modify, and then select **Modify**.

   The Flexible Body Modify dialog box appears.

Tip:	Double-click the flexible body to display the modify dialog box.

To use the Database Navigator to display the Flexible Body Modify dialog box:

1. Double-click the background of the Adams View main window to clear any selections.
2. From the **Edit** menu, select **Modify**.

   The Database Navigator appears.
3. Select the current model and the flexible body whose properties you want to modify.
4. Select **OK**.

   The Flexible Body Modify dialog box appears.

Changing Flexible Body and Modal Initial Conditions

You can set the initial velocity and position for a flexible body before the Simulation starts just as you can for any part in Adams View. You can set how you want Adams View to calculate these properties as well as define these properties yourself. You can also set modal initial conditions.

To set the flexible body initial velocity and position:

1. In the Flexible Body Modify dialog box, select either of the following:
   - **Position ICs**

- **Velocity ICs**

2. Set the velocity and position values in the dialog boxes that appear. Learn about:
   - Position ICs with Modify Body - Position Initial Conditions dialog box help.
   - Velocity ICs with Modify Body - Velocity Initial Conditions dialog box help.

To set the modal initial velocity and displacement:

1. In the Flexible Body Modify dialog box, select Modal ICs:

   The Modify Modal ICs dialog box appears.
2. Highlight the mode whose initial velocity or displacement you want to set.
3. Enter the velocity or displacement in the text box:
   - Enter the velocity in the text box above the button Apply Velocity IC.
   - Enter the displacement in the text box above the button **Apply Displacement IC**.
4. Select either **Apply Velocity IC** or **Apply Displacement IC**.
5. To make Adams Flex enforce the displacement initial conditions exactly, select **Set Exact**.
6. Select **OK**.

Modifying the Modal Formulation

Adams Flex computes the time varying mass matrix of the flexible body using nine inertia invariants. (For details, see Theory of Flexible Bodies.) Four combinations of invariants have special significance and they are provided with Adams Flex. In most cases, the modal basis in the MNF is an orthogonal set including six rigid body modes. Theoretically, invariant 3 and 4 are zero in this situation even though you may see some small non-zero entries due to numerical errors. So, invariants 3 and 4 are disabled in all the four combinations by default. If you want to enable them, you can choose to customize the invariant formulation. The standard formulations are:

- **Rigid body** - In this formulation, Adams Flex disables invariant 6, the modal mass, and the flexible body is considered rigid. Adams View ignores all modes, even those you enable, during the simulation. The results of the flexible body simulation closely resemble those for an Adams rigid part although formulation differences can cause subtle result variations.
- **Constant** - In this formulation, Adams Flex disables invariants 3, 4, 5, 8 and 9. The flexible body's inertial properties are unaffected by deformation (that is, deformation and rigid body motion are uncoupled).

  The **Constant** option may only have academic value because computational savings will be modest while potentially having a dramatic effect on results. When you select **Constant**, Adams View does not account for changes in the moment of inertia due to deformation.
- **Partial coupling** - In this formulation, which is the default, Adams Flex disables invariants 3, 4, 5 and 9. Invariants 5 and 9 provide a second-order correction to the flexible body inertia tensor. These invariants impose the greatest computational overhead on the evaluation of the flexible body equations of motion. Disabling these invariants can reduce CPU time significantly while having minor impact on results in most cases.

- **Full coupling** - In this formulation, Adams Flex enables all of the invariants except for invariants 3 and 4. Use this method to achieve full accuracy.

When Adams Flex creates a flexible body, it uses the Partial Coupling formulation by default because Partial Coupling has significant computational efficiency over the more accurate Full Coupling formulation. You should verify, however, that your model does not require Full Coupling.

You should always be careful when using the Constant formulation even when you expect deformations to be small. Use it only after careful experimentation.

The Rigid Body formulation removes all flexibility effects, and you should only use it as a debugging tool.

To set a modal formulation:

1. In the Inertia modeling area of the Flexible Body Modify dialog box, select a formulation option or select **Custom**.

   When you select Custom, Adams Flex displays a Custom Inertial Modeling dialog box that lets you set up the invariants that you want selected.

2. Use the dialog box to select the inertia invariants, and then select **OK**.

   You can also use this dialog box to view the effects of the different options. For example, select **Partial Coupling** to view the invariants that option disables and enables.

Controlling Errors During Simulations

You can customize the error control for flexible body numerical results using the Solver Settings - Dynamics dialog box. This is the same dialog box you use in Adams View to control the amount of error control during a Dynamic simulation. You use the Scale text box to control the flexible body numerical results.

The Scale text box takes three values: r1, r2, and r3. These values are used to rescale the sum of the relative and absolute error tolerances. If T is the sum of the relative and absolute error tolerances applied to the state vector, then:

- A tolerance of r1 x T is applied to the translational coordinates.
- A tolerance of r2 x T is applied to the rotational coordinates.
- A tolerance of r3 x T is applied to the modal coordinates.

The scaling applies to only the WSTIFF, DSTIFF, and ABAM integrators. It does not apply to GSTIFF and Constant BDF. See Comparison of Integrators.

The default value of r3 is 1E-3, but if the flexible body deformations are so subtle that the modal coordinates are of the order 1E-7, then the integrator will probably have insufficient accuracy. In this case, you should decrease the scale factor, for example, to 1E-7.

To set the error control:

1. On the **Settings** menu, point to **Solver**, and then select **Dynamics**.

   The Solver Settings - Dynamics dialog box appears.

2. If necessary, at the bottom of the dialog box, select **More**.

3. In the **Scale** text box, enter the three values representing the error control.

Changing Modal Content

The Flexible Body Modify dialog box lets you manage which modes of your flexible body you want included during a simulation. To help you determine which modes to include, you can view each mode individually and animate its deformations.

Learn more about how to change the properties of your flexible body:

- About Flexible Body Modal Content
- Viewing Modes
- Animating a Mode
- Enabling and Disabling Modes

About Flexible Body Modal Content

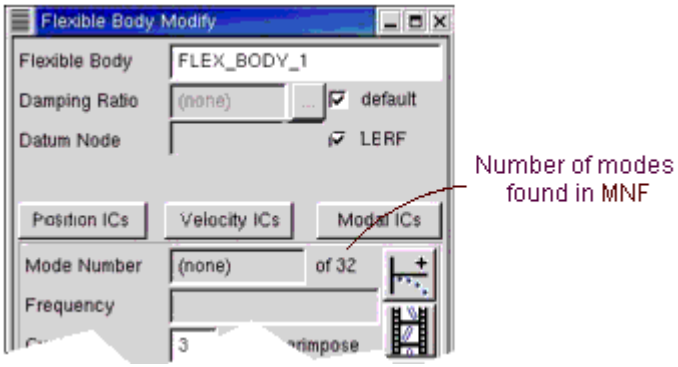
By default, when you create a flexible body from a Modal Neutral File (MNF) file in Adams View, Adams Flex enables all modes in the MNF except those that it considers rigid body modes. Because Adams Flex supplies each flexible body with six nonlinear rigid body degrees-of-freedom, you can get a singularity in the results if you try to run a simulation with the rigid body modes enabled. Note, however, that because the zero natural frequency of a rigid body mode is sometimes inaccurate, Adams Flex sometimes fails to recognize a rigid body mode. If that happens, you need to disable the rigid body mode as explained in Enabling and Disabling Modes.

To increase the efficiency of simulations, you should disable any modes that do not contribute to the motion that you are attempting to model. You should use caution when disabling modes, however. Once you disable a mode, the flexible body cannot take on the particular shape of the disabled mode. This is the same as applying a constraint to the model.

Using Adams Flex, you can enable and disable modes individually based on their mode number or strain energy contribution, as a group based on their mode number or frequency, and through a table that lets you view all modes at once.

Viewing Modes

When you create a flexible body from a Modal Neutral File (MNF) in Adams View, Adams Flex calculates the number of modes in the flexible body. It displays the number of modes on the Flexible Body Modify dialog box as shown below.



Mode Display in Flexible Body Modify Dialog Box

When you display a mode, Adams Flex displays its frequency in the **Frequency** text box. Also, when you display a mode, the mode deformations appear along with the undeformed flexible body. You can turn this off to display only the deformed mode.

To display a specific mode:

- In the **Mode Number** text box, enter the mode number, and then press **Enter**.

To view the next mode:

- Select the **Next Mode** tool  .

To view the previous mode:

- Select the **Previous Mode** tool  .

To select a mode based on its frequency:

- In the **Frequency** text box, enter the frequency, and then press **Enter**.

  The mode closest to the specified frequency appears.

To turn off superimposing:

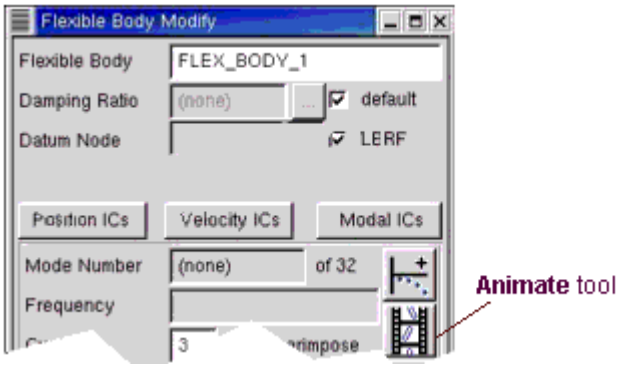
- Clear the selection of **Superimpose**.

Note:	You can also filter out graphically insignificant modes before running an animation. Learn more about Filtering Modes

Animating a Mode

The Flexible Body Modify dialog box contains an **Animate** tool  that lets you animate a particular mode to see how it deforms. By default, the animation runs 3 times or through 3 cycles.

The location of the **Animate** tool is shown below. You can also use the **Animation** tool on the Main toolbox to animate the entire model containing the flexible body after you've run a simulation.



Animate Tool on Flexible Body Modify Dialog Box

To animate a mode:

1. View the mode that you want to animate as explained in Viewing Modes.
2. In the **Cycles** text box, enter the number of animation cycles.
3. Select the **Animate** tool  .

Note: You can also filter out graphically insignificant modes before running an animation. Learn more.

Enabling and Disabling Modes

You can enable and disable modes in the following ways:

- Enabling and Disabling Modes Individually
- Enabling and Disabling Groups of Modes
- Enabling and Disabling Modes Based on Strain Energy Contribution

Enabling and Disabling Modes Individually

To enable or disable modes individually based on their number:

1. View the mode that you want to enable or disable as explained in the Viewing Modes.

2. In the Flexible Body Modify dialog box, select **Disable** or **Enable** as appropriate. If you disable a mode, its mode number and natural frequency appear in parentheses.
3. Select **OK**.

Enabling and Disabling Groups of Modes

You can disable or enable a group of modes based on either their mode number or frequency by entering a range of values in a dialog box or by selecting the modes from a table.

To enable or disable a range of modes through a dialog box:

1. From the Flexible Body Modify dialog box, select **range**.

   The Enable or Disable a Range of Modes dialog box appears.
2. Select the options from the dialog box to set how you'd like to disable or enable the range, and select **OK**.
3. From the Flexible Body Modify dialog box, select **OK**.

To disable or enable range of modes using a table:

1. From the Flexible Body Modify dialog box, select **Modal ICs**.

   The Modify Modal ICs dialog box appears. Click here for an image. It displays all the modes in the flexible body. An asterisk (*) appears next to all modes that are enabled.
2. Highlight the modes that you want to enable or disable.
3. Select **Disable Highlighted Modes** or **Enable Highlighted Modes**.
4. Select **Close** to return to the Flexible Body Modify dialog box.

Enabling and Disabling Modes Based on Strain Energy Contribution

After performing a pilot simulation, you can disable those modes that do not significantly contribute to the total strain energy of a flexible body. You can set Adams View to automatically disable any modes that contributed less than a specified fraction of the total strain energy during the test simulation. After disabling the modes that do not significantly contribute to strain energy, simulation times should be reduced.

The pilot simulation that you run should be typical of the simulations that you are going to run on the flexible body. Disabling a particular set of modes based on the simulation of one event might be totally inappropriate when simulating another unrelated event.

An important energy contribution from a model might be of a short duration that its total energy can seem insignificant but yet the mode is important to the simulation. For example, although the strain energy a mode contributes may be quite low compared to the total strain energy for the entire simulation, Adams Solver may be unable to complete the simulation without the mode. Therefore, you should choose a very conservative strain energy fraction (for example, 0.001 or smaller). If a mode is truly inactive, it will be disabled in spite of a low setting.

Disabling an important mode can adversely affect the numerical conditioning of a flexible body model, however. The computational savings that you gain by disabling a mode can be quickly lost by the computational cost of a difficult simulation.

To disable or enable modes based on their contribution of strain energy:

1. Run a pilot simulation of the model containing the flexible body.
2. From the Flexible Body Modify dialog box, select **auto**.

   The Auto Disable Modes by Strain Energy dialog box appears.
3. In the **Analysis Name** text box, enter the name of the pilot simulation.
4. In the **Energy Tolerance** text box, enter a fractional value. Adams Flex will disable all modes that contributed less than the specified fraction to the total strain energy during the test simulation. For example, to disable all modes that contributed less than 0.1% of the strain energy, enter 0.001.
5. Select **OK**.

Specifying Damping

Several types of damping are inherently present in mechanical systems. Understanding the source and level of damping are important in the simulation and testing of dynamic systems. For example, a mechanism having low natural frequencies and relatively low damping, could produce damaging motions under resonant conditions.

Applying damping judiciously can also improve simulation performance for models containing flexible bodies. For example, consider a flexible body with a 10 kHz mode whose shape is considered essential to allowing the body to assume a particular deformation. Any response in this mode at its resonant frequency dictates integration steps on the order of 1E-5 seconds, which can be unacceptable for long duration simulations. If the damping for this mode is set at 100% of critical damping, however, any resonant response is immediately suppressed. The mode's compliance is retained but its dynamics are eliminated and the simulation performance is improved.

Learn more:

- Damping Dissipation and Damping Matrix
- Specifying Modal Damping
- Specifying Generalized Damping

Damping Dissipation and Damping Matrix

The damping force on a flexible body is proportional to its generalized velocities and is assumed to be derivable from the quadratic form:

$$\Im = \frac{1}{2}\dot{\bar{q}}^T \boldsymbol{D}\dot{\bar{q}}$$

where:

D	is a symmetric matrix of damping coefficients
$\dot{\bar{q}}$	is a vector of generalized velocities:
	$\dot{\bar{q}} = \begin{bmatrix} \dot{x} \\ \dot{y} \\ \dot{z} \\ \omega_x \\ \omega_y \\ \omega_z \\ \dot{q}_1 \\ \dots \\ \dot{q}_2 \end{bmatrix}$

- $\dot{x}, \dot{y}, \dot{z}$ are the absolute time derivatives of the position vector coordinates of the local part reference frame with respect to the local part reference frame.
- $\omega_x, \omega_y, \omega_z$ are the angular velocity vector coordinates with respect to the local part reference frame.
- $\dot{q}_1, \ldots, \dot{q}_n$ are the time derivatives of the modal coordinates.

The matrix **D** is composed of two parts, $\mathbf{D_m}$ and $\mathbf{D_g}$:

$$D = D_m + D_g$$

$\mathbf{D_m}$ represents the contribution of proportional modal damping, and $\mathbf{D_g}$ represents the contribution of a generalized damping matrix. Both are explained in the next section.

Specifying Modal Damping

The modal damping matrix $\mathbf{D_m}$ is diagonal and defined using critical damping ratios ζ_i for each mode i= 1,*n*.

$$\mathbf{D}_m = \begin{bmatrix} \mathbf{0}_{3x3} & \mathbf{0}_{3x3} & \mathbf{0}_{3x1} & \mathbf{0}_{3x1} & \cdots & \mathbf{0}_{3x1} & \mathbf{0}_{3x1} \\ \mathbf{0}_{3x3} & \mathbf{0}_{3x3} & \mathbf{0}_{3x1} & \mathbf{0}_{3x1} & \cdots & \mathbf{0}_{3x1} & \mathbf{0}_{3x1} \\ \mathbf{0}_{1x3} & \mathbf{0}_{1x3} & 2\zeta_1\sqrt{k_1 m_1} & 0 & \cdots & 0 & 0 \\ \mathbf{0}_{1x3} & \mathbf{0}_{1x3} & 0 & 2\zeta_2\sqrt{k_2 m_2} & \cdots & 0 & 0 \\ \vdots & \vdots & \vdots & \vdots & \ddots & \vdots & \vdots \\ \mathbf{0}_{1x3} & \mathbf{0}_{1x3} & 0 & 0 & \cdots & 2\zeta_{n-1}\sqrt{k_{n-1} m_{n-1}} & 0 \\ \mathbf{0}_{1x3} & \mathbf{0}_{1x3} & 0 & 0 & \cdots & 0 & 2\zeta_n\sqrt{k_n m_n} \end{bmatrix}$$

where k_i and m_i are the generalized stiffness and mass for mode *i*.

Note: The damping ratio ζ_i does not need to be constant. It can be a function of time or system state.

If you do not specify modal damping when you create the flexible body, Adams Flex applies a default, non-zero critical damping ratio as follows:

- 1% damping for all modes with frequency lower than 100.
- 10% damping for modes with frequency in the 100 to 1000 range.
- 100% critical damping for modes with frequency above 1000.

You can change the default modal damping in three ways:

- Assign a single scalar critical damping ratio that Adams Flex applies uniformly to all modes.
- Enter Adams run-time function expressions to create complex damping phenomena in your flexible body. In addition, function expressions, such as FXFREQ and FXMODE, allow you to apply different levels of damping to individual modes.
- Control the damping using the DMPSUB user-written subroutine. DMPSUB lets you set different levels of damping for different modes and the damping can vary over time. For more on writing subroutines, see the Subroutines section of the Adams Solver online help.

To assign modal damping when creating or modifying a flexible body:

1. In either the Create a Flexible Body dialog box or Flexible Body Modify dialog box, clear the selection of **default** (use **default** in the Create a Flexible Body dialog box).
2. In the **Damping Ratio** text box, either:
   - Enter the critical damping ratio.
   - Enter a function. To get help building the function, next to the **Damping Ratio** text box, select the **More** button ... . The Adams View Function Builder appears. For information on using the Function Builder, see the Adams View Function Builder online help.

3. Continue creating or modifying the body, and then select **OK**.

Specifying Generalized Damping

The generalized damping matrix $\mathbf{D_g}$ is a constant symmetric matrix of the form:

$$D_g = \begin{bmatrix} D_{g_{TT}} & D_{g_{TR}} & D_{g_{TM}} \\ D_{g_{TR}} & D_{g_{RR}} & D_{g_{RM}} \\ D_{g_{TM}} & D_{g_{RM}} & D_{g_{MM}} \end{bmatrix}$$

To better understand how the generalized damping matrix is handled in Adams Flex, it is helpful to start with the discrete finite element equations of motion:

$$M\ddot{x} + B\dot{x} + Kx = F$$

where:

- **M**, **K**, and **B** are the finite element mass, stiffness, and damping matrices, respectively
- **x** is the nodal coordinate vector
- **F** is the applied force vector

The damping matrix **B** is derived from damping elements and parameters defined in the finite element model. The previous equation can be transformed into modal coordinates:

$$P^T MP\ddot{q} + P^T BP\dot{q} + P^T KPq = P^T F$$

where **P** is the matrix of mode shapes stored column-wise and **q** is the vector of modal coordinates. $\mathbf{P^T BP}$ represents the generalized damping matrix. However, before Adams Flex can use the generalized damping matrix, the portion of $\mathbf{P^T BP}$ that projects onto the rigid body modes must be transformed to the nonlinear, large motion, generalized coordinates: X, Y, Z, ψ, θ, and ϕ used to represent the flexible body's large overall motion in Adams Solver (C++). To this end, a *m* x 6 transformation matrix, **A**, is constructed and transforms *m* rigid body modes to the six coordinates X, Y, Z, ψ, θ, and ϕ, and the final generalized matrix $\mathbf{D_g}$ is computed:

$$D_g = \begin{bmatrix} A^T & 0 \\ 0 & I_{nxn} \end{bmatrix} P^T BP \begin{bmatrix} A & 0 \\ 0 & I_{nxn} \end{bmatrix}$$

If the damping description in the finite element model results in a resultant damping force, there will be nonzero entries in the sub-matrices $D_{g_{TT}}$, $D_{g_{TR}}$, $D_{g_{TM}}$, $D_{g_{RR}}$, $D_{g_{RM}}$. Because the resultant damping force was derived from a linear finite element model governed by small strain approximations and infinitesimal

rotations, a resultant damping force may yield unexpected results in the context of large overall motion supported in Adams. Therefore, Adams Flex provides the option to ignore the resultant damping force. Ignoring a resultant damping force is referred to as *internal-only* generalized damping.

Because the generalized matrix $\mathbf{D_g}$ is derived from the component finite element model, you can leverage the damping elements and features in the finite element program. This is particularly useful for defining non-proportional and spatially-dependent damping. Furthermore, the generalized damping matrix is stored in the MNF to be optionally applied to the flexible body. That is, because you defined damping in the finite element model, it is not necessary to employ it in Adams. To enable generalized damping, however, you must have a generalized damping matrix stored in your MNF.

To specify generalized damping when creating or modifying a flexible body:

1. In either the Create a Flexible Body dialog box or Flexible Body Modify dialog box, set **Generalized Damping to**:
   - **Off** - Disables the generalized damping.
   - **Full** - Enables the complete generalized damping matrix, including the effects of a resultant damping force.
   - **Internal Only** - Only enables the portion of the generalized damping matrix corresponding to the modal coordinates (that is, ignore the resultant damping force).
2. Continue creating or modifying the body, and then select **OK**.

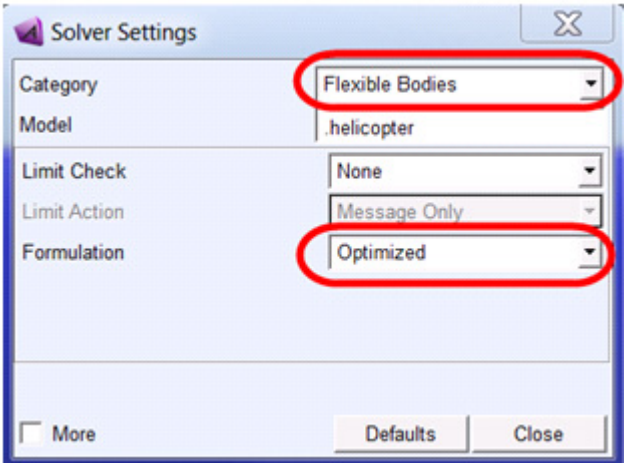
Formulations in Adams Solver

There are three formulation options for solving Adams Flex flexible bodies. The default "Original" formulation and two other choices: the "optimized" and "max optimization" formulations. All these formulations are based the 'floating frame of reference' method. They only differ in the ways the equations of motion are solved numerically.

These formulations can be selected through the Adams Solver PREFERENCES statement as

```
PREFERENCES/FLEX_BODY_FORMULATION = {ORIGINAL, OPTIMIZED,
MAX_OPTIMIZATION}
```

Also, in Adams View, the formulation can be selected from **Settings → Solver → Flexible Bodies - Formulation**:



OPTIMIZED FORMULATION

In this formulation, the 'modal equations' of each flex body are solved separately from the rest of equations. A separate numerical integrator is used for each subsystem of modal equations. The performance gain is obtained using better linear algebra techniques which leverage special structure of the Jacobian matrix used in the modal equations.

Current limitations:

- Generalized damping of flex body is not supported; users are advised to use modal damping
- Full or custom inertia coupling is not supported. Users are advised to use either constant or partial inertia coupling
- Flex contacts (flex-to-flex or flex-to-rigid) are not supported
- SI2 integrators are not supported. Only HHT and GSTIFF- I3 integrators are supported
- Motion on flex marker is not supported (workaround - use dummy part)
- Some models need kmax=1 settings in GSTIFF for speed and robustness improvement

In this formulation some of the entries in Jacobian matrix are neglected which are multiplied by h or h^2 where h is the time step of numerical integrator. As h approaches $\rightarrow 0$, these entries become negligible. Due to this assumption this formulation shows better performance when the step size is restricted through HMAX settings. Without this restriction, for some models the Newton-Raphson scheme takes more iterations to converge making this formulation slower than the Original formulation.

MAX OPTIMIZATION FORMULATION

In this formulation, the 'modal equations' of each flex body are solved separately from the rest of equations using a semi-analytical method. This is quite similar to the Optimized formulation; but, for some models it can outperform the Optimized formulation.

Current limitations:

- All the limitations of Optimized formulation
- Does not support constraints applied directly to the flexible body; one could consider replacing the flex body joints/jprims with stiff bushings

FLEXIBLE BODY FORMULATION SELECTION

The Optimized and Max Optimization formulations will show performance gain (without losing accuracy) for models where there is no 'strong coupling' between the modal deformation and rigid body motion in the flex body. They are targeted for models where 'weak coupling' is present. Note that the Original formulation is the most robust formulation as it can handle any type of coupling.

Due to the current limitations of the Optimized and Max Optimization formulations described above, some type of models are not supported. However, if the one's global setting for formulation (it is not possible to specify this setting per flexible body) is not compatible with some flexible bodies in the model, solver will automatically revert to the compatible formulation.

For example:

- If the Max Optimization formulation is selected but a flexible body in the model is connected directly to a kinematic joint, Adams Solver will automatically revert to usage of the Optimized formulation for all flexible bodies in the model
- If the Optimized formulation is selected but a flexible body in the model uses full inertia coupling, Adams Solver will automatically revert to usage of the Original formulation for all flexible bodies in the model

Runtime Switching between Flexible and Rigid

A flexible body's representation can be switched between rigid and flexible during an analysis. This is useful when only a subset of an event requires flexibility in a given component. Simulation time can be reduced by treating some components flexibly for only those portions of the analysis where the result accuracy actually requires it. To use this capability one must, either in Adams View or in the .adm file, declare a flexible body to be a "dual-representation" body and specify its initial representation as rigid or flexible (modal). For details on the procedure, see the Flexible Body Modify dialog box help or the FLEX_BODY statement description within the Adams Solver documentation. Then, in an .acf one can use the extended FLEX_BODY command to switch the representation at a specific time or after a SENSOR has been triggered and returned control back to the .acf. The command syntax follows:

```
FLEX_BODY/id, REPRESENTATION = MODAL
FLEX_BODY/id, REPRESENTATION = RIGID
```

One can switch from rigid to flexible or from flexible to rigid. One can also perform multiple switches in the same simulation script or .acf. Whenever the representation is set to rigid, the flexible body behaves as though the inertia modeling selection is set to rigid (also known as disabling INVAR6). Since Adams 2012 flexible bodies set to have a rigid representation via the inertia modeling options perform much more closely to true rigid bodies (parts) in terms of simulation speed. This makes it much easier to compare the accuracy/performance balance of your model versus swapping true rigid parts with flexible bodies.

When post-processing a dual-representation flexible body one will of course notice that outputs specific to the flexible representation (stress/strain recovery, flexible mode states and so on.) lack any data for the duration(s) in which the representation was set to rigid. Also, one will see a significant speed up in animation for those time intervals when a flexible body is rigid since the animation code does not need to compute changes in deformations or stress/strain.

Reviewing Results

Visually Investigating Results

Because Adams Flex is built right into Adams View, you can view the deformations of each mode and change their visual display to help you investigate the results of the simulations. The next sections explain how you can change the visual display to investigate the deformations of your flexible body.

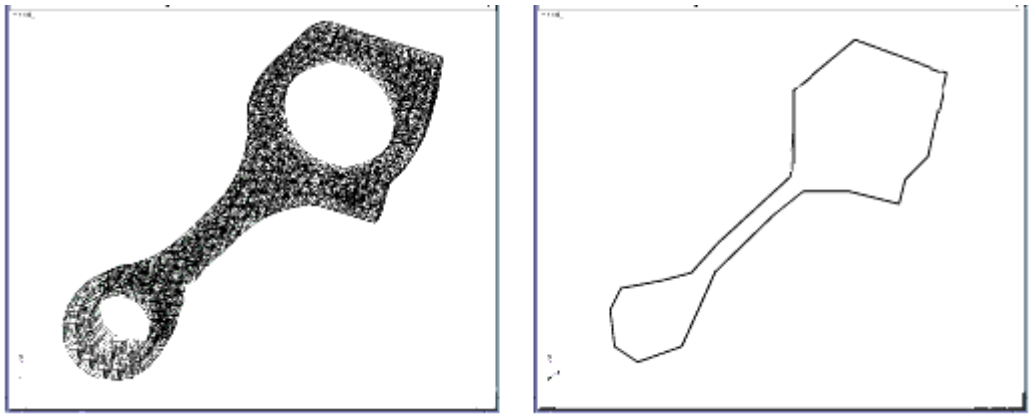
- Substituting Outline Graphics for the Finite Element Mesh
- Scaling the Deformations
- Adding Contour or Vector Plotting
- Specifying a Deformation Datum Node
- Filtering Modes

Substituting Outline Graphics for the Finite Element Mesh

When you build a flexible body into Adams View, Adams Flex creates a mesh on the body based on the description of the flexible body in the Modal Neutral File (MNF). Often flexible bodies are based on a finite element model with such a high level of detail that it stretches the capabilities of your graphical hardware. In other cases, the detailed mesh can hide other modeling elements.

If you encounter either of these, you can substitute a graphical outline for the graphics from the MNF by sketching an outline.

Note:	To reduce animation times, you can also set Adams Flex so it treats the flexible body as a rigid body during animations. For more information, see Scaling the Deformations. In addition, you can use the MNF optimization capabilities of the Adams Flex toolkit to permanently reduce the complexity of MNF geometry. For more information, see Optimizing an MNF or an MD DB.



Outline Graphics Applied to a Flexible Body

To sketch an outline:

1. Access the Flexible Body Modify dialog box as explained in the Accessing the Flexible Body Modify Dialog Box.
2. Select the **Graphics Outline** tool  .
3. Draw the outline by selecting nodes on the flexible body using the left mouse button.
4. When the outline is complete, select the right mouse button.

To turn on the viewing of the outline:

- In the Flexible Body Modify dialog box, under the Graphics section, select **outline**.

To turn off the viewing of the full MNF graphics:

- In the Flexible Body Modify dialog box, under the Graphics section, clear the selection of **full MNF graphics**.

To delete an outline:

1. Right-click the outline graphic you want to delete.
2. From the shortcut menu that appears, select **Delete**.
3. In the Flexible Body Modify dialog box, under the Graphics section, clear the selection of **outline**.

Scaling the Deformations

You can change the amount by which Adams Flex deforms a mode. You can exaggerate deformations so you can see deformations that might otherwise be too subtle to see, or you can limit the deformations. The default scale factor is 1.

Note that setting the scale factor to a value other than 1 can make the joints at the flexible body appear to separate. This is because the motion of a point on a flexible body is the sum of the deformation that has been scaled and a rigid body motion that is **not** scaled.

In addition, if you set the scale to 0, Adams Flex treats the flexible body as a rigid body during animations.

To scale the deformations:

1. Display the Flexible Body Modify dialog box as explained in Accessing the Flexible Body Modify Dialog Box, if necessary.
2. Move the **Deformation Scale Factor** slider to the desired scale value or, for greater exaggeration, type a value in the text box next to the slider.
3. Animate the flexible body to see the changes. Learn Playing an Animation.

   Adams Flex now exaggerates the deformations of the flexible body by the amount you requested.

Adding Contour or Vector Plotting

You can set Adams Flex so that it displays Contour plots or Vector plots.

In addition, in Adams PostProcessor, you can view the contours of deformations, as well as modal forces and stress and strain results you obtained using Adams Durability. For more information on displaying contour and vector plots in Adams PostProcessor, see Animating Flexible Bodies and Adams Durability Results.

By default, Adams Flex considers the deformation to be relative to the origin of the flexible body (its local body reference frame (LBRF) or coordinate system). You'll notice that at the start of the animation, the flexible link is completely blue. As the animation runs, it changes to red to indicate where and when the maximum deformation occurred.

To set color contouring or vector plots:

1. Display the Flexible Body Modify dialog box as explained in the Accessing the Flexible Body Modify Dialog Box, if necessary.
2. Set **Plot Type** to **Contour** or **Vector**. You can also set it to **Both**.
3. Animate the flexible body to see the changes. Be sure to turn on the display of contour plots by selecting **Contour Plots** in the Animation Controls dialog box in Adams View. See Using Animations.

   Notice that there is a short hesitation before the animation starts because Adams View computes and scales colors based on the deformation that occurred during the simulation.

Specifying a Deformation Datum Node

You can set the datum node for which you want deformation color changes to be relative to using Adams Flex. Adams Flex considers the deformation to be relative to the origin of the flexible body (its local body reference frame (LBRF) or coordinate system) by default. For example, if you were modeling a cantilever beam in Adams Flex, you could specify that deformations should be relative to the clamped end as was illustrated in the first tutorial, Building and Simulating a Flexible Model, of Getting Started Using Adams Flex.

To specify a datum node:

1. Display the Flexible Body Modify dialog box as explained in Accessing the Flexible Body Modify Dialog Box, if necessary.
2. Clear the selection of **LBRF**.
3. In the **Datum Node** text box, enter the number of the desired node.

Tip:	To select a node from the screen, right-click the Datum Node text box, and then select Pick Flexbody Node. Select the node from the screen. The node number appears in the Datum Node text box.

4. Select **OK**.

Filtering Modes

You can select a filter type to remove modes from the animation display. By default, all enabled modes are used to generate nodal displacements for each flexible body during animations. To increase animation performance, Adams Flex has three filters that let you remove graphically insignificant modes for animations. A mode that is filtered out is excluded from the modal superposition and any contribution to the deformation of the body is ignored. Note that these modes are not filtered out for numeric operations, such as signal processing or xy plotting.

To create a filter:

1. Display the Flexible Body Modify dialog box as explained in Accessing the Flexible Body Modify Dialog Box, if necessary.
2. Select **Mode Filter**.

   The Flexible Body Mode Filter dialog box appears.
3. In the **Flex Body** text box, enter the name of the flexible body.
4. Set **Filter Modes** By to the desired mode:
   - **None** - Includes all modes for computing the graphics display.
   - **Frequency** - Excludes any mode that is activated above the specified frequency.
   - **Min Displacement** - Excludes any mode that does not contribute the minimum displacement specified for at least one vertex of the flexible body. For example, if you are viewing the animation of a vehicle driving down the road, it is unlikely that you would be able to see deformations of 0.5 mm or less. Therefore, if you set a mode filter value of 0.5, any mode that contributes less then 0.5 is considered insignificant and is ignored for animations. This calculation is performed at each frame of the animation, allowing the set of significant modes to change throughout the simulation.

- **Percentage** - Determines the maximum displacement contributed by all modes, and excludes any mode that doesn't contribute displacement of one vertex at least as significant as a percentage of the maximum. For example, setting the percentage filter at 15% excludes any mode not contributing at least 15% of the most dominant mode. This calculation is performed for each frame of the animation, therefore, allowing the set of significant modes to change throughout the simulation.

5. In the **Filter Value** text box, enter the frequency, minimum displacement, or percentage for the specified filter.
6. Select **OK**.
7. Animate the flexible body to see the changes.

Animating After a Flexible Body Swap

Caution should be taken when animating prior analyses after substituting a body with a flexible body via either the Rigid-to-Flex swap or the Flex-to-Flex Swap capability. See Replacing Existing Bodies with Flexible Bodies for details on preforming such substitutions.

If one or more analyses which reference the original representation of the substituted body exists in the session, the body will not appear correctly in subsequent animations of those analyses. This is because only the latest representation of that body is stored in the session. So, the visual deformation and any contour plot coloring will no longer be accurate.

The animated motion and color contours of the rest of the model's bodies will continue to be accurate. Also, any XY-plotting of result sets, requests, or measures will continue to be accurate.

Examining Numeric Results

Adams View provides the capability to measure specific attributes of flexible bodies, as well as attributes of markers and Points on flexible bodies. For information on how to use and view measures, see About Measures in the Adams View online help.

The measures available for flexible bodies are:

- Center of mass position
- Center of mass velocity (translational and angular)
- Center of mass acceleration (translational and angular)
- Potential energy delta
- Kinetic energy (total, angular, and translational)
- Strain energy (due to deformation)
- Momentum (translational and angular)

Markers and points on a flexible body have standard measurable quantities. These quantities are:

- Force (point and location)
- Torque (point and location)

- Acceleration (translational and angular)
- Displacement (translational)
- Velocity (translational and angular)

In addition, flexible body markers have these measurable quantities not available for rigid bodies:

- Deformation (translational and angular)
- Deformation velocity (translational and angular)
- Deformation acceleration (translational and angular)

Note:	Adams View returns results for all deformations relative to the LBRF regardless of whether or not you set a deformation datum node as explained in Specifying a Deformation Datum Node.

Modeling Distributed Loads and Predeformed Flexible Bodies

Overview

When working with flexible bodies, you will often find that Adams standard forces and torques are inefficient for defining distributed loads acting on a large number of nodes. Adams Flex, therefore, provides you with methods for modeling distributed loads and predeformed flexible bodies.

Select a topic for scenarios for when modeling distributed loads and preloaded flexible bodies is helpful and an introduction of the Adams modal force (MFORCE), which you can use to define distributed applied loads. There are also tutorials so you can try out these features, and explanations of how to create a loadcase file.

For general information on creating and managing MFORCEs and preloads in Adams View and Adams Solver, see:

- Working with Modal Forces in the Adams View online help
- MFORCE in the Adams Solver online help

Examples Illustrating Applied Loads and Preloads

To illustrate the ways in which you model applied loads and preloads, we've provided two user scenarios. One is an example of a distributed applied load and the other is an example of a preloaded flexible body.

- Example 1: Distributed Applied Load - Thermal Expansion
- Example 2: Preload - A Modal Tire

Example 1: Distributed Applied Load - Thermal Expansion

A flexible component that undergoes a change in temperature expands or contracts. A finite element analysis (FEA) program determines the associated thermal loads based on the material properties of the component. Adams View, however, does not possess finite element capabilities and, therefore, cannot determine these loads. The thermal load has components acting on each node of the finite element model. Introducing this load to Adams as a collection of point forces is impractical and inefficient.

The alternative is to express the load in terms of the modal degrees of freedom (DOF) rather than nodal DOF. After the FEA program prepares a component mode description of the flexible component for Adams, it takes any user-defined loadcases, such as the thermal loadcase, and transforms them from nodal DOFs to modal DOFs by projecting the load vector on the mode shapes. It then exports the load to Adams in the form of modal loads, one set of modal loads for each user-defined loadcase.

In Adams Flex, you can scale the modal loads corresponding to a particular loadcase and combine them with other loadcases. You can make the scale factor a function of time or of system response. In the example of a flexible component undergoing temperature change, you might scale the thermal load as a function of simulation time, of the velocity of the object, or of the proximity to a heat-source, and others.

Other sources of distributed loads include the aerodynamic lift on a flexible airplane wing or the combustion force on the dome of a piston. You can also model these distributed loads using an FEA program and export them to Adams Flex, where you can combine them in many different ways.

For a more detailed technical description, see MFORCE in the Adams Solver Help.

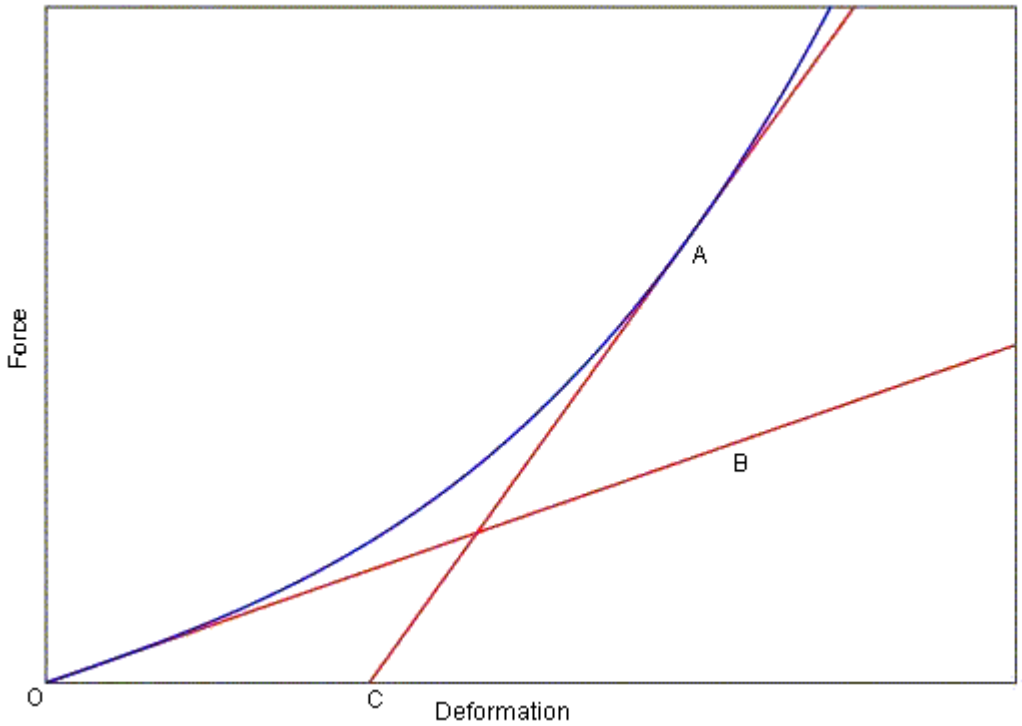
Example 2: Preload - A Modal Tire

An automobile tire has highly nonlinear material properties and experiences large nonlinear deformations. For these reasons, you would not normally consider modeling a modal representation of the tire in Adams Flex. However, in certain Adams analyses, you can safely assume that the deformations of the tire will remain within a small range around a fixed operating point and a linearization of the tire about this operating point would yield a useful modal representation of the tire. A nonlinear finite element model of the tire is brought to this operating point by applying a combination of a contact force, inflation pressure, and a spin. The tire is linearized at the operating point and the modes are extracted and exported to Adams Flex.

The figure below illustrates the process described above. An undeformed tire is defined by operating point O. As it deforms, it is brought through a nonlinear path to a deformed state A. A linearized description of the tire at O, such as the one earlier versions of Adams Flex would have been limited to, would incorrectly predict a deformed operating point at B rather than A. Using the preload support in Adams Flex, it is now possible to define an alternative linearized description of the tire, at A rather than O. This Adams flexible body might be quite suitable for an analysis of a tire *near* operating point A. A limited range of validity of the linearized representation must still be observed. For example, fully unloading the Adams flexible body that represents A would bring it to operating point C, which is not correct.

You may also want to define predeformed flexible bodies when working with models of linear elastic components. For example, it can be more convenient to define a valve-train model when a valve-spring has been predeformed so that its size matches its configuration when the components of the model have been assembled.

Tire Model Linearized at Two Different Operating Points



To further distinguish the two forms of loads, consider that preloads have been applied to the flexible component in the FEA program before it is exported to Adams. The applied distributed loads, while formulated in the FEA program, are not applied in the FEA program. Preloads become a permanent part of the flexible body description and cannot be deactivated. Distributed applied loads are exported to Adams where you can scale, combine, and apply them to the flexible body as necessary. You export both types of loads through the Modal Neutral File (MNF).

Viewing Modal Forces

You can review modal forces on flexible bodies in Adams PostProcessor as:

- Curves
- Contour plots
- Vector plots

No matter what form, the modal force results are presented with respect to the flexible body's local part reference frame. This is unlike most other Adams force elements that are plotted with respect to the ground coordinate system, by default. For a detailed overview of modal forces and a tutorial that steps you through an example of creating a modal force, see Modeling Distributed Loads and Predeformed Flexible Bodies.

Notes:

- To create a contour or vector plot of a modal force, the MNF of the associated flexible body must contain nodal masses. You can use the MNF browser to check if the MNF contains nodal masses, see Browsing an MNF or an MD DB.
- Because modal forces can depend on the state of the system, you must run a simulation before viewing the results of a modal force.

To review a modal force component as a curve:

1. Start Adams PostProcessor, and then set its mode to **plotting**.
2. From the Dashboard, set **Source** to **Result Sets**.

   The dashboard changes to show the results available for plotting.
3. From the **Result Set** list, select the modal force object whose characteristics you want to plot.
4. From the **Component** list, select the component of the modal force. FX, FY, FZ, TX, TY, and TZ are the resultant force and torque components with respect to the flexible body's local part reference frame.FQi is the ith modal component of the modal force.
5. Select **Add Curves** to add the data curve to the current plot.

To review a modal force as a contour plot:

1. Set the Adams PostProcessor mode to **animation**.
2. Right-click the background of a viewport, and then select **Load Animation**.
3. From the Treeview in Adams PostProcessor, select the flexible body on which you want to display the modal force plot.
4. In the Property Editor, set **Plot Type** to **Both**.
5. In the dashboard, select the **Contour Plots** tab.
6. Set **Contour Plot Type** to the component of the modal force you want to review. Remember that the modal force components are with respect to the flexible body's local part reference frame.

   Next, Adams PostProcessor computes the minimum and maximum values of the modal force. This can take a few minutes because it requires interrogating the modal force values at every node in every mode at every animation frame.
7. Select the **Play** button to animate the modal force contour plot.

To review a modal force as a vector plot:

1. Follow Steps 1 through 4 in **To Review a modal force as a contour plot** above.
2. Select the **Vector Plots** tab in the dashboard.
3. Set **Vector Plot Type** to either **Force** or **Torque**.
4. Select the **Play** button to animate the modal force contour plot.

Tutorials

The following tutorials let you try out the features helpful when modeling distributed loads and preloaded flexible bodies.

- Example of Modal Loads
- Example of Working with a Preloaded Flexible Body

Creating Loadcase Files

The MFORCE distributed load and modal preloads capabilities assume that additional values are stored in the flexible body's Modal Neutral File (MNF). Currently, few finite element analysis (FEA) programs that support Adams Flex provide this capability. As an interim solution, MSC Software has developed a tool, mnfload, which adds applied modal loads and preloads to existing MNFs. Support for these capabilities from FEA programs will be announced as it becomes available.

Note:	The effect of mnfload to MNF is cumulative. Subsequent invocations of mnfload will not replace or override but add to a modal load case if it already exists.

Learn more on how to use mnfload:

- Executing mnfload
- Syntax of Loadcase File
- Example of Loadcase File

Executing mnfload

You invoke mnfload from the command line using the command:

On Linux

```
adams2021_2 -c flextk mnfload existing.mnf new.mnf loadfile
```

On Windows

```
adams2021_2 flextk mnfload existing.mnf new.mnf loadfile
```

where:

The Parameter:	Specifies:
existing.mnf	Existing MNF to which you want to add load data.
new.mnf	MNF to be created which combines the MNF and load data.
loadfile	An ASCII text file containing a concatenation of loadcases. For more information, see Syntax of Loadcase File.

Syntax of Loadcase File

The variable loadfile in the command to invoke mnfload is an ASCII file containing a concatenation of loadcases. Each loadcase starts with the identifier line:

```
% key load_case_label
```

where:

The parameter:	Specifies:
% (percent symbol)	Start of new load definition. Percent sign is the first character in the line.
key	Either: ■ PC - Cartesian preload (nodal coordinates) ■ PM - Modal preload (modal coordinates) ■ C - Cartesian-applied load ■ M - Modal-applied load
load_case_label	text string identifying the content of the loadcase.

Note that there can only be one preload and it must be a balanced load. There can be no global resultant load components. In other words, the preload must be an internal load.

Each loadcase identifier line is followed by lines defining either nodal forces and torques or modal loads. When defining the nodal force, the format of the lines are:

```
node_id dof value
```

where:

The parameter:	Specifies:
node_id	One of the nodes in the flexible body.
dof	Either FX, FY, FZ, TX, TY, TZ.
value	Force value in the same units is the MNF of the flexible body to which the force will be added.

When defining a modal load in modal coordinates the format of the load lines is:

```
mode_number value
```

where:

The Parameter:	Specifies:
mode_number	Number of a mode.
value	Load that is to be exerted on this mode.

Example Loadcase Files

The following file defines three loadcases:

- Cartesian preload
- Applied load in cartesian coordinates
- Second applied load in modal coordinates

Note that the preload is invalid unless nodes 1000 and 1001 have the same y and z coordinates; otherwise this would not be a balanced load.

```
%PC this string is ignored

1000 FX 1e5
1001 FX -1e5
%C push the body in the X direction by forces on 1000 and 1001
1000 FX 1e5
1001 FX 1e5
%M load modes 6 and 10
10 1.0e5
6 1.5e4
```

Note:	When a preload is applied to an MNF, the mnfload tool attempts to deform the geometry corresponding to the load.

Translating FE Model Data

Introduction

Generating a Modal Neutral File (MNF) from a finite element (FE) model requires that you perform a special analysis on the model in the finite element analysis (FEA) program and then translate the output of the analysis to an MNF. Generating an MNF is usually performed by a finite element analyst or an Adams user with experience running a finite element program. Once you generate the MNF, you can distribute the MNF to other users for use in Adams Flex.

The entries in this section describes the aspects of a finite element model you should consider before you generate an MNF, and explain the steps you perform to generate an MNF using different FEA programs.

They assume that you know how to run Adams Flex, Adams View, and Adams Solver. They also assume that you have a moderate level of finite element modeling proficiency. You need access to an FEA program that interfaces with Adams, so you can prepare an FE model for use with Adams and create an MNF to be built into your Adams model.

FE Model Requirements

You can use any finite element (FE) model of a component as the foundation for an Adams Flex flexible body. The following describes aspects of an FE model that you should consider when you transfer it for use in Adams. It starts by listing the vendors that support MNF translation.

- FEA Programs and MNF Translation
- Node Number Limitation
- Attachment Points
- Mode Selection
- Units
- Constraints and Rigid Body Modes

FEA Programs and MNF Translation

The following table list the FEA vendors and their available Modal Neutral File (MNF) functionality.

FEA Vendors and MNF Translation

	MSC Nastran	MSC Marc	ANSYS	ABAQUS	I-DEAS	PERMAS
CMS	✓	✓	✓	✓	✓	✓
Interface nodes	✓	✓	✓	✓	✓	✓
Orthogonalization	✓		✓		✓	
Modal Loads	✓	✓	✓			

	MSC Nastran	MSC Marc	ANSYS	ABAQUS	I-DEAS	PERMAS
Residual Vectors	✓					
Preloads	✓	✓		✓		
Stress Modes	✓	✓	✓	✓	✓	*
Generalized damping	✓					

*Solid elements only. PERMAS is a European finite element code.

Node Number Limitation

There is no limit on the number of nodes in the FE model. The number that you select, however, can affect data storage, transfer requirements, and the rendering performance of the graphics hardware. For example, an FE model with more than 20,000 nodes taxes the capabilities of video subsystems during animation and requires approximately 1 Mb of disk storage for each modal degree of freedom (DOF) you retain from the finite element analysis.

You should note, however, that during the Dynamic simulation:

- Adams uses a modal description of the flexible body.
- Computational speed is completely independent of the number of finite element nodes in the body.

Attachment Points

When you build a flexible body into an Adams model, you interface with the body using a variety of attachments, either joints or forces. In Adams Flex, you can model the variable boundary conditions at attachment points, which are nodes that have been idealized for attachment, by preserving all six Cartesian degrees of freedom (DOFs) of those points as you export the flexible body from your finite element analysis (FEA) program. An attachment point is equivalent to a superelement exterior grid point.

The FEA of the component is usually performed without detailed information about external constraints. These boundary conditions are often an unknown function of time. If the Adams analysis could only accommodate normal modes (eigenvectors), the number of modes that would be required to model the effect of attachments would be dramatically increased. With prior knowledge of nodes at which joint or force elements are applied to the body, Adams can achieve a high-fidelity solution with a minimal number of modes.

Attachment points are not without drawbacks, however. Each attachment point normally contributes six modal DOF. Corresponding to each attachment point DOF is a constraint mode, which is a static mode shape due to a unit displacement of that DOF while holding all other DOFs of all attachment points fixed. A large number of attachment points can result in unwieldy data files and can significantly impact CPU time, if the associated modes are enabled during an Adams dynamic simulation. You should note that you can always apply joints and forces to any node without it having been identified as an attachment point during the FEA.

The ability to capture structural interactions at attachment points is accomplished by applying a component mode synthesis method similar to that proposed by Craig and Bampton ("*Coupling of Substructures for*

Dynamics Analyses", Craig, R.R. and Bampton, M.C.C., 1968 AIAA Journal, Vol.6, No.7, pp.1313-1319). The Craig-Bampton modes are the constraint modes mentioned earlier, plus a set of fixed interface normal modes, the eigenvectors of the system, while holding all attachment point DOFs fixed.

The modes Adams Flex uses are a slightly modified version of the Craig-Bampton modes, which are better suited to modeling large rigid body motion. These modified Craig-Bampton modes also give you additional information about the frequency contribution of individual modes and allow more freedom during mode selection. For a detailed technical description of the modified Craig-Bampton modes, see the Theory of Flexible Bodies.

Mode Selection

You can categorize the modified Craig-Bampton component modes that Adams Flex uses into two different types:

- Eigenvectors of the component
- Eigenvectors of the boundary (the attachment point DOFs)

These modes are obtained by orthogonalizing the Craig-Bampton modes. Any one of these modes can be enabled or disabled during the dynamic simulation. You should disable a mode if it does not contribute to the response of the flexible component during a simulation. You should not, however, base the inclusion of a mode solely on its resonant frequency and the frequency of the excitation. For example, a static load can cause the deformation of the component in such a way that can only be captured by the shape that corresponds to a particular high frequency-mode. You should only disable modes because of unacceptable computational overhead or when you are certain that a particular mode does not contribute to overall response. You should never disable modes based on an arbitrary notion of a frequency range of interest.

Units in MNFs

When you generate a Modal Neutral File (MNF), you must specify which units were used in the FEA program. This units information is stored in the MNF. You should note that the finite element analysis does not need to have been performed in the same set of units as subsequent Adams simulations. As long as the MNF is labeled with the proper units, Adams can convert data to correctly represent the flexible body in the Adams model.

Some finite element programs do not accommodate inconsistent sets of units (for example, millimeter, Newton, kilogram, and seconds). You can still perform your finite element analysis in these units but you should be aware of the strange behavior that can occur, such as frequency being reported in units that are not Hz. If the data in the MNF is labeled with the correct units, Adams handles the data correctly.

Constraints and Rigid Body Modes

You should avoid using constraints in the FE model. Only in rare cases are constraints appropriate. For example, a constraint is useful when a particular node on the component is fixed on ground in the Adams model. An improperly constrained finite element model can seriously misrepresent the component. Consider, for example, an FE model in which two separate nodes are completely constrained. These two nodes never

move relative to each other during the Adams simulation because none of the mode shapes feature any relative motion between the two nodes.

When you read modal information for an unconstrained component into Adams, you must ensure that all rigid body modes are disabled. Adams adds six nonlinear rigid body DOF to each flexible body and numerical singularities arise if the rigid body mode shapes are also included. Adams Flex attempts to disable rigid body modes by default, but Adams View can fail to detect some rigid body modes because rigid body modes sometimes have nonzero frequencies caused by numerical inaccuracies.

Setting Up Translation Options through the MNF Toolkit

You can set up the translation of FE data using the MNF toolkit.

Learn more:

- About the MNF Toolkit
- Automatic Interior Geometry Removal
- Automatic Modal Load (MFORCE) Nodes Optimizer
- Automatic Mesh Coarsening
- Manual Mesh Simplification
- Invariants Computation
- Optimized Units
- Single-Precision MNFs
- Sparse Stress and Strain Modes
- Rigid-Only MNFs
- High- and Low-Pass Frequency Filter

Note:	All environment variable examples are shown for the C-shell. Make the appropriate changes for a Bourne shell derivative.

About the MNF Toolkit

The MNF toolkit is a library of functions that assists third-party developers of FE model translators. MSC Nastran, MSC Marc, ANSYS, I-DEAS, and ABAQUS use the MNF toolkit to generate MNFs. The MNF toolkit is highly flexible and easy to configure using environment variables.

Because the MNF toolkit is embedded in third-party products, you communicate with it through the MDI_MNFWRITE_OPTIONS environment variable, shown below, where *options* is a set of options you can set as explained in the list below:

```
MDI_MNFWRITE_OPTIONS options
```

You can combine the options in any order with no particular separation of the keywords. For example, you can enter:

```
setenv MDI_MNFWRITE_OPTIONS "fast_invar, MKS, strip_face"
```

Automatic Interior Geometry Removal

To enhance graphics performance, the MNF toolkit can recognize certain interior geometry, such as the mated faces of two brick elements, and remove it. When used with invariants computation (explained on subsequent help topics), removing interior geometry can significantly reduce the size of an MNF.

To turn on the removal of interior geometry, type the following in a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "strip_face"
```

Note that when you use strip_face with the option for invariants computation, Adams Flex removes FE model nodes that were only connected to geometry it also removed. Occasionally, you may find this undesirable when a particular interior node is to be the target for an attachment in Adams. To prevent node removal, connect the node with exterior nodes (for example, using zero stiffness, zero mass, and beam elements).

Automatic Modal Load (MFORCE) Nodes Optimizer

Allow the MNF Optimizer to remove nodes that have modal loads defined. All modal load nodes are retained by default so that detailed MFORCE plots can be generated in View. This may result in exported MNF that are larger than expected in size when modal loads are defined. If detailed MFORCE plotting is not necessary and smaller MNF is required, then you can set this option to also eliminate modal load nodes when optimizing the MNF.

To turn on the modal load nodes optimizer, type the following in a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "optimize_mforce_nodes"
```

Automatic Mesh Coarsening

Finite element (FE) models used with Adams frequently contain greater detail than that required for dynamic simulations. The MNF toolkit can coarsen the mesh without losing significant geometric features. When you use the mesh coarsening option with the invariants computation option, mesh coarsening can significantly reduce the size of the MNF.

For an example of mesh coarsening, see Knowledge Base Article:

http://simcompanion.mscsoftware.com/KB8016333

To turn mesh coarsening on, type the following at a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "coarsen(res,ang,colin,N1,N2,...)"
```

where:

res	The desired resolution, which is the fraction of the total component size below which Adams Flex removes the detail of the mesh. For example, if your component is approximately 1 meter long, and you select 15% mesh resolution, the coarsening results in a mesh with 15 cm-wide mesh cells. You enter the percentage as a fraction. Example of Mesh Resolution.
ang	The angle tolerance for parallel faces, in degrees, below which Adams Flex merges faces. For example, if you select 15, the coarsening algorithm does not merge two faces when one face is more than 15o out of the plane of the other face. Example of Face Smoothing.
colin	1 if you want to remove the nodes that are intermediate nodes on the straight edge of a face, or 0 if you do not want to control the nodes. Example of Collinear Point Removal.
N1,N2,...	List of nodes that Adams Flex should **not** remove. Only visible nodes can be used to place markers, joints, and so on, onto the flexible body. Therefore, make sure to include relevant nodes.

For example, if you want res to be 20% and ang to be 15 degrees, the syntax is:

```
setenv MDI_MNFWRITE_OPTIONS "coarsen(0.20,15,0)"
```

Note that you must enter the percentage as a fraction and it must be within the accepted range:

- Mesh Resolution, res: 0.0 to 0.50
- Angle Tolerance, ang: 0.0 to 45.0 degrees

If you exceed the limits of this range, Adams Flex uses the default values of 0.02 for res and 10 degrees for ang.

Example of Mesh Resolution

In the following example:

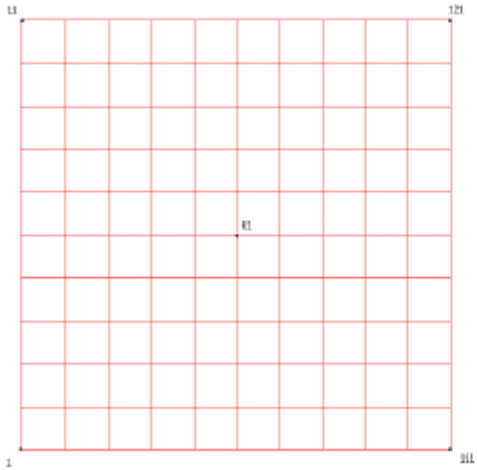
- Diagonal dimension of original mesh = 1000 * $\sqrt{2}$ = 1414.21 mm

Mesh coarsening resolution is set to 15% = 1414.21 * .15 = 212.13 mm

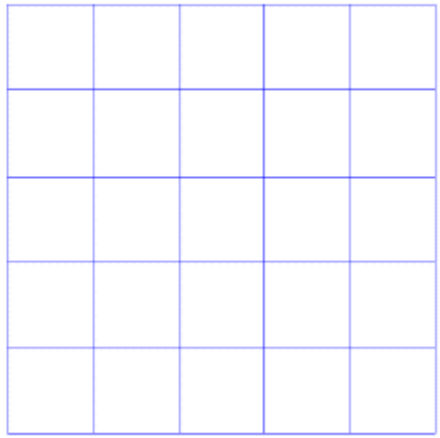
Therefore, any element whose diagonal is smaller than 212.13 mm is combined with its neighboring element to produce a larger element. Multiple elements are combined to satisfy this parameter. The result is a coarser mesh.

For this example, 4 (100 mm * 100 mm) elements were combined to produce a larger element with a diagonal of 282.84.

Original Mesh



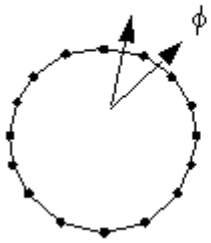
Mesh Coarsened 15%



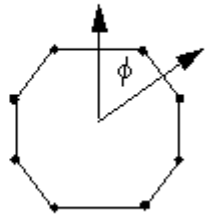
Example of Face Smoothing

During the coarsening algorithm, the adjacent elements in the following example were merged together if the resulting angle, phi, was less than 45 degrees.

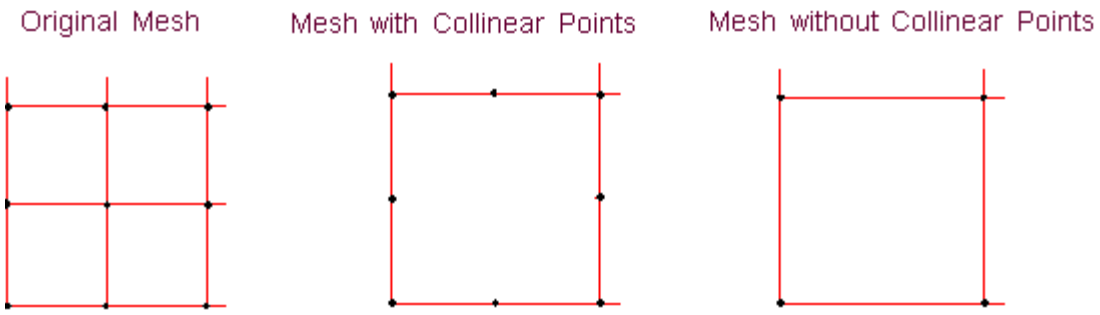
Original Face



Face Smoothed 45 degrees



Example of Collinear Point Removal



Manual Mesh Simplification

You can also manually reduce the mesh down to a user-defined sketch of the mesh using the MNF toolkit. If the visual representation of the flexible body is less important than animation speed or file size, then this technique may be ideal. Note that the MNF toolkit only eliminates disconnected nodes from the MNF if you also requested precomputation of invariants. Therefore, to reduce the size of the Modal Neutral File (MNF), you must both coarsen the mesh and request that the invariants be computed. In addition, remember that you can only place markers on nodes; therefore, any flexible body node where you might place a marker must be represented in the simplified mesh.

You use a sketch file to describe the mesh. The format of the sketch file, which describes the mesh as a collection of faces, must be as follows:

```
face_count
face_1_node_count face_1_nodeid_1 face_1_nodeid_2 ...
face_2_node_count face_2_nodeid_1 face_2_nodeid_2 ...
<etc>
```

Faces must have a node count of at least two. For example, a mesh comprised of a single brick element might be described as follows:

```
6
4 1000 1001 1002 1003
4 1007 1006 1005 1004
4 1000 1004 1005 1001
4 1001 1005 1006 1002
4 1002 1006 1007 1003
4 1003 1007 1004 1000
```

Alternatively, the mesh might be described as a stick figure using a collection of lines (two node faces), as shown below:

```
8
2 101 102
2 102 103
```

```
2 103 104
2 104 105
2 105 106
2 106 107
2 107 108
2 108 109
```

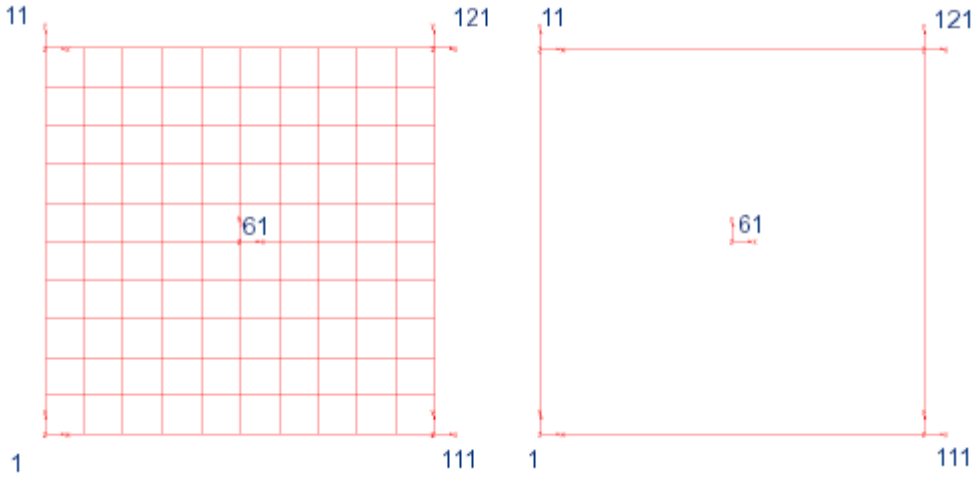
Example of manual mesh simplification

Sketch File and Result

2 Number of total faces

4 1 11 121 111 4 total nodes defining the face. The face connects nodes 1, 11, 121, and 111.

1 61 1 node defining this face. The face just contains node 61.



To turn on manual simplification, type the following at a prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "sketch(mysketch.dat) fast_invar"
```

Do not combine manual mesh simplification with any of the automatic mesh reduction methods.

Invariants Computation

The MNF toolkit lets you compute the modal flexibility inertia invariants and store them in the MNF. If the invariants are in the Modal Neutral File (MNF), the mnf2mtx translator, which translates an MNF to a matrix file, selects the required invariant components from the invariants stored in the MNF rather than recomputing them.

To turn on the computation of invariants, type either of the following at a C-shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "full_invar"
```

```
setenv MDI_MNFWRITE_OPTIONS "fast_invar"
```

where:

full_invar	Computes all nine invariants.
fast_invar	Suppresses the computation of the more computationally expensive invariants five and nine. Note that fast_invar corresponds to the Partial Coupling formulation for modal flexibility. The fast_invar setting is also suitable for use with the Constant Coupling formulation. Only Full Coupling requires all nine invariants. Unless you think you might need Full Coupling, you can safely select fast_invar. (For more information on interia coupling, see Modifying the Modal Formulation.)

Note also that when you use either fast_invar or full_invar with the internal geometry removal or mesh coarsening option described earlier, the MNF toolkit discards disconnected nodes after computing their invariant contribution, therefore, greatly reducing the size of the MNF.

Optimized Units

The output of an FEA program is labeled with the units of the data and the FE model translator, by default, transfers the labels to the Modal Neutral File (MNF) . When you use the MNF with Adams View, Adams View transfers all the data in the MNF into its internal units (SI), which are meters, kilogram, seconds, and Newtons. Adams View performs the unit scaling as it performs operations, which can significantly degrade performance. As an alternative, you can store the data in the MNF in SI units.

To store the data in the MNF in SI units, type the following at a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "MKS"
```

Single-Precision MNFs

By default, the numerical values stored in the MNF are in double precison. You can optionally generate a single-precision MNF. This reduces the MNF size by 50%, and speeds up the animation of flexible bodies.

To store the data in the MNF in single precison, type the following at a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "single"
```

Sparse Stress and Strain Modes

By default, the latest version of the MNF (version 6) only stores stress and strain components for nodes where either stress or strain has been recovered in the finite element program and exported to the MNF. Therefore, the stress and strain modes are said to be stored in sparse format. In older versions of the MNF, the stress and strain modes are stored in dense format. If neither stresses nor strains were provided by the finite element program, the MNF sets the nodal values to zero. This unnecessarily increased the size of the MNF.

If you have an MNF where the stress and strain modes are stored in dense format, you can remove the zero entries by typing the following at a shell prompt before the MNF is written:

```
setenv MDI_MNFWRITE_OPTIONS "sparse_str"
```

If you have an MNF where the stress and strain modes are stored in sparse format, and you want to store them in a dense format, type the following:

```
setenv MDI_MNFWRITE_OPTIONS "full_str"
```

Rigid-Only MNFs

Depending on the component and application, the size of the MNF can be very large, exceeding several gigabytes, and difficult to manage. If you are in the process of building your Adams Flex body model, you may consider treating the body as rigid until you are confident in how you have assembled your model. Learn about Modifying the Modal Formulation. If temporarily using a rigid body formulation for an Adams Flex body makes sense, you can drastically reduce the size of the MNF.

To set a rigid-only MNF, type the following at a shell prompt before the MNF is written:

```
setenv MDI_MNFWRITE_OPTIONS "empty"
```

This creates a reduced MNF that only contains enough information to build a rigidized flexible body. With this MNF, you cannot build an Adams Flex body with Constant, Partial, or Full modal formulations, but it may be convenient to work with while you are assembling and verifying your model. When you are confident with your model, you can easily replace the reduced MNF with the full MNF. Learn about Replacing Existing Bodies with Flexible Bodies.

High- and Low-Pass Frequency Filter

By default, a Modal Neutral File (MNF) only keeps modes with frequencies in the range of –1.0 to 1.0E+08 Hertz. The lower default threshold of –1.0 is chosen because some rigid body modes may have a slightly negative frequency due to numerical round-off. Even though they are automatically disabled in Adams View, rigid body modes are important to keep in the MNF to correctly treat the resultant components of modal forces vectors and generalized damping matrices.

You can change the frequency range of the modes kept in the MNF, by specifying the **highpass** and **lowpass** options in the MDI_MNFWRITE_OPTIONS environment variable.

To change the upper frequency threshold, enter the following in a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "lowpass(XXX)"
```

where XXX is a numerical value indicating the maximum frequency for modes in the MNF. For example, to lower the maximum frequency from 1.0E+08 to 1.0E+06, type the following at the shell prompt.

```
setenv MDI_MNFWRITE_OPTIONS "lowpass(1.0e6)"
```

To change the lower frequency threshold, enter the following in a shell prompt before running the FE model translator:

```
setenv MDI_MNFWRITE_OPTIONS "highpass(YYY)"
```

where YYY is a numerical value indicating the minimum frequency for modes in the MNF. For example, to lower the minimum frequency from –1.0 to –10.0, enter the following at the shell prompt.

```
setenv MDI_MNFWRITE_OPTIONS "highpass(-10.0)"
```

Note that the highpass and lowpass options behave like V1ORTHO and V2ORTHO on the AdamsMNF Case Control command in MSC Nastran. These options are exposed through the MNF Toolkit for those not using MSC Nastran but still would like to control the frequency content of modes in the MNF.

Translating MSC Nastran Data

There are two different interfaces that you use to translate MSC Nastran data for use in Adams Flex. Learn about:

- Using MSC Nastran 2004 and Above
- Verifying the Model
- Computing MSC Nastran Stress/Strain Modes
- MSC Nastran XDB Support for Stress/Strain Modes
- Shortened Stress/Strain Modes

Using MSC Nastran 2004 and Above

Starting in version 2004, MSC Nastran provides an improved interface for generating a Modal Neutral File (MNF) for flexible bodies in Adams. The new MSC Nastran Adams Interface allows you to generate an MNF directly from MSC Nastran without generating an OUTPUT2 (op2) file. The MSC Nastran Adams Interface no longer requires a DMAP alter (alt file) or a translator to convert MSC Nastran output files to MNF.

The MSC Nastran Adams Interface is provided by the ADAMSMNF command in Nastran's Case Control section. Refer to the *MSC Nastran Quick Reference Guide and Reference Manual* for information on how to use it.

Verifying the Model

The MSC Nastran translator writes a summary of the modal neutral file (MNF) export to the terminal window. If you are using MSC Nastran 2004 or above, the Adams interface writes a summary of the MNF export to the MSC Nastran output file. Please review this data for any concerns. In particular, ensure that the:

- Mass, center of mass location, and moments of inertia are as expected.
- During the MNF write, the constraint modes and the constrained normal modes are orthogonalized. This yields modes that are:
  - An approximation of the free-body normal modes.
  - Interface modes, where the interface is the collection of all the attachment point DOFs.

Also, verify that the free body normal modes have a reasonable natural frequency. You should expect to see six rigid-body modes, unless displacement boundary conditions are present.

Computing MSC Nastran Stress/Strain Modes

For Adams Durability to process stresses or strains on flexible bodies, modal stress or strain shapes need to be present in the modal neutral file (MNF) of the flexible body. You do this by having MSC Nastran recover a stress or strain mode for every mode shape computed for Component Mode Synthesis (CMS).

- MSC Nastran Grid Point Stresses
- Example
- Known Limitations, Problems, and Restrictions

MSC Nastran Grid Point Stresses

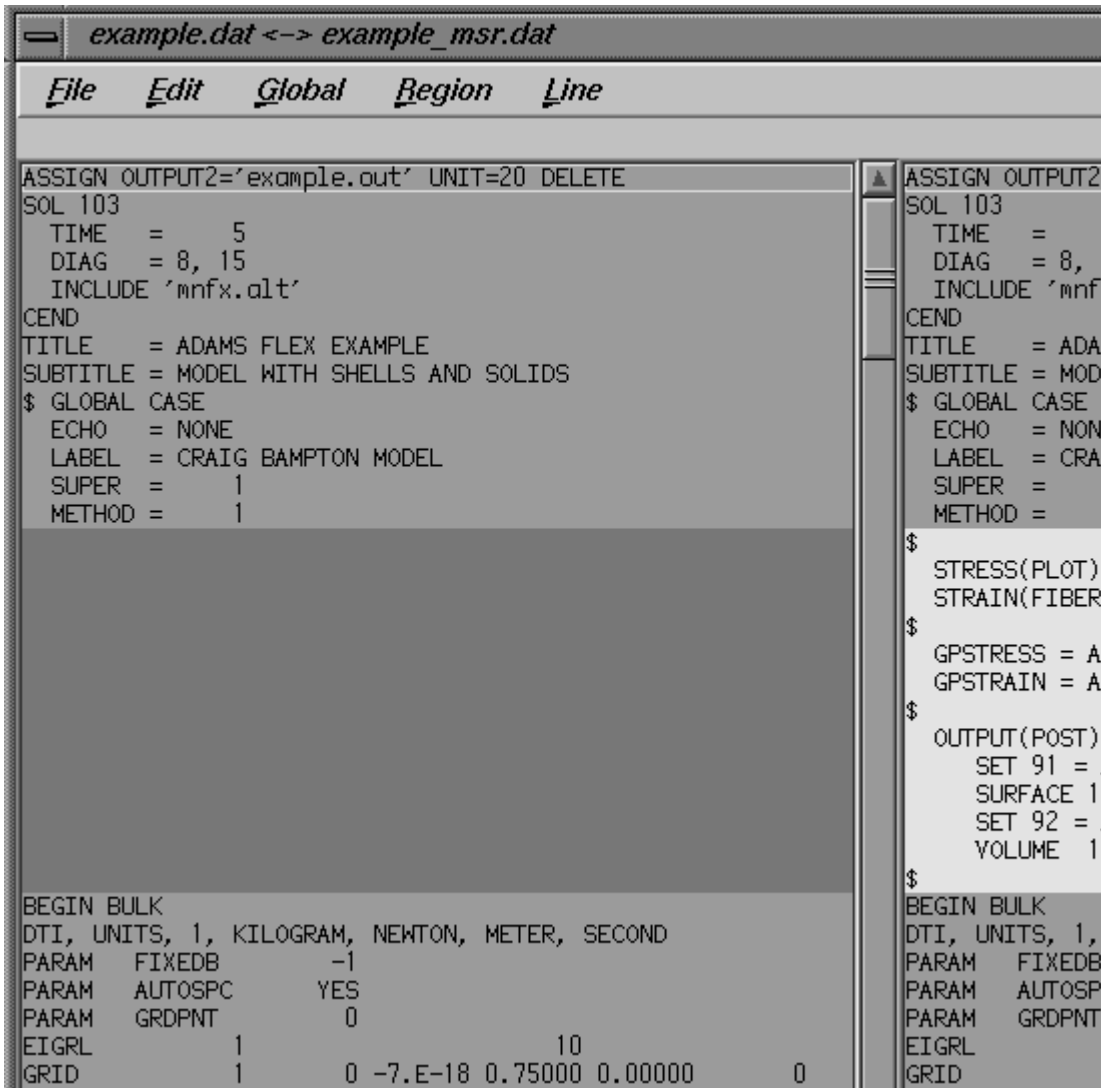
Because modal information contained in the MNF can only be associated with nodes, the MSC Nastran grid-point stress data recovery option is required. The following Case Control commands are required in the MSC Nastran input file to compute stress or strain modes for the MNF:

- GPSTRESS: Requests grid point stresses output.
- GPSTRAIN: Requests grid point strains output.
- STRESS(PLOT): Requests element stress output.
- STRAIN(FIBER,PLOT): Requests element strain output.
- OUTPUT(POST): Delimiter.
- SET: Defines a set of elements for a surface or volume.
- SURFACE: Defines a surface of plate elements referenced by the SET command.
- VOLUME: Defines a volume of solid elements referenced by the SET command.

For more information on these commands, see the Case Control section of the *MSC visualNastran Quick Reference Guide*. For more information on computing grid point stresses, see the *MSC Nastran Linear Static Analysis User's Guide*.

Notes:	You can only transfer one surface stress or strain fiber of plate elements to the MNF for processing in Adams. If more than one fiber is specified on the SURFACE card, the msc2mnf translator issues a warning message and only transfers the first surface stress fiber it finds in the OUTPUT2 file. Including stress or strain modes in the MNF can significantly increase the file size. Therefore, it becomes even more important to optimize the MNF if possible. For information on optimizing the MNF, see Optimizing an MNF or an MD DB. Including both stress and strain modes will further increase the size of the MNF and is generally not recommended for large models, unless both quantities are needed. When defining subcases in Case Control, you must have the GPSTRESS, GPSTRAIN, STRESS, and STRAIN cards before the first SUBCASE card. In addition, the OUTPUT, SURFACE, and VOLUME cards should follow all subcase definitions and appear at the end of the Case Control.

Example



Example above shows the changes that are required in the MSC Nastran input file when the computation and transfer of both stress and strain modes are desired. Because the model contains solid and shell elements, a surface and a volume are defined for computing these grid-point stresses and strains. The surface fiber selected is Z1 and the grid-point stress/strain coordinate system is consistently defined to be the basic FE model system.

Known Limitations, Problems, and Restrictions

- Only one FIBER is output on SURFACE.
- SURFACE or VOLUME should be defined in consistent coordinate (basic) system.

MSC Nastran XDB Support for Stress/Strain Modes

You can store the ortho-normal stress and/or strain modes in XDB file format that are compatible with the mode shapes in the modal neutral file (MNF) and subsequent modal responses from an Adams simulation. The benefits of this capability are:

- **Unlimited model size** - MSC Patran can access results from an XDB file of any size and with much more efficiency than from an OP2 file.
- **MSC Fatigue analysis** - Modal coordinates from Adams can be combined with stress or strain modes in XDB file for very efficient MSC Fatigue analysis using modal superposition.
- **Element-based support** - The XDB file format supports element-based and/or grid-point based stress or strain. Element-based results allow you to perform advanced fatigue analyses such as multi-axial fatigue and weldments.

Learn more:

- Creating an XDB File
- Limitations
- Examples

Creating an XDB File

To create an XDB file with stress or strain modes, add the following entry in the Bulk Data section:

```
PARAM,POST,0
```

This is in addition to the necessary commands that are added to Case Control (see Computing MSC Nastran Stress/Strain Modes). In the case of grid point stresses or strain, however, one additional command is required to output grid point stress or strain modes:

```
STRFIELD = ALL
```

Note that if you are only interested in working with element-based stress or strain, this command is not needed. For more information on these entries and commands see the, MSC Nastran Quick Reference Guide.

Limitations

- Grid-point strain modes cannot be stored in the XDB nor can MSC Patran post-process them.
- Element-based stress or strain modes cannot be stored in the MNF nor can Adams Durability postprocess them.

Examples

The following are examples of MSC Nastran input decks. See the Case Control section of the MSC Nastran Quick Reference Guide for more information on the AdamsMNF command that is being used in these examples.

Example of Requesting No Grid Point Stress/Strain

In the following example, no grid point stress or strain modes have been requested. Only element-based strain modes have been requested with STRAIN(PLOT) = ALL. These strains will be stored in the XDB (PARAM,POST,0) for postprocessing in MSC Patran or for combining with Adams modal responses from Adams Durability for an MSC Fatigue analysis. This is the most efficient process for obtaining strains for the sole purpose of performing a fatigue analysis. If you are not interested in viewing strains in Adams, there is no need to compute grid-point strain modes nor storing them in the MNF. You will also seea savings in file size and processing time from this. The same is true for stress modes if they are desired over strains.

```
SOL 103
CEND
AdamsMNF, FLEXBODY=YES, OUTGSTRS=NO, OUTGSTRN=NO
...
STRAIN(PLOT) = ALL
...
BEGIN BULK
...
PARAM,POST,0
...
ENDDATA
```

Example of Requesting Grid-Point Stress on All Solid Elements

In the following example, grid-point stress (GPSTRESS) modes have been requested on all solid elements (VOLUME). This data, as well as the element-based stress (STRESS) modes, will be stored in the XDB due to the STRFIELD=ALL command and the PARAM,POST,0 card. The grid point stress modes will also be stored in the MNF with the OUTGSTRS=YES option set on the AdamsMNF command. This allows Adams Durability to postprocess stresses on the flexible body in Adams using the modal stress recovery technique.

```
SOL 103
CEND
AdamsMNF, FLEXBODY=YES, OUTGSTRS=YES, OUTGSTRN=NO
STRFIELD = ALL
...
STRESS(PLOT) = ALL
GPSTRESS = ALL
OUTPUT(POST)
  SET 92 = ALL
  VOLUME 12 SET 92 DIRECT
BEGIN BULK
...
PARAM,POST,0
...
ENDDATA
```

Example of Requesting Grid-Point Stress

In the following example, again, grid-point stress modes have been requested. They will not be stored in the XDB, however, because the STRFIELD=ALL command is missing. Therefore, only element-based stress modes will be available in the XDB. Grid-point stress modes will be stored in the MNF because the AdamsMNF option, OUTGSTRS is still set to YES.

```
SOL 103
CEND
AdamsMNF, FLEXBODY=YES, OUTGSTRS=YES, OUTGSTRN=NO
...
STRESS(PLOT) = ALL
GPSTRESS = ALL
OUTPUT(POST)
  SET 92 = ALL
  VOLUME 12 SET 92 DIRECT
BEGIN BULK
...
PARAM,POST,0
...
ENDDATA
```

Shortened Stress/Strain Modes

Shortened stress/strain modes refers to the capability of defining a group or subset of elements in FEA for stress/strain recovery during modal neutral file (MNF) generation. FEA programs allow you to judicially define subregions of your component where stress/strain is of interest. If these subregions are defined during MNF generation, the node length of the stress/strain modes becomes shorter than that for the mode shapes. This reduces the amount of stress/strain data in the MNF, and allows you to avoid doubling the file size when including stress or strain modes. Adams Durability, however, will only be able to recover stress or strain at those subregions.

Support for this capability was first introduced in version 2005. Before 2005, a null tensor (all zero values) would be stored in the MNF for those nodes that did not have stress/strain computed by the FEA program. No reduction in file size was obtained, but worse yet, Adams Durability would report zero stress/strain for those nodes, which could be misleading. In Adams Flex 2005 or greater, it is now possible to remove these zero stress/strain states during MNF optimization. More information on how to do this is provided in the next sections.

Starting in MSC Nastran 2005, only grid point stresses that are computed for a subset of the component are output to the MNF. Support for this capability by the other FEA programs is not yet available.

Learn more:

- Note on MNF Compatibility
- MNF Translation and Optimization
- Version Scenarios

Note on MNF Compatibility

In general, an MNF is upward, but not necessarily backward, compatible. Adams will always support earlier versions of the MNF. For example, an MNF generated in a version of MSC Nastran before 2005 will be supported. However, an MNF generated by MSC Nastran 2005 or later will be incompatible in a version of Adams earlier than 2005. This is because, by default, MSC Nastran generates a version of the MNF that supports shortened stress/strain modes, or in other words, a reduced MNF. However, an option exists in Adams Flex to convert a reduced MNF to a full MNF, so that it can be processed by earlier versions of Adams.

MNF Translation and Optimization

Support for shortened stress/strain modes is available in the **Adams Flex MSC->MNF Translator** and **MNF->MNF Optimizer** through the menu option **Stress & Strain Modes**. Three options are available as listed in the table below.

The option:	Does the following:
Sparse	Stores modal stress/strain information only for those nodes for which MSC Nastran had computed stress/strain and stored in the OUTPUT2 file. During MNF optimization, allows you to maintain a reduced MNF. If the original MNF is full, has no effect during optimization. It also has no effect on an MNF that already has shortened stress/strain modes.
Remove zero entries	Allows you to shorten the stress/strain modes in an MNF by removing stress or strain entries that contain all zero values. Also updates the version of the MNF, so that it is no longer compatible with a version of Adams earlier than 2005. Has no effect on a reduced MNF that supports shorten stress/strain modes.
Full	Makes the **MSC->MNF Translator** behave as before by storing modal stress information at every node. Allows you to convert a reduced MNF to one that is full by padding zero values for those nodes that do not have stress/strain defined. This option is useful even if the MNF does not contain stress or strain modes because it converts the MNF to an older version that did not support shortened stress/strain modes. For example, this allows you to convert an MNF that was generated by MSC Nastran 2005 to one that is compatible with an earlier version of Adams. This option has no effect on an older version of the MNF, that is, one that is already full.

Version Scenarios

For the version:	The scenario is:
MSC Nastran 2004	If you have an MNF with stress/strain modes that was generated in MSC Nastran 2004 and you want to reduce it, run the **MNF -> MNf Optimimzer** with the **Stress & Strain Modes** option set to **Remove zero entries**.
MSC Nastran 2005	If you have an MNF generated in MSC Nastran 2005 and want to use it in Adams 2003, first convert it by running the **MNF->MNF Optimizer** with the **Stress & Strain Modes** option set to **Full**.

Example

In this MSC Nastran example, ten shell elements (CQUAD4) are used to model a beam. Grid point strains are requested (GPSTRAIN) on only four of the elements (4,5,6,7) because of the SET 100 specification on the SURFACE card. This results in a reduced MNF with shortened strain modes on grids that are common to those elements (grids 104 through 108 and 204 through 208).

```
SOL 103
CEND
$
AdamsMNF FLEXBODY=YES,OUTGSTRN=YES,OUTGSTRS=NO
METHOD=300
RESVEC=NO
$
STRAIN(PLOT)=ALL
GPSTRAIN(PLOT)=ALL
OUTPUT(POST)
SET 100 = 4,5,6,7
SURFACE 101 SET 100 NORMAL X3 FIBRE=Z1
$
BEGIN BULK
ASET1,123,101,111,201,211
SPOINT,1001,thru,1003
QSET1,0,1001,thru,1003
DTI,UNITS,1,KG,N,M,S
PARAM,GRDPNT,0
$
EIGRL   300     -1.             3
$
GRID    101             0.      0.      0.
GRID    102             0.05    0.      0.
GRID    103             0.1     0.      0.
GRID    104             0.15    0.      0.
GRID    105             0.2     0.      0.
GRID    106             0.25    0.      0.
GRID    107             0.3     0.      0.
GRID    108             0.35    0.      0.
GRID    109             0.4     0.      0.
GRID    110             0.45    0.      0.
GRID    111             0.5     0.      0.
GRID    201             0.      0.03    0.
GRID    202             0.05    0.03    0.
GRID    203             0.1     0.03    0.
GRID    204             0.15    0.03    0.
GRID    205             0.2     0.03    0.
GRID    206             0.25    0.03    0.
GRID    207             0.3     0.03    0.
GRID    208             0.35    0.03    0.
GRID    209             0.4     0.03    0.
GRID    210             0.45    0.03    0.
GRID    211             0.5     0.03    0.
$
CQUAD4  1       1       101     102     202     201
CQUAD4  2       1       102     103     203     202
CQUAD4  3       1       103     104     204     203
CQUAD4  4       1       104     105     205     204
CQUAD4  5       1       105     106     206     205
CQUAD4  6       1       106     107     207     206
CQUAD4  7       1       107     108     208     207
CQUAD4  8       1       108     109     209     208
CQUAD4  9       1       109     110     210     209
```

```
CQUAD4   10       1        110      111      211      210
$
MAT1     1        2.+11             .3       7800.
PSHELL   1        1        .01      1
ENDDATA
```

Translating MSC Marc Data

You can obtain modal neutral files (MNFs) from a MSC Marc FE model using the MSC Marc - Adams Results Interface. For more information, see *MSC Marc Theory and User Information* guide.

Translating ANSYS Data

ANSYS writes modal neutral files (MNFs) directly, without creating intermediate data files, using the ANSYS-developed **Adams** macro included with ANSYS 5.3 and higher. As of ANSYS 7.1, there is no support for modal loads or preloads. However, ANSYS 6.0 and higher does support stress/strain modes for Adams Durability.

Learn more about how to use the macro for ANSYS 5.5 and higher. For earlier versions, contact ANSYS Technical Support.

- Generating MNFs from ANSYS
- Special Considerations for ANSYS 5.6 and 5.7
- Verifying the Model

For example input files, see the directory *install_dir*/flex/examples/ANSYS.

Generating MNFs from ANSYS

Whether you invoke ANSYS in interactive or batch mode, generating modal neutral files (MNFs) are straightforward. Beginning with a well-defined structural finite-element model, you:

- Specify model units
- Specify attachment points
- Run the ADAMS macro

Specifying Model Units

Because Adams can manipulate units, the MNF must contain the units you were implying during the ANSYS session. You identify the units using the /UNITS command in your ANSYS input file or in the input window during an interactive session. The command syntax is shown below where *LABEL* is either SI, CGS, BFT, BIN, or USER.

```
/UNITS, LABEL
```

Units for force, mass, length, and time corresponding to *LABEL* values are shown in the table below.

Values for LABEL

For the LABEL value:	Force is:	Mass is:	Length is:	Time is:
SI	Newton	Kilogram	Meter	Second
CGS	Centimeter	Gram	Dyne	Second
BFT	Pound-force	Slug	Foot	Second
BIN	Pound-force	Slinch	Inch	Second

To use a units system other than those listed above, use the USER label:

```
/UNITS, USER, L, M, T, ,,, F
```

where L, M, T, and F indicate unit conversion factors from the implied units to SI units. For example, to register mm, Mg, N, and S units, enter:

```
/UNITS, USER, 1000, 0.001, 1 ,,,, 1
```

Specifying Attachment Points

After defining the units, select the attachment points using the NSEL command. The NSEL command is described in detail in the ANSYS documentation. However, for convenience, we've provided a simplified overview of its syntax:

```
NSEL, S,,, NMIN, NMAX, NINC
NSEL, A,,, NMIN, NMAX, NINC
```

NMIN, NMAX, and NINC are the minimum, maximum, and increment in the selected node range.

NSEL,S creates a new attachment point list, while NSEL, A adds to the current attachment point list. For example, to promote nodes 1, 21, 211, and 231 to attachment points, add the following commands to your ANSYS input file or enter them in the input window during an interactive session:

```
NSEL, S,,, 1, 21, 20
NSEL, A,,, 211, 231, 20
```

It is important to note that these commands should be made just before the **ADAMS** macro command, because the **ADAMS** macro takes the current selected nodes as the attachment points. If keypoints have been defined in the input file or during the interactive session, either the KSEL or NSLK commands can be used to select attachment points. For details concerning keypoints and the KSEL and NSLK commands, see the ANSYS documentation.

Running the ADAMS Macro

After defining units and attachment points, you can generate an MNF file using the ADAMS macro. The syntax for the ADAMS macro is:

```
ADAMS, NMODES, KSTRESS, KSHELL (ANSYS 6.0 and higher)
ADAMS, NMODES (other versions)
```

where the required parameter NMODES is the number of normal modes to be compared. The Adams Flex Toolkit automatically orthogonalizes the normal and constraint modes, and writes the results to an MNF with the same name as your ANSYS input file or database file.

The last two arguments of the **ADAMS** macro are for modal stress recovery. *KSTRESS* is an integer flag that determines whether to include stress or strain modes. Its values are shown in the table below.

KSTRESS Values

The *KSTRESS value:*	Indicates:
0	Do not include stress/strain modes (default)
1	Stress modes included
2	Strain modes included
3	Stress and strain modes included

Additionally for shell models, use *KSHELL* to specify the location of the stress or strain results. Its values are shown in the table below.

KSHELL Values

The *KSHELL value:*	Indicates:
0,1	Shell top surface (default)
2	Shell middle surface
3	Shell bottom surface

For more information about modal stress recovery, see Adams Durability online help.

Special Considerations for ANSYS 5.6 and 5.7

If you create the model geometry by reading in a *jobname*.cdb file, an error in the **ADAMS** macro can occur if the *jobname*.cdb file contains a MODOPT command. The MODOPT command prevents the spectrum analysis from correctly assembling the mass matrix and generating a correct MNF. This problem has been fixed for ANSYS 6.0. To work around the problem for ANSYS 5.6 and 5.7, you must issue the following commands after reading in the database file:

```
/SOLU
ANTYPE, STATIC
ANTYPE, MODAL
```

For more details, contact ANSYS Technical Support.

Verifying the Model

The ANSYS translator writes the results of the modal neutral file (MNF) export to the terminal window. Please review this data for any obvious problems. In particular, ensure that the:

- Mass, center of mass location, and moments of inertia are as expected.

- During the MNF write, the constraint modes and the constrained normal modes are orthogonalized. This yields modes that are:
  - An approximation of the free body normal modes.
  - Interface modes, where the interface is the collection of all the attachment point DOFs.

Also, verify that the free body normal modes have a reasonable natural frequency. You should expect to see six rigid body modes, unless DOFs were fixed with zero displacement boundary conditions.

Translating ABAQUS Data

You can obtain a modal neutral file (MNF) from an ABAQUS FE model using the ABAQUS/Adams Interface that ABAQUS, Inc. distributes.

For more information, refer to the ABAQUS documentation provided with the interface.

Translating I-DEAS Data

In general, you build flexible bodies in Adams Flex by importing an Adams modal neutral file (MNF) representation of the flexible component. You can obtain an MNF of a flexible component in I-DEAS by performing a Superelement Creation analysis of the component's finite element (FE) model.

The steps are listed below. Click a link in a step to learn more.

- Define connection degrees of freedom (DOFs) in the Boundary Conditions task.
- Mesh the model in the Meshing task. Note: You can choose to mesh the model first, particularly if you want to create DOFs on nodes.
- Solve the model in Model Solution.
- Optionally, perform a series of tests to check the validity of your model.

For more information also see the I-DEAS guide, Finite Element Modeling User's Guide.

Creating a Connection DOF Set

To create a connection degree-of-freedom (DOF) set:

1. Create a finite element (FE) model of the flexible component. Make sure the task is the Boundary Conditions task.
2. Set the analysis type to **Superelement Creation**.
3. Define DOFs at each joint or applied force marker location of the flexible component. You can define DOFs on geometry, or, if the model is already meshed, on nodes.
4. In general, specify all six (translational and rotational) DOFs at interface locations with beam or shell elements, but only three (translational) DOFs for solid-element meshes.
5. Create a boundary conditions set using the **Boundary Conditions Management** form. Select **Connection Dof Set** to use the DOF set you've created to define connection DOFs.

At this stage, you can also specify additional sets, such as restraint, constraint, or temperature sets. Use constraint sets to introduce constraint equations in your FE model. Use temperature sets to specify the temperature of the analysis for temperature-dependent materials. In general, a restraint set isn't recommended for superelement creation because it permanently grounds the component at each specified restraint location, possibly providing insufficient flexibility information for your component.

Meshing the Model

Mesh the model in the Meshing task. After you generate a mesh of your FE model, it's a good idea to obtain a list of the solid properties of your mesh. While this isn't necessary for modal neutral file (MNF) transfer, you can use this data to check the results of the MNF transfer. See Checking the Results.

Solving the Model

To solve the model:

1. In the Model Solution task, create a Superelement Creation solution set using the boundary conditions set you created earlier.
2. On the **Create Solution Set** form, select **Options**, and then select **Solution Control**. Specify the number of normal modes and the optional frequency range that you want to capture for your component in the modal neutral file (MNF) for Adams simulation.
3. Request stress modes in the MNF by selecting **Output Selection...** in the Solution Set form, and then specifying **Stress... Store** in the Output Selection form.

   For shell element models, specify in which shell surface to output stresses via the MDI_MNFWRITE_OPTIONS environment variable. For example, one of the following keywords needs to appear somewhere in the MDI_MNFWRITE_OPTIONS environment variable depending on the desired surface:

   ```
   Surface KEYWORD options
   Top TOP / top
   Middle MIDDLE / middle
   Bottom BOTTOM / bottom
   Membrane MEMBRANE / membrane
   Bending BENDING / bending
   ```
4. Select **Manage Solve**. Specify the name of the MNF to be created, with the .mnf extension.
5. Select **Solve**. If errors or warnings occur, select **Report Errors/Warnings**. Resolve any significant errors before you proceed.

Checking the Results

The following steps are not necessary for superelement creation but they provide simple checks and measures of a successful modal neutral file (MNF) transfer.

After you solve, compare the total mass and inertia properties of the MNF listed either in the analysis log file or list region with those obtained when you meshed the model. Check that they are in close agreement. Note

that the values listed for the MNF are given in SI units, and you may need to convert them to model units for comparison.

Switch to the Post Processing task to perform the following checks:

- Select **Select Results**. On the **Results Selection** form, check that the number of normal modes solved equals the number of normal modes requested on the *Solution Control* form plus the number of degrees of freedom specified in the Connection DOF set.
- Animate some of these modes to ensure that there are no problems (such as breaks or cracks) in your FEA. The first few modes (up to six) should be rigid body modes (with a frequency close to zero), and the next few modes should closely resemble the natural free vibration modes of the component. The higher frequency modes should represent local flexible effects of the component at the connection DOF.

Translating Modal Test Data

Introduction

It is not necessary to perform a finite element analysis of your component to generate an Adams Flex body. It is also possible to generate a flexible body in Adams from modal test data stored in a Universal file format. You can translate the Universal file into a Modal Neutral File (MNF) that you can use to create a flexible body with a constant coupling inertia invariant formulation. For more information on invariant formulations, see Modifying the Modal Formulation.

Learn more about:

- Universal File Format Requirements
- Translating Universal File to MNF

Universal File Requirements

To create a modal neutral file (MNF) from modal test data, you must provide a single Universal file that contains:

- Nodal positions stored in Universal file dataset 15
- Element connectivity, or trace lines, stored in Universal file dataset 82
- Mode shapes and natural frequencies stored in Universal file block 55

Note:	Only the real, translational components of the mode shapes will be considered by Adams Flex. Consequently, the component will be infinitely stiff with respect to rotational deformations.

Adams View supports Universal files exported from I-DEAS, CADAX, or STAR, which meet these requirements.

Units

The Universal file can be in any set of units that Adams View supports. Learn about Units of Measurement in Adams View.

Other Required Data

In addition to providing a Universal File containing the modal response of the component, you will also be required to provide the total mass and inertia of the component as well as the center of mass location.

Limitations

Because it is difficult to capture a component's distribution of mass using experiment methods, the MNF will only contain inertia invariants describing the global inertia properties, namely invariants 1, 2, and 7. These

three inertia invariants are all that is required for the constant coupling inertia invariant formulation. Consequently, an Adams Flex flexible body created from modal test data should exclusively use the constant coupling inertia invariant formulation.

Translating Universal File to MNF

To translate a Universal file using Adams View:

1. From the Ribbon menu, click the **Bodies** tab. From the **Flexible Bodies** container, click the **Adams Flex** tool  .

   or

   (Classic interface) From the **Build** menu, point to **Flexible Bodies**, and then select **Adams Flex**.

   The Create a Flexible Body dialog box appears.
2. In the **Universal File** text box, enter the name of the Universal file containing datasets 15, 55, 82 and, in the option menu to the right, select the program from which it was generated (**CADA-X**, **I-DEAS**, **STAR**).
3. In the **MNF File** text box, enter the name of the MNF to be created. If you do not provide a file name, Adams Flex generates an MNF with the same path and prefix as the Universal file.
4. In the **Total Mass** text box, enter the global mass of the tested component.
5. In the **Center of Mass** text boxes, enter the x, y, and z coordinates of the tested component's center of mass, relative to datum used to measure the nodal positions in dataset 15 of the Universal file.
6. In the **Inertia Tensor** text boxes, enter the inertia tensor of the tested component relative to the center of mass.
7. If desired, in the **Title** and **Comments** text boxes, enter a title for the MNF and any comments you want to identify it.
8. Set **Length Units,** Mass Units, Time Units, and **Force Units to** the units used in the Universal file and the data entered in this dialog box.
9. Select **OK**.
10. After successfully translating the Universal file to an MNF, return to the Create a Flexible Body dialog box to create your flexible body in Adams View.

Creating an MD DB

Overview

Create an MD DB using MSC Nastran

An MD DB can be created in the same way as creating an MNF. A new option, EXPORT, is added to ADAMSMNF card to specify the output option.

```
ADAMSMNF FLEXBODY=YES EXPORT=MNF/DB/BOTH
```

MNF: generate modal neutral file

DB: generate MD DB

BOTH: generate both MNF and DB

Please refer to MSC Nastran Quick Reference Guide and Reference Manual for more details.

Use Flex Toolkit to convert MNF to MD DB

You can also use the mnf2mtx utility to convert MNF to MD DB. The usage is:

```
adams flextk  mnf2mtx source.mnf -O dest.MASTER
```

where source.mnf is the mnf you want to convert and dest.MASTER is the Database name. If dest.MASTER exists, mnf2mtx will append the flexible body in source.mnf to dest.MASTER. So you can combine MNFs into one MD DB using mnf2mtx. For example,

```
adams flextk  mnf2mtx source1.mnf -O dest.MASTER
```

then

```
adams flextk  mnf2mtx source2.mnf -O dest.MASTER
```

will append the flexible body in source2.mnf to dest.MASTER with INDEX=2.

Note:	With version 2015 or before, Adams MD DB import only supported Nastran i4 database files. In version 2015.1 or later Adams MD DB supports only i8 files. Support for i8 begun with MSC Nastran version 2016.

Using the Adams Flex Toolkit

Working with the Adams Flex Toolkit GUI

Learn how to work with the Adams Flex Toolkit. For information on accessing the Adams Toolbar, see Running and Configuring Adams.

Accessing the Adams Flex Toolkit GUI

To access the Adams Flex Toolkit GUI from the Adams Toolbar in Linux:

- On the Adams Toolbar, click the **Adams Flex Toolkit** tool  .

To access the Adams Flex toolkit GUI from the Start menu in Windows:

- From the **Start** menu, point to **Programs**, point to **Adams 2021.2**, and then select **Adams Flex**.

In Windows, to view all the information that appears in a command window, you need to enable scroll bars.

To enable scroll bars:

1. From the **Start** menu, point to **Programs**, and then select **Command Prompt**.
2. Hold down the right mouse button on the title bar, and select **Properties** from the shortcut menu.
3. On the **Layout** tab, change the Screen Buffer Size **height** from 25 to **2500**.
4. Select **OK**, and then select to make the changes for the shortcut.

Browsing an MNF or an MD DB

Using the Adams Flex Toolkit, you can browse individual aspects of a Modal Neutral File (MNF) or an MD DB, such as the file version, units, eigenvalues, or modal shapes. You can also view the contents of the entire file. You can view the MNF or an MD DB on the screen in the MNF Browser or have Adams Flex create a series of files for later viewing. The files are stored in your working directory with a .rpt extension. The MNF toolkit uses the name of the MNF or the MD DB as the basis for the file name.

To browse an MNF or an MD DB:

1. Select the tab **MNF Browser**.
2. Right-click the **MNF or MD DB Input File** text box, and then select **Select A File**.

   Then a file browsing dialog box appears.
3. Select the MNF or MD DB of interest, and then close the dialog box.

4. If you selected MD DB and it has more than one flexible body in it, the **Index** textbox will be enabled. Right-click it and select a flexible body from the popup dialog. The index of it will be input in the **Index** textbox.
5. From the different areas of the toolkit, select the type of information that you want displayed or select **All** to display the entire contents of the selected flexible body.
6. Select one of the following:
   - To display the flexible body information in the MNF Browser, select  .

     The MNF Browser appears. View the information about the flexible body by selecting the different tabs along the top of the Browser.
   - To store the flexible body information in a file, select  . A dialog box appears listing the files generated. Select **Close**.

Translating an MNF or an MD DB into a Matrix File

You use the Adams Flex Toolkit to translate a Modal Neutral File (MNF) or an MD DB file into an Adams Solver matrix file. Learn about Creating Matrix Files.

To translate an MNF into a matrix file:

1. Select the tab **MNF -> MTX Translator**.

   Options appear in the Adams Flex Toolkit for translating an MNF file to a matrix file.
2. Right-click the **MNF or MD DB Input File** text box, and then select **Select A File**.

   Then a file browsing dialog box appears.
3. Select the MNF of MD DB interest, and then close the dialog box.
4. If you selected MD DB and it has more than one flexible body in it, the **Index** textbox will be enabled. Right-click it and select a flexible body from the popup dialog. The index of it will be input in the **Index** textbox.
5. In the **Matrix Output File** text box, enter the name for the matrix file that you are creating.
6. From the pull-down menus at the left, select the units that match the units in your Adams Solver dataset that references the matrix file you are creating. The units do not have to match those in the MNF or MD DB.
7. In the **Node ID List** and **Mode Number List** text boxes, enter a list of nodes or modes that you want to include or exclude in the matrix file. Note that the list of modes to include can contain a colon (for example, 7 10 : 16) to indicate a range. Select **Include** or **Exclude** as appropriate.
8. If you are going to have Adams Flex compute all nine invariants, clear the selection of **Fast Invar**. If you leave **Fast Invar** selected, Adams Flex does not compute invariants five and nine, which provides you with a faster translation.

   Selecting **Fast Invar** is appropriate for the default Partial Coupling formulation for flexible bodies. You can also use it with the Constant Coupling formulation or any custom combination of invariants that does not require invariants 5 and 9. Learn about Modifying the Modal Formulation.

9. Select  .

   A command window appears as Adams Flex calculates the inertia invariants and writes the matrix file.

10. Press **Enter** to exit the command window.

Optimizing an MNF or an MD DB

You can optimize the flexible body in a Modal Neutral File (MNF) or an MD DB through the Adams Flex Toolkit, such as set the number of invariants stored in the flexible body or set the units in which the data is stored. The optimizations should help make the MNF smaller and more efficient. You can optimize a flexible body interactively or generate a script to run the optimization. Generating a script lets you run the optimization on another computer platform on which you have Adams installed.

The options for optimizing an MNF through the Adams Flex toolkit correspond to the parameters in the MDI_MNFWRITE_OPTIONS environment variable that controls the MNF toolkit, which you use to generate an MNF. For more information on the MDI_MNFWRITE_OPTIONS, see Setting Up Translation Options through the MNF Toolkit.

Note:	If you generate an MNF without all its invariants or geometry, it loses mass information, and you cannot generate a matrix file or MNF that contains a full set of inertia invariants from such an MNF.

To optimize an MNF:

1. Select **MNF -> MNF Optimizer**.

   Options appear in the Adams Flex Toolkit for optimizing an MNF.

2. Right-click the **MNF or MD DB Input File** text box, and then select **Select A File**.

   Then a file browsing dialog box appears.

3. Select the MNF of MD DB of interest, and then close the dialog box.
4. If you selected MD DB and it has more than one flexible body in it, the **Index** textbox will be enabled. Right-click it and select a flexible body from the popup dialog. The index of it will be input in the **Index** textbox.
5. In the **MNF or MD DB Output File** text box, enter the name for the file in which to store the optimized flexible body. Please be noted that if you specify an existing MD DB, the optimized flexible body will be appended to the Database instead of overwriting it..
6. Set the options in the dialog box as explained in the table below, and then select either  to perform the optimization immediately or  to generate a script that you can run later.

Options for Optimizing Flexible Body

To set:	Do the following:
Invariant calculations	From the **Invariants** pull-down menu, set which inertia invariants should be computed and stored in the MNF. You can select: ■ **Fast Set** - If you select **Fast Set**, Adams Flex does not compute invariants five and nine. It corresponds to the Partial Coupling formulation mode for modal flexibility. It is also suitable for use with the Constant Coupling formulation. Only Full Coupling requires all nine invariants. Unless you think you might need the Full Coupling formulation, you can safely select **Fast Set**. Learn more. ■ **Full Set** - If you select **Full Set**, Adams Flex computes all inertia invariants, including invariants five and nine. ■ **None** - If you select **None**, Adams Flex does not perform any invariant calculations, and must compute invariants each time you save an Adams Solver dataset with a modified selection of modes or nodes.
Units used	From the **Units** pull-down menu, do the following: ■ To preserve the units in the original Flexible Body, select **Original**. If you select to preserve the units, Adams Flex performs the unit scaling as it performs different operations, which can degrade performance noticeably. ■ To convert all data to Adams View internal units, which are meters, kilogram, seconds, and Newtons, select **SI**.
Portability and backward compatibility	From the **Formatting** pull-down menu, do either of the following: ■ To turn off the encoding that makes the MNF platform independent, select **Platform Specific**. The encoding has some computational overhead that you may want to remove if you are not concerned about MNF portability. ■ To keep the encoding and portability, select **Standard Portable**. Please be noted that MD DB will be platform-dependent no matter which option you specified here.

To set:	Do the following:
Precision	From the **Precision** pull-down menu, set the level of precision for real numbers in the MNF: ■ **Double** - All real data in the MNF will be stored in double-precision, typically 64-bit floating point values. ■ **Single** - All real data in the MNF will be stored in single-precision, typically 32-bit floating point values. A single-precision MNF is nearly half the size of a double-precision MNF. A single-precision MNF does not adversely effect the accuracy of your results provided that your optimized MNF contains either *Fast* or *Full* set of inertia invariants. Some accuracy can be lost if you request *None* for the inertia invariant calculations in a single-precision optimized MNF. See Invariant calculations.
Stress and strain modes	If stress (strain) recovery was requested from the finite element program when generating the MNF, the MNF contains grid point stresses (strains) for every mode. The collection of grid point stresses (strains) for a given mode is referred to as a stress (strain) mode. Typically, stress (strain) values are requested from the finite element program for a subset of nodes in the MNF. You can specify how the MNF stores stress (strain) modes, particularly for nodes where stress (strain) was not requested from the finite element program: ■ **Sparse** - If you select Sparse, the optimized MNF only stores stresses (strains) for nodes that were retained in the optimized MNF and for which stress (strain) values existed in the original MNF. If a node had zero values for stresses (strains) in the original MNF, and that node was retained in the optimized MNF, the zeroes are written to the optimized MNF. ■ **Full** - If you select Full, the optimized MNF stores nodal stresses and strains for all nodes that were retained in the optimized MNF. For nodes that did not have stress (strain) values, the optimized MNF stores zeroes. ■ **Remove zero entries** - If you select Remove zero entries, the MNF only stores non-zero stresses (strains) for nodes that exist in the optimized MNF. If you have an MNF that has several zero entries in the stress (strain) modes, this option can significantly reduce the size of the MNF.

To set:	Do the following:
Create a rigid-only MNF	Depending on the component and application, the size of the MNF can be very large, exceeding several gigabytes, and difficult to manage. If you are in the process of building your Adams Flex body model, you may consider treating the body as rigid until you are confident in how you have assembled your model. If temporarily using a rigid body formulation for an Adams Flex body makes sense, you can drastically reduce the size of the MNF by selecting **Rigid-Only MNF**. Rigid-Only MNF creates a reduced MNF that only contains enough information to build a rigidized flexible body. With this MNF, you cannot build an Adams Flex body with Constant, Partial, or Full modal formulations, but it may be convenient to work with while you are assembling and verifying your model. When you are confident with your model, you can easily replace the reduced MNF with the full MNF.
Removal of interior geometry, such as the mated faces of two brick elements, from solid finite element models to enhance graphics performance	Select the tab **Automatic,** and then select **Remove Internal Solid Element Geometry**. When you remove the interior geometry, the graphics performance of Adams View is greatly enhanced. When you remove both interior geometry and calculate the invariants, Adams Flex removes nodes that were only connected to the geometry that it also removed. Occasionally, the removal of the geometry may be undesirable especially when a particular interior node is to be the target for an attachment in Adams.

To set:	Do the following:
Allow the MNF Optimizer to remove nodes that have modal loads defined.	Select the tab **Automatic**, and then select **Optimize Modal Load (MFORCE) Nodes**. All modal load nodes are retained by default so that detailed MFORCE plots can be generated in View. This may result in exported MNF that are larger than expected size when modal loads are defined. If detailed MFORCE plotting is not necessary and smaller MNF is required, then you can eliminate set this option to also eliminate modal load nodes when optimizing the MNF.
Mesh coarsening and colinear point removal to remove excessive detail from an MNF.	Select the tab **Automatic**, and then select **Apply Mesh Coarsening Algorithm**. Set the following: ■ **Mesh Resolution** - Slide the **Mesh Resolution** slider to the fraction of the total component size below which Adams Flex removes the detail of the mesh. For example, if your component is approximately 1 m long, and you select 15% mesh resolution, the coarsening results in a mesh with 15 cm-wide mesh cells. Example of Mesh Resolution. ■ **Face Smoothing** - Slide the **Face Smoothing** slider to the angle between adjacent faces below which Adams Flex should merge faces. For example, if you select 15, the coarsening algorithm does not merge two faces when one face is more than 15o out of the plane of the other face. Example of Face Smoothing. ■ **Colinear Point Removal** - Select **Remove Colinear Points** to control removal of nodes that are intermediate nodes on the straight edge of a face. Example of Collinear Point Removal. ■ **Preserve Stress and Strain Modes** - Select Preserve Stress and Strain Modes to control retention of nodes that have stress or strain values. ■ **Retain Particular Nodes** - In the **Retained Node List** text box, specify a list of nodes that Adams Flex should **not** remove during coarsening. Only visible nodes can be used to place markers, joints, and so on, onto the flexible body. Therefore, make sure to include relevant nodes. When you use mesh coarsening and also calculate the invariants, Adams Flex removes nodes that were only connected to the geometry that was removed by coarsening, which results in a great reduction in MNF size.
Manually specify how to reduce the mesh.	Select the tab **Manual**, and then enter the name of the sketch file. For more information on the format of the sketch file and its impact on the mesh, see Manual Mesh Simplification.

Editing an MNF or an MD DB File

The Adams Flex Toolkit used to edit the node interface data from a Modal Neutral File (MNF) or an MD DB file and write updated data to an output MNF or MD DB file.

To edit an MNF and write to an output MNF:

1. Select the tab **MNF -> MNF Editor**.

   Options for editing the MNF appear in the Adams Flex toolkit.

2. Right-click the **MNF** or **MD DB Input File** text box, and then select "**Select A File**".

   The file browsing dialog box appears. Select the MNF or MD DB of interest, and then close the dialog box.

3. If MD DB is selected and it has more than one flexible body in it, the Index textbox will be enabled. Right-click it and select a flexible body from the popup dialog box. The index of it will be input in the Index textbox.

4. Enter **MNF** or **MD DB Output File** name or right-click **Output File** text box, and then select "**Select A File**". Select the MNF of MD DB interest, and then close the dialog box.

5. Click the **Get Interface Node Coordinates** button to read the MNF and populate the table with interface node data.

6. This data table has four columns:

   a. **Node ID**: Column lists the node ID's of all the interface/attachment nodes (a.k.a "ASET's") in the specified MNF / MD DB Index. This is a read-only column.

   b. **X Coordinate**: Column lists the X coordinates of each row's interface node, these cells are editable and could accept only real values. If it is edited, it will turn cyan.

   c. **Y Coordinate**: Column lists the Y coordinates of each row's interface node; these cells are editable and could accept only real values. If it is edited, it will turn cyan.

   d. **Z Coordinate**: Column lists the Z coordinates of each row's interface node; these cells are editable and could accept only real values. If it is edited, it will turn cyan.

7. To modify the modal scaling factors for generalized mass and generalized stiffness, enter positive real values in **Generalized Mass** and **Generalized Stiffness** text boxes.

8. Set **Precision** option in the dialog box according to the precision desired for the new MNF. The Precision option is explained in the table in Optimizing an MNF or an MD DB.

9. Once all editing is done, click the **File Processing button**  to generate a new MNF based on editing.

Working with the Adams Flex Toolkit Command Line

Adams Flex Toolkit Command Line

In additional to the GUI, the Adams Flex Toolkit also provides three extra command line tools, which are:

- **MNFXFORM** - Translating, rotating or mirroring an MNF (MD DB)
- **MNFRES** - Recovering nodal displacement, velocity or acceleration of a flexible body
- **ABQ2NAS** - Converts Abaqus input files to MSC Nastran input files
- **MNF2MTX** - Edit interface node coordinates or generalized mass/stiffness of MNF

To run the Adams Flex toolkit command line from the program menu, enter **adams2021_2** -c flextk on Linux systems or **adams2021_2** flextk on Windows systems.

Transforming an MNF or an MD DB

In MNF (MD DB), the data are defined with respect to the FE origin. MNFXFORM is a tool to translate, rotate or mirror the MNF (MD DB) with respect to the FE origin. Please be noted that this is different from specifying position and orientation of a flexible body in AView, which only changes how the FE origin is positioned and oriented and could not change how the flexible body is positioned and oriented with respect to the FE origin.

Without this tool, the only way user can change how a flexible body is positioned or oriented with respect to its FE origin is to go back to FE preprocessor to transform the FEA model and generate a new MNF (MD DB).

Some of the benefits this tool provides include:

1. In high speed rotation simulation, the simulation speed can be significantly improved if the rotation axis is aligned with the Z axis of the FE origin. With MNFXFROM, user can easily rotate the flexible body to this configuration.
2. It involves some extra efforts to generate a mirrored copy of an MNF (MD DB) from FEA code. With this tool, this task becomes trivial. Adams Car users whose models often involve symmetrical parts will find this tool is very handy.

MNFXFORM Usage

Following is the usage of the mnfxform command under flextk:

mnfxform <option> <input_flex_file> <output_flex_file> <parameters>

[**-offset inc**] [**-id nid n1 n2 n3 ...**]

Following are explanation of the arguments.

Argument:	Description:
<option>	Specify which transformation to perform. The option should be one of the following: **-t** for Translation. This transformation needs to input direction and distance in <parameters> **-r** for Rotation. This transformation needs to input axis and angle in <parameters> **-m** for Mirroring. This transformation needs to input plane in <parameters>
<input_flex_file>	MNF or MD DB File. MD DB File is in the form of *.MASTER[::#], *.MASTER is the database and # is the index of the body. For example, foo.MASTER::2 indicates the second flexible body in foo.MASTER. foo.MASTER, without ::#, indicates the first flexible body.
<out_flex_file>	Output MNF or MD DB File for given options. MD DB File is in the form of *.MASTER. If *.MASTER already exists in the output directory, the transformed flexible body will be appended to it.
<parameters>	Input the parameters needed by specified transformation: **-p px py pz** Specify a point P **-r rx ry rz** Specify a point R **-s sx sy sz** Specify a point S **-v vx vy vz** Specify a Vector V **-d dist** Specify Distance dist **-a angle** Specify Angle (Anti-clockwise in degrees) See Notes for details.
[**-offset inc**]	Optional argument to offset ALL the node IDs in the MNF (MD DB) by **inc**. New node IDs will be old IDs plus **inc**. If non-positive IDs are resulted by this argument, the MNFXFORM process fails and reports an error.
[**-id nid n1 n2 ...**]	Optional argument to specify new interface node IDs. **nid** is the number of new IDs will be specified, **n1 n2 ...** are the new IDs.

Note:

1. How to specify direction and distance. In the following table **-v** means **-v vx vy vz**, and **-d** means **-d dist** and so on.

Parameters	Explanation
-v -d	Direction is V, Distance is specified by -d
-p -r -s -d	Direction is the normal of plane PRS, Distance is specified by -d
-p -r -d	Direction is vector PR, Distance is specified by -d
-p -r	Direction is vector PR, Distance is length of segment PR

2. How to specify axis and angle.

Parameters	Explanation
-v -p -a	Axis is in direction V and passes point P, Angle is specified by -a
-p -r -s -a	Axis is the normal direction of plane PRS and passes point P, Angle is specified by -a
-p -r -a	Axis is in the direction of vector PR and passes point P, Angle is specified by -a

3. How to specify plane .

Parameters	Explanation
-v -p	Plane passes point P and its normal direction is V
-p -r -s	Plane PRS
-p -r	Plane passes point P and its normal direction is vector PR

4. If both **-offset** and **-id** are specified, **-offset** will take precedent. That is, all the node IDs are first renumbered using the **-offset** argument, then the interface node IDs are renumbered using **-id** argument.

Examples:

1. **mnfxform.exe -m input.mnf output.mnf -v 1 0 0 -p 0 0 0 -offset 1**

   This example mirrors input.mnf about yz plane and increase the ids of the interface nodes by 1. Then the transformed flexible body is saved as output.mnf.

2. **mnfxform.exe -r input.MASTER::3 output.MASTER -p 1 0 0 -r 0 1 0 -s 0 0 1 -a 30 -id 5 10 11 12 13 14**

This example rotates the 3rd flexible body in input.MASTER normal direction of plane PRS (defined by three points) by 30 degree (anti-clockwise). Then the first five interface node ids are renumbered to (10, 11, 12, 13, 14). Finally the transformed flexible body is saved to MD DB output.MASTER. If output.MASTER exists, the transformed body will be appended to it.

Limitation:

- For translation, rotation about an axis not passing origin or mirroring about a plane not passing origin, nodal mass and mode shape data is required to compute Invar4.

Converting Abaqus Input Files

To convert Abaqus input files into MSC Nastran input files use the command Abq2Nas. This is implemented within the Adams Flex toolkit command line. The exported MSC Nastran input files are written as SOL400 models. These can be used, for example, as input BDFs to Adams Flex Nonlinear.

Abq2Nas Usage

Following is the usage of the Abq2Nas command under flextk:

```
abq2nas <input_abaqus_file> <output_nastran_file> [direct_text_inputs]
```

Following are explanation of the arguments.

Argument:	Description:
<**input_abaqus_file**>	Abaqus input file name.
<**output_nastran_file**>	MSC Nastran BDF file name. The translation processing information including warnings and errors will be written to a log file named as the prefix of BDF with ".abq2nas" extension.
[**direct_text_inputs**]	Optional arguments for setting text inputs directly to step, case control and bulk data sections. Below is are examples of the options: abq2nas aa.inp bb.bdf CASE="NLOPRM NLDBG=NRDBG" abq2nas aa.inp bb.bdf STEP="AUTOSPC(RESIDUAL,PUNCH)=YES" abq2nas aa.inp bb.bdf BULK="NLMOPTS,LRGS,1;,,SPROPMAP,2" Multiple lines can be expressed by using semicolons ";", it is user's responsibility to provide a correct MSC Nastran entry, the translator just takes it as is. Another option mergecontact=yes/no controls contact merging, which will be explained later. abq2nas aa.inp bb.bdf mergecontact=yes

Supported entities and the map

The following shows the supported entities and how they map from Abaqus 2014 to MSC Nastran.

Notice here that "activate hyperelements formulation" is achieved by adding the card shown below into the MSC Nastran file:

```
NLMOPTS LRGSTRN 2
```

Abaqus	MSC Nastran
Plane Strain	
CPE3	CTRIA3, PLPANE, PSHLN2
CPE3H	CTRIA3, PLPANE, PSHLN2
CPE4	CQUAD4, PLPLANE + PSHLN2
CPE4H	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE4I	CQUAD4, PLPLANE + PSHLN2
CPE4IH	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE4R	CQUAD4, PLPLANE + PSHLN2
CPE4RH	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE6	CTRIA6, PLPLANE + PSHLN2
CPE6H	CTRIA6, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE6M	CTRIA6, PLPLANE + PSHLN2
CPE6MH	CTRIA6, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE8	CQUAD8, PLPLANE + PSHLN2
CPE8H	CQUAD8, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPE8R	CQUAD8, PLPLANE + PSHLN2
CPE8RH	CQUAD8, PLPLANE + PSHLN2, activate hyperelasticity formulation
Plane Stress	
CPS3	CTRIA3, PLPLANE + PSHLN2
CPS4	CQUAD4, PLPLANE + PSHLN2
CPS4I	CTRIA3, PLPLANE + PSHLN2
CPS4R	CTRIA3, PLPLANE + PSHLN2
CPS6	CTRIA6, PLPLANE + PSHLN2
CPS6M	CTRIA6, PLPLANE + PSHLN2
CPS8	CQUAD8, PLPLANE + PSHLN2
CPS8R	CQUAD8, PLPLANE + PSHLN2
Generalized Plane Strain	

Abaqus	MSC Nastran
CPEG3	CTRIA3, PLPLANE + PSHLN2
CPEG3H	CTRIA3, PLPLANE + PSHLN2
CPEG4	CQUAD4, PLPLANE + PSHLN2
CPEG4H	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG4I	CQUAD4, PLPLANE + PSHLN2
CPEG4IH	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG4R	CQUAD4, PLPLANE + PSHLN2
CPEG4RH	CQUAD4, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG6	CTRIA6, PLPLANE + PSHLN2
CPEG6H	CTRIA6, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG6M	CTRIA6, PLPLANE + PSHLN2
CPEG6MH	CTRIA6, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG8	CQUAD, PLPLANE + PSHLN2
CPEG8H	CQUAD, PLPLANE + PSHLN2, activate hyperelasticity formulation
CPEG8R	CQUAD, PLPLANE + PSHLN2
CPEG8RH	CQUAD, PLPLANE + PSHLN2, activate hyperelasticity formulation
Axisymmetric, Stress/displacement elements without twist	
CAX3	CTRIAX, PLPLANE + PSHLN2
CAX3H	CTRIAX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX4	CQUADX, PLPLANE + PSHLN2
CAX4H	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX4I	CQUADX, PLPLANE + PSHLN2
CAX4IH	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX4R	CTRIAX, PLPLANE + PSHLN2
CAX4RH	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX6	CTRIAX, PLPLANE + PSHLN2
CAX6H	CTRIAX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX6M	CTRIAX, PLPLANE + PSHLN2
CAX8	CQUADX, PLPLANE + PSHLN2
CAX8H	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CAX8R	CQUADX, PLPLANE + PSHLN2
CAX8RH	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
Axisymmetric, Stress/displacement elements with twist	

Abaqus	MSC Nastran
CGAX3	CTRIAX, PLPLANE + PSHLN2
CGAX3H	CTRIAX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX4	CQUADX, PLPLANE + PSHLN2
CGAX4H	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX4R	CQUADX, PLPLANE + PSHLN2
CGAX4RH	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX6	CTRIAX, PLPLANE + PSHLN2
CGAX6H	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX6M	CTRIAX, PLPLANE + PSHLN2
CGAX6MH	CTRIAX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX8	CTRIAX, PLPLANE + PSHLN2
CGAX8H	CQUADX, PLPLANE + PSHLN2
CGAX8R	CQUADX, PLPLANE + PSHLN2, activate hyperelasticity formulation
CGAX8RH	CQUADX, PLPLANE + PSHLN2
3D Stress/displacement elements	
C3D4	CTETRA, PSOLID + PSLDN1
C3D4H	CTETRA, PSOLID + PSLDN1
C3D6	CPENTA, PSOLID + PSLDN1
C3D6H	CPENTA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D8	CHEXA, PSOLID + PSLDN1
C3D8H	CHEXA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D8I	CHEXA, PSOLID + PSLDN1
C3D8IH	CHEXA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D8R	CHEXA, PSOLID + PSLDN1
C3D8RH	CHEXA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D10	CTETRA, PSOLID + PSLDN1
C3D10H	CTETRA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D10I	CTETRA, PSOLID + PSLDN1
C3D10M	CTETRA, PSOLID + PSLDN1
C3D10MH	CTETRA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D15	CPENTA, PSOLID + PSLDN1
C3D15H	CPENTA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D20	CHEXA, PSOLID + PSLDN1

Abaqus	MSC Nastran
C3D20H	CHEXA, PSOLID + PSLDN1, activate hyperelasticity formulation
C3D20R	CHEXA, PSOLID + PSLDN1
C3D20RH	CHEXA, PSOLID + PSLDN1, activate hyperelasticity formulation
Membrane	
M3D3	CTRIA3, PSHELL
M3D4	CQUAD4, PSHELL
M3D4R	CQUAD4, PSHELL
M3D6	CTRIA6, PSHELL
M3D8	CQUAD8, PSHELL
M3D8R	CQUAD8, PSHELL
M3D9	CQUAD8, PSHELL
M3D9R	CQUAD8, PSHELL
2D truss	
T2D2	CROD, PROD
T2D2H	CROD, PROD/PRODN1
T2D3	CROD, PROD
T2D3H	CROD, PROD/PRODN1
3D truss	
T3D2	CROD, PROD
T3D2H	CROD, PROD
T3D3	CROD, PROD
T3D3H	CROD, PROD
Beam in plane	
B21	CBEAM, PBEAM/PBEAML
B21H	NA, but write out the same one as B21 with a warning
B22	CBEAM3, PBEAM/PBEAML
B22H	NA, but write out the same one as B22 with a warning
B23	NA, but write out the same one as B21 with a warning
B23H	NA, but write out the same one as B21 with a warning
Beam in 3D space	
B31	CBEAM, PBEAM/PBEAML
B31H	NA, but write out the same one as B31 with a warning
B32	CBEAM3, PBEAM/PBEAML

Abaqus	MSC Nastran
B32H	NA, but write out the same one as B32 with a warning
B33	NA, but write out the same one as B31 with a warning
B33H	NA, but write out the same one as B31 with a warning
Conventional shells	
STRI3	CTRIA3, PSHELL
S3	CTRIA3, PSHELL
S3R	CTRIA3, PSHELL + PSHLN1
S3RS	CTRIA3, PSHELL + PSHLN1
STRI65	CTRIA3, PSHELL
S4	CQUAD4, PSHELL
S4R	CQUAD4, PSHELL + PSHLN1
S4RS	CQUAD4/PSHELL + PSHLN1
S4RSW	CQUAD4/PSHELL + PSHLN1
S4R5	CQUAD4/PSHELL + PSHLN1
S8R	CQUAD8/PSHELL + PSHLN1
S8R5	CQUAD8/PSHELL + PSHLN1
Continuum shell(Solid shell)	
SC8R	CHEXA, PCOMPLS
Spring	
SPRINGA	CBUSH, PBUSH + PBUSHT
SPRING1	CELAS1, PELAS + PELAST
SPRING2	CBUSH, PBUSH + PBUSHT
Dashpot	
DASHPOT1	CBUSH, PBUSH + PBUSHT
DASHPOT2	CBUSH, PBUSH + PBUSHT
DASHPOTG	CBUSH, PBUSH + PBUSHT
Flexible joint	
JOINTC	CBUSH, PBUSH + PBUSHT
Distributing coupling	
DCOUP2D	RBE2/RBE3
DCOUP3D	RBE2/RBE3
Gasket	
GK3D8	CHEXA, PSOLID, MATG

Abaqus	MSC Nastran
GK3D8N	CHEXA, PSOLID, MATG
Other geometry entities	
*NGEN	GRID
*NFILL	GRID
*ELGEN	Elements
*ASSEMBLY/*INSTANCE/*PART	offset ids of GRID and elements, *ELSET or *NSET with option INSTANCE=part_name is available also
*SYSTEM	CORD2R and write the cord2r id to CP field of related GRID
*ORENTATION	CORD2R and write the cord2r id to CBUSH, COMN1, 2D elements, shell elements and PSOLID
*TRANSFORM	CORD2R and write the cord2r id to CD field of related GRID
Properties	
*SOLID SECTION	PSOLID
*SHELL SECTION	PSHELL/PCOMP/PCOMPL/PCOMPLS
*BEAM SECTION	PBEAM/PBEAML
*BEAM GENERAL SECTION	PBEAML
*GASKET SECTION	MATG
Materials	
*ELASTIC	MAT1/MAT8/MATORT/MAT9
*DENSITY	Å@
*PLASTIC	MATEP
*HYPERELASTIC	MATHE
*VISCOELASTIC	MATVE
*CREEP	MATVP
Contact	
*SURFACE	BCBODY1,BSURF, BCNURB2, BCPATCH depending on *RIGID BODY
*RIGID BODY	BCNURB2 for analytical curves, a BCPACH for 3D model when the elements used by a *SURFACE
*CONTACT PAIR	BCONNECT
*SURFACE INERACTION	BCONPRP
*TIE	Glue contact (BCONNECT)
Loads Boundaries	

Abaqus	MSC Nastran
*CLOAD	FORCE/MOMENT
*TEMPERATUE	TEMP
*INTIAL CONDITION TYPE=TEMPERATUE	TEMP and TEMPERATURE(INIT) in case control
*DLOAD	PLOAD4/PLOAD1 GRAV RFORCE PLOADX
*DSLOAD	PLOAD4 for shell and solid, PLOAD1for beam elements PLOADX
MPC, Coupling and so on.	
*Kinematic Coupling	RBE2
*Distributing Coupling	RBE3
*Coupling + *Kinematic	RBE2
*Coupling + *Distributing	RBE3
*MPC Beam type	RBAR
*Pre-tension	BOLT
*Equation	MPC
Analysis procedure	
*STATIC, Perturbation	ANALYSIS=STATIC
*STATIC, no perturbation	ANALYSIS=NLSTATIC
*Frequency	ANALYSIS=MODES

In contact area, there is a strict limitation in MSC Nastran: one element can only be used by one contact body; the translator is highly possibly to make an element belong to multiple bodies. To overcome this issue, merging contact bodies that share common elements into one body is required. Command line option mergecontact=yes/no will control this, the default is yes, no will be provided for debugging purpose.

Important known limitations

The following input syntax is known to be not supported. This is not necessarily a complete list:

*AMPLITUDE

*BULK VISCOSITY

*CONTACT

*CONTACT INCLUSION

*CONTACT PAIR is not completely supported

*CONTACT PROPERTY ASSIGNMENT

*CYCLIC HARDENING

*DISTRIBUTION

*DLOAD - several parameters including PY are not supported

*DRUCKER PRAGER

*DRUCKER PRAGER HARDENING

*DSLOAD – several parameters including HP are not supported

*DYNAMIC

*FRAME SECTION, also B33 element is not supported

*HYPERELASTIC – parameter MODULI AND POLYNOMIAL

*INITIAL CONDITIONS type HARDENING

*INITIAL CONDITIONS type STRESS

*MOHR COULOMB

*MOHR COULOMB HARDENING

*MPC – type C BIQUAD and QUADRATIC are not supported

*NCOPY

*PLASTIC, DEPENDENCIES

*RBE - only 1D analytical surfaces and 2D R3D4, R3D3 elements can be translated

*REBAR LAYER

*STEADY STATE DYNAMICS

*STEP – parameter UNSYMM

*SUBMODEL

*SURFACE BEHAVIOR

*SURFACE is not completely supported

*SWELLING

*VISCO

BEH4 and BEH8 are not supported

Generalized plane strain elements are translated into equivalent plane strain elements

Recovering Flexible Body Deformation

MNFRES is a tool to recover the nodal displacement, velocity and acceleration of a flexible body after a simulation.

MNFRES Usage

Following is the usage of the mnfres command under flextk:

mnfres [options] <Adams_result_file> <input_flex_file>

Argument:		Description:
[**option**]	`-t <time> or`	output results only till specified time
	`<tms,tme>`	output results at the times <tms> to <tme>
	`-n <name>`	specify flexible body when multiple exist
	`-g`	include rigid body motion
	`-r`	also report nodal rotations. Only effective when -g is not specified
	`-s <file>`	report only on nodes listed in <file>
	`-L <unit>`	specify length unit used in the Adams model, e.g.,METER,INCH, FOOT, MILE, MILLIMETER,CENTIMETER, KILOMETER. Abbreviation is accepted. Default value is METER.
	`-T <unit>`	specify time unit used in the Adams model, e.g.,SECOND, MILLISECOND, DAY, HOUR, MINUTE. Default value is SECOND. Abbreviation is accepted.
	`-i <key>`	report specific results. <key> values are: d : Nodal displacements v : Nodal velocities a : Nodal accelerations
<Adams_result_file>		Adams result file.
<input_flex_file>		Modal Neutral File or MD DB File. MD DB File is in the form of foo.MASTER[::#], foo.MASTER is the database and # is the index of the body. For example, foo.MASTER::2 indicates the second flexible body in foo.MASTER. foo.MASTER, without ::#, indicates the first flexible body.

Notes:	1. If -g option is not specified, the output value is in flexbody PBCS. If specified, the output is with respect to ground. 2. By default the output result use METER and SECOND. So if the length unit used in the Adams model is not METER, you must specify -L <unit> option in order to get correct results. Same is true for time unit. 3. By default, the output will be printed to screen. User can use > filename to redirect the output to a file.

Examples:

1. **mnfres -i d -n FLEX_BODY_1 example.res foo.mnf**

   Output all the nodal deformation of flexible body FLEX_BODY_1, whose mnf file is foo.mnf and the result file is example.res.

2. **mnfres -i d -n FLEX_BODY_1 example.res foo.mnf > out.dat**

   Redirect the output in example 1 to out.dat.

3. **mnfres -i v -L MM -g -n FLEX_BODY_3 example.res foo.MASTER::2**

   Output all the nodal velocity of flexible body FLEX_BODY_3, which is the second flexible body stored in MD DB file, foo.MASTER, using result file example.res. The velocity includes rigid motion and the length unit is MILLIMETER.

Editing an MNF or an MD DB

MNF2MTX allows you to edit the interface (ASET) node coordinates or generalized mass/stiffness of an existent MNF or MD DB.

MNF2MTX Usage

Following is the usage of the mnf2mtx command under flextk:

```
mnf2mtx <input_flex_file> -O <output_flex_file> [-I
id1,x1,y1,z1,id2,x2,y2,z2,...,idn,xn,yn,zn]
 [-mscale scale_factor] [-sscale scale_factor]
```

Argument:	Description:
<input_flex_file>	Input Modal Neutral File or MD DB File. MD DB File is in the form of foo.MASTER[::#], foo.MASTER is the database and # is the index of the body. For example, foo.MASTER::2 indicates the second flexible body in foo.MASTER. foo.MASTER, without ::#, indicates the first flexible body.
<output_flex_file>	Output Modal Neutral File or MD DB File. MD DB File is in the form of foo.MASTER[::#], foo.MASTER is the database and # is the index of the body. For example, foo.MASTER::2 indicates the second flexible body in foo.MASTER. foo.MASTER, without ::#, indicates the first flexible body.
[**-I id1,x1,y1,z1,...**]	Optional argument to specify interface node coordinates to be edited. id* are the interface node IDs, and x*, y* and z* are the modified coordinates. IDs must be existed in the flex file and 4 parameters must be specified per 1 interface node. This option affects the interface node coordinate only.
[**-mscale scale_factor**]	Optional argument to specify scale factor of generalized mass. The scale factor affects the generalized mass and all eigenvalues are divided by the factor. In addition, nodal mass/inertia are multiplied by the factor. Note that scaled modal masses are normalized (reverted to 1.0) during flex file exportation and then eigenvectors are divided by square root of the factor. As a result, mass invariant matrices are scaled like as follows. INVAR1, INVAR2, INVAR7: Multiplied by the factor INVAR3, INVAR4, INVAR8: Multiplied by square root of the factor
[**-sscale scale_factor**]	Optional argument to specify scale factor of generalized stiffness. The scale factor affects the generalized stiffness and all eigenvalues are multiplied by the factor. In addition, mode stress values are also multiplied by the factor if exists.

Examples:

1. **mnf2mtx.exe input.mnf -O output.MASTER -I 11,0.0,0.5,10.0,121,1.0,0.75,15.0**

   This example edits input.mnf for the coordinates of interface node 11 and 121 to X=0.0, Y=0.5 and Z=10.0 and X=1.0, Y=0.75 and Z=15.0 respectively. The edited flexible body is saved to MD DB output.MASTER. If output.MASTER exists, the edited flexible body will be appended to it.

2. **mnf2mtx.exe input.MASTER::3 -O output.mnf -mscale 1.2 -sscale 0.8**

This example edits the 3rd flexible body in input.MASTER to scale generalized mass with the factor of 1.2 and scale generalized stiffness with the factor of 0.8. The edited flexible body is saved as output.mnf.

Theory of Flexible Bodies

Introduction

This manuscript gives you a working knowledge of the theory used to implement flexible bodies in Adams. The topics covered include:

- modal superposition
- component mode synthesis
- mode shape orthonormalization
- kinematics of markers on flexible bodies
- applied forces
- flexible body equations of motion

History of Flexible Bodies in Adams

The first attempt to automatically interface with Finite Element Method (FEM) software was in a product called Adams FEA. In Adams FEA the FEM software used *Guyan Reduction* to automatically condense the entire set of FEM degrees of freedom (DOF) to a reduced number of DOF.

In the Guyan reduction method, a set of user-defined *master* nodes are retained and the remaining set of *slave* nodes are removed by condensation. Only stiffness properties are considered during the condensation, and inertia coupling of master and slave nodes are ignored. This is why Guyan reduction is sometimes referred to as *static condensation*.

Guyan reduction condenses the large, sparse FEM mass and stiffness matrices down to a small, dense pair of matrices, with respect to the master DOF.

The challenge in Adams FEA was to represent the master nodes using PART elements and an NFORCE element. While the condensed stiffness could be captured correctly by the NFORCE, the dense, condensed mass matrix from the Guyan reduction did not always lend itself to being represented by an “equivalent” lumped mass matrix.

The goals of matching:

- total mass
- center-of-mass location
- moments of inertia
- natural frequencies

could not always be met. Adams FEA was difficult to use successfully and did not win favor with Adams customers.

In 1996 an alternative modal flexibility method was introduced in a product called Adams Flex. Rather than being based on Adams primitives like PART and NFORCE elements, Adams Flex introduced a new inertia element, the FLEX_BODY.

Modal Superposition

The single most important assumption behind the FLEX_BODY is that we only consider small, linear body deformations relative to a local reference frame, while that local reference frame is undergoing large, non-linear global motion.

The discretization of a flexible component into a finite element model represents the infinite number of DOF with a finite, but very large number of finite element DOF. The linear deformations of the nodes of this finite element mode, $\mathbf{u}$, can be approximated as a linear combination of a smaller number of shape vectors (or mode shapes), ϕ.

$$u = \sum_{i=1}^{M} \phi_i q_i \tag{1}$$

where M is the number of mode shape. The scale factors or amplitudes, q, are the modal coordinates.

As a simple example of how a complex shape is built as a linear combination of simple shapes, observe the following illustration:

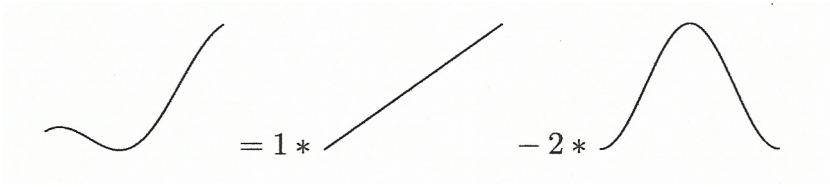


Figure 1 Complex shape built as a linear combination of simple shapes

The basic premise of modal superposition is that the deformation behavior of a component with a very large number of nodal DOF can be captured with a much smaller number of modal DOF. We refer to this reduction in DOF as modal truncation.

Equation (1) is frequently presented in a matrix form:

$$u = \Phi q \tag{2}$$

where q is the vector of modal coordinates and the modes ϕ_i have been deposited in the columns of the modal matrix, Φ. After modal truncation Φ becomes a rectangular matrix. The modal matrix Φ is the transformation from the small set of modal coordinates, q, to the larger set of physical coordinates, u.

This raises the question: How do we select the mode shapes such that the maximum amount of interesting deformation can be captured with a minimum number of modal coordinates? In other words, how do we optimize our modal basis?

Component Mode Synthesis--The Craig-Bampton Method

In an early release of Adams Flex it was assumed that eigenvectors would provide a useful modal basis. To prevent accidental constraints in the system, it was recommended that the eigenvectors of an unconstrained system be used.

Users struggled trying to capture the effects of attachments on the flexible body. To achieve model fidelity, an excessive number of modes was often required. Eigenvectors were found to provide an inadequate basis in system level modeling.

The solution was to use Component Mode Synthesis (CMS) techniques, and the most general methodology was adopted, the Craig-Bampton method.

The Craig-Bampton method allows the user to select a subset of DOF which are not to be subject to modal superposition. These DOF, which we refer to as *boundary* DOF (or *attachment* DOF or *interface* DOF), are preserved exactly in the Craig-Bampton modal basis. There is no loss in resolution of these DOF when higher order modes are truncated.

The Craig-Bampton method achieves this with a very simple scheme. The system DOF are partitioned into boundary DOF, u_B, and *interior* DOF, u_I. Two sets of mode shapes are defined, as follows:

- **Constraint modes:** These modes are static shapes obtained by giving each boundary DOF a *unit* displacement while holding *all* other boundary DOF fixed. The basis of constraint modes completely spans all possible motions of the boundary DOFs, with a one-to-one correspondence between the modal coordinates of the constraint modes and the displacement in the corresponding boundary DOF, $q_C = u_B$.

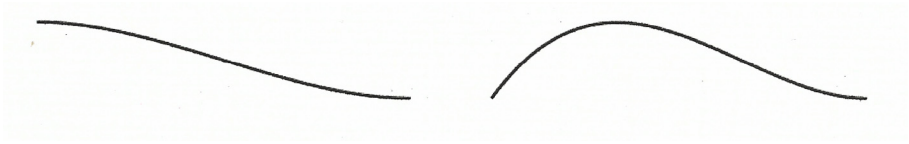


Figure 2 Two *constraint modes* for the left end of a beam that has attachment points at the two ends. The figure on the left shows the constraint mode corresponding to a unit translation while the figure on the right corresponds to a unit rotation.

- **Fixed-boundary normal modes:** These modes are obtained by fixing the boundary DOF and computing an eigensolution. There are as many fixed-boundary normal modes as the user desires. These modes define the modal expansion of the interior DOF. The quality of this modal expansion is proportional to the number of modes retained by the user.

Figure 3 Two *fixed-boundary normal modes* for a beam that has attachment points at the two ends.

The relationship between the physical DOF and the Craig-Bampton modes and their modal coordinates is illustrated by the following equation.

$$u = \begin{Bmatrix} u_B \\ u_I \end{Bmatrix} = \begin{bmatrix} I & 0 \\ \Phi_{IC} & \Phi_{IN} \end{bmatrix} \begin{Bmatrix} q_C \\ q_N \end{Bmatrix} \tag{3}$$

where:

u_B	=	the boundary DOF.
u_I	=	the interior DOF.
$I, 0$	=	identity and zero matrices, respectively.
Φ_{IC}	=	the physical displacements of the interior DOF in the constraint modes.
Φ_{IN}	=	the physical displacements of the interior DOF in the normal modes.
q_C	=	the modal coordinates of the constraint modes.
q_N	=	the modal coordinates of the fixed-boundary normal modes.

The generalized stiffness and mass matrices corresponding to the Craig-Bampton modal basis are obtained via a modal transformation. The stiffness transformation is:

$$\hat{K} = \Phi^T K \Phi = \begin{bmatrix} I & 0 \\ \Phi_{IC} & \Phi_{IN} \end{bmatrix}^T \begin{bmatrix} K_{BB} & K_{BI} \\ K_{IB} & K_{II} \end{bmatrix} \begin{bmatrix} I & 0 \\ \Phi_{IC} & \Phi_{IN} \end{bmatrix} = \begin{bmatrix} \hat{K}_{CC} & 0 \\ 0 & \hat{K}_{NN} \end{bmatrix} \tag{4}$$

while the mass transformation is:

$$\hat{M} = \Phi^T M \Phi = \begin{bmatrix} I & 0 \\ \Phi_{IC} & \Phi_{IN} \end{bmatrix}^T \begin{bmatrix} M_{BB} & M_{BI} \\ M_{IB} & M_{II} \end{bmatrix} \begin{bmatrix} I & 0 \\ \Phi_{IC} & \Phi_{IN} \end{bmatrix} = \begin{bmatrix} \hat{M}_{CC} & \hat{M}_{NC} \\ \hat{M}_{CN} & \hat{M}_{NN} \end{bmatrix} \tag{5}$$

where the subscripts I, B, N, and C denote internal DOF, boundary DOF, normal mode and constraint mode, respectively. The caret on $\hat{M}$ and $\hat{K}$ denotes that this is *generalized* mass and stiffness.

Equation (4) and Equation (5) have a few noteworthy properties:

- $\hat{M}_{NN}$ and $\hat{K}_{NN}$ are diagonal matrices because they are associated with eigenvectors.

- $\hat{K}$ is block diagonal. There is no stiffness coupling between the constraint modes and fixed-boundary normal modes.
- Conversely, $\hat{M}$ is *not* block diagonal because there is inertia coupling between the constraint modes and the fixed-boundary normal modes.

Mode Shape Orthonormalization

The Craig-Bampton method is a powerful method for tailoring the modal basis to capture both the desired attachment effects and the desired level of dynamic content. However, the raw Craig-Bampton modal basis has certain deficiencies that make it unsuitable for direct use in a dynamic system simulation. These are:

1. Embedded in the Craig-Bampton constraint modes are 6 rigid body DOF which must be eliminated before the Adams analysis because Adams provides its own large-motion rigid body DOF.
2. The Craig-Bampton constraint modes are the result of a static condensation. Consequently, these modes do not advertise the dynamic frequency content that they must contribute to the flexible body. Successful simulation of a non-linear system with unknown frequency content is unlikely.
3. Craig-Bampton constraint modes cannot be disabled because to do so would be equivalent to applying a constraint on the system.

These problems with the raw Craig-Bampton modal basis are all resolved by applying a simple mathematical operation on the Craig-Bampton modes.

The Craig-Bampton modes are not an orthogonal set of modes, as evidenced by the fact that their generalized mass and stiffness matrices $\hat{K}$ and $\hat{M}$, encountered in Equation (4) and Equation (5), are not diagonal.

By solving an eigenvalue problem:

$$\hat{K}q = \lambda \hat{M}q \tag{6}$$

we obtain eigenvectors that we arrange in a transformation matrix N, which transforms the Craig-Bampton modal basis to an equivalent, orthogonal basis with modal coordinates q^*:

$$Nq^* = q \tag{7}$$

The effect on the superposition formula is:

$$u = \sum_{i=1}^{M} \phi_i q_i = \sum_{i=1}^{M} \phi_i Nq^* = \sum_{i=1}^{M} \phi_i{}^* q^* \tag{8}$$

where ϕ_i are the orthogonalized Craig-Bampton modes.

The orthogonalized Craig-Bampton modes are not eigenvectors of the original system. They are eigenvectors of the Craig-Bampton *representation* of the system and as such have a *natural frequency* associated with them. A physical description of these modes is difficult, but in general the following is observed:

- Fixed-boundary normal modes are replaced with an approximation of the eigenvectors of the unconstrained body. This is an approximation because it is based only on the Craig-Bampton modes. Out of these modes, 6 modes are usually the rigid body modes.
- Constraint modes are replace with *boundary eigenvector*, a concept best illustrated by comparing the modes before and after orthogonalization of a rectangular plate which has Craig-Bampton attachment points along one of its long edges. The Craig-Bampton mode in Figure 4 features a unit displacement of one of its edge nodes with all the other nodes along that edge fixed. After orthonormalization we see modes like the one depicted in Figure 5, which has a sinusoidal curve along the boundary edge.
- Finally, there are modes in a gray area between the first two sets that defy physical classification.

We conclude that the orthonormalization of the Craig-Bampton modes addresses the problems identified earlier, because:

1. Orthonormalization yields the modes of the unconstrained system, 6 of which are rigid body modes, which can now be disabled.

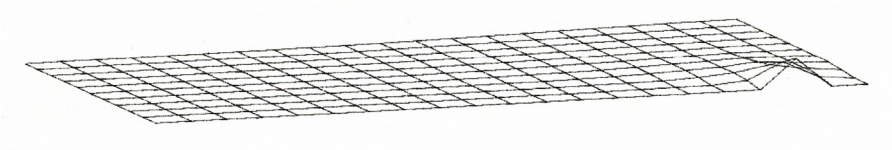


Figure 4 Constraint mode with an unknown frequency contribution.

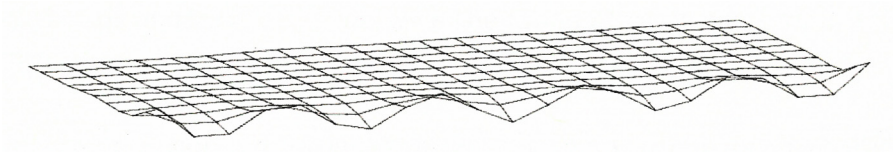


Figure 5 *Boundary eigenvector* with a 1250 Hz natural frequency.

2. Following the second eigensolution, all modes have an associated natural frequency. Problems arising from modes contributing high-frequency content can now be anticipated.
3. Although the removal of any mode constrains the body from adopting that particular shape, the removal of a high-frequency mode such as the one depicted in Figure 5 is clearly more benign than removing the mode depicted in Figure 4. The removal of the latter mode prevents the associated boundary node from moving relative to its neighbors. Meanwhile, the removal of the former mode only prevents boundary edge from reaching this degree of "waviness."

Modal flexibility in Adams

In this section we show how Adams capitalizes on modal superposition in the two key areas of the Adams formulation:

- Flexible marker kinematics
- Flexible body equations of motion

Flexible Marker Kinematics

Marker kinematics refers to the position, orientation, velocity, and acceleration of markers. Adams uses the kinematics of markers in three key areas:

- Marker position and orientation must be known in order to satisfy constraints like those imposed in JOINT and JPRIM elements.
- To project point forces applied at markers on generalized coordinates of the flexible body.
- The marker measures, (for example DX, WZ, PHI, ACCX, and so on) that appear in expressions and user-written subroutines require information about position, orientation, velocity, and acceleration of markers

Position

The instantaneous location of a marker that is attached to a node, P, on a flexible body, B, is the sum of three vectors (see Figure 6).

$$\vec{r}_p = \vec{x} + \vec{s}_p + \vec{u}_p \tag{9}$$

where:

$\vec{x}$	=	the position vector from the origin of the ground reference frame to the origin of the local body reference frame, B, of the flexible body.
$\vec{s}_p$	=	the position vector of the undeformed position of point P with respect to the local body reference frame of body B.
$\vec{u}_p$	=	the translational deformation vector of point P, the position vector from the point's undeformed position to its deformed position

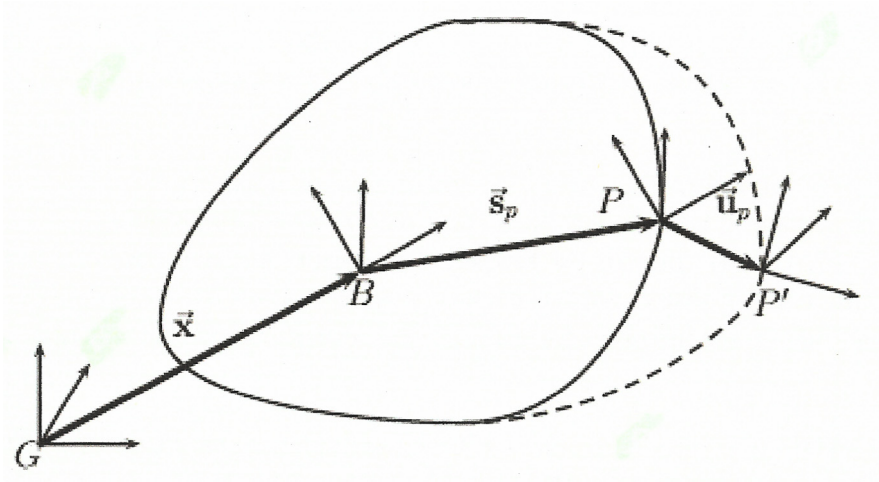


Figure 6 The position vector to a deformed point P' on a flexible body relative to a local body reference frame B and ground G

We rewrite Equation (9) in a matrix form, expressed in the ground coordinate system

$$r_p = x + {}^{G}A^{B}(s_p + u_p) \tag{10}$$

where:

x	=	the position vector from the ground origin to the origin of the local body reference frame, B, of the flexible body, expressed in the ground coordinate system. The elements of the x vector, x, y and z, are generalized coordinates of the flexible body.
s_p	=	the position vector from the local body reference frame of B to the point P, expressed in the local body coordinate system. This is a constant.
${}^{G}A^{B}$	=	the transformation matrix from the local body reference frame of B to ground. This matrix is also known as the direction cosines of the local body reference frame with respect to ground. In Adams, orientation is captured using a body fixed 3-1-3 set of Euler angles, ψ, θ and ϕ. The Euler angles are generalized coordinates of the flexible body.
u_p	=	the translational deformation vector of point P, also expressed in the local body coordinate system. The deformation vector is a modal superposition. $u_p = \Phi_p q$ (11) where Φ_p is the slice from the modal matrix that corresponds to the translational DOF of node P. The dimension of the Φ_p matrix is $3 \times M$ where M is the number of modes. The modal coordinates q_i, $i = 1 \ldots M$ are generalized coordinates of the flexible body.

Therefore, the generalized coordinates of the flexible body are:

$$\xi = \left\{ \begin{array}{c} x \\ y \\ z \\ \psi \\ \theta \\ \phi \\ q_i, (i = 1 \ldots M) \end{array} \right\} = \left\{ \begin{array}{c} x \\ \psi \\ q \end{array} \right\} \tag{12}$$

Velocity

For the purpose of computing kinetic energy, we compute the instantaneous translational velocity of P relative to ground which is obtained by differentiating Equation (10) with respect to time:

$$v_p = \dot{x} + {}^{G}\dot{A}^{B}(s_p + u_p) + {}^{G}A^{B}\,\dot{u}_p \tag{13}$$

Taking advantage of the relationship:

$${}^{G}\dot{A}^{B}\,s = {}^{G}A^{B}({}^{G}\omega_{B}^{B} \times s) = {}^{G}A^{B}\,{}^{G\tilde{}}\omega_{B}^{B}\,s = -{}^{G}A^{B}\tilde{s}\,{}^{G}\omega_{B}^{B} \tag{14}$$

where ${}^{G}\omega_{B}^{B}$ is the angular velocity of the body relative to ground (expressed in body coordinates with the tilde denoting the skew operator of Equation (19) we can write:

$$v_p = \dot{x} - {}^{G}A^{B}(\tilde{s}_p + \tilde{u}_p)B\dot{\psi} + {}^{G}A^{B}\Phi_p\dot{q} \tag{15}$$

We have introduced the relationship:

$${}^{G}\omega_{B}^{B} = B\dot{\psi} \tag{16}$$

relating the angular velocity to the time derivative of the orientation states.

Orientation

To satisfy angular constraints, Adams must instantaneously evaluate the orientation of a marker on a flexible body, as the body deforms. As the body deforms, the marker rotates through *small* angles relative to its reference frame. Much like translational deformations, these angles are obtained using a modal superposition, similar to Equation (11)

$$\theta_p = \Phi_p^{*} q \tag{17}$$

where Φ_p^* is the slice from the modal matrix that corresponds to the *rotational* DOF of node P. The dimension of the Φ_p^* matrix is $3 \times M$ where M is the number of modes.

The orientation of marker J relative to ground is represented by the Euler transformation matrix, ${}^G A^J$. This matrix is the product of three transformation matrices:

$${}^G A^J = {}^G A^B \, {}^B A^P \, {}^P A^J \tag{18}$$

where:

${}^G A^B$	=	the transformation matrix from the local body reference frame of B to ground.
${}^B A^P$	=	the transformation matrix due to the orientation change due to the deformation of node P.
${}^P A^J$	=	the constant transformation matrix that was defined by the user when the marker was placed on the flexible body.

The matrix ${}^B A^P$ requires more attention. The direction cosines for a vector of small angles, θ_p, are:

$${}^B A^P = \begin{bmatrix} 1 & -\theta_{pz} & \theta_{py} \\ \theta_{pz} & 1 & -\theta_{px} \\ -\theta_{py} & \theta_{px} & 1 \end{bmatrix} = I + \tilde{\theta}_p \tag{19}$$

where the tilde denotes the skew operator:

$$a \times b = \begin{bmatrix} 0 & -a_z & a_y \\ a_z & 0 & -a_x \\ -a_y & a_x & 0 \end{bmatrix} b = \tilde{a} b = -\tilde{b} a \tag{20}$$

Angular velocity

The angular velocity of a marker, J, on a flexible body is the sum of the angular velocity of the body and the angular velocity due to deformation:

$${}^G\omega_B^J = {}^G\omega_B^P = {}^G\omega_B^B + {}^B\omega_B^P = {}^G\omega_B^B + \Phi_p^* \dot{q} \tag{21}$$

Applied loads

The treatment of forces in Adams distinguishes between point loads and distributed loads. This section discusses the following topics:

- Point forces and torques
- Distributed loads
- Residual forces and residual vectors

Point Forces and Torques

A point force $\vec{F}$ and a point torque $\vec{T}$ that are applied to a marker on a flexible body must be projected on the generalized coordinates of the system.

The force and torque are written in matrix form, and expressed in the coordinate system of marker K.

$$F_K = \begin{bmatrix} f_x \\ f_y \\ f_z \end{bmatrix} \qquad T_K = \begin{bmatrix} t_x \\ t_y \\ t_z \end{bmatrix} \tag{22}$$

The generalized force Q consists of a generalized translational force, a generalized torque (a generalized force on the Euler angles) and a generalized modal force, thus:

$$Q = \begin{bmatrix} Q_T \\ Q_R \\ Q_M \end{bmatrix} \tag{23}$$

Generalized Translational Force: Since the governing equations of motion, Equation (41), are written in the global reference frame, the generalized force on the translational coordinates is obtained by transforming F_K to global coordinates.

$$Q_T = {}^{G}A^{K} F_K \tag{24}$$

where ${}^{G}A^{K}$ is given in Equation (18). The generalized translational force is independent of the point of force application.

An applied torque does not contribute to Q_T.

Generalized Torque: The total torque on a flexible body, due to $\vec{F}$ and $\vec{T}$ is $T_{tot} = \vec{T} + \vec{p} \times \vec{F}$ where $\vec{p}$ is the position vector from the origin of the local body reference frame of the body to the point of force application. The total torque, can be written in matrix form, with respect to the ground coordinate system as:

$$T_{tot} = {}^{G}A^{K}\, T_K + p \times {}^{G}A^{K}\, F_K \tag{25}$$

where p is expressed in the ground coordinates. Using the tilde notation of Equation (20), this can be written as:

$$T_{tot} = {}^{G}A^{K}\, T_K + \tilde{p}\,{}^{G}A^{K}\, F_K \tag{26}$$

The transformation from torque in physical coordinates to the generalized torque on the body Euler angles is provided by the B matrix in Equation (16).

$$Q_R = [{}^{G}A^{B}\, B]^{T}\, T_{tot} = [{}^{G}A^{B}\, B]^{T}[{}^{G}A^{K}\, T_K + \tilde{p}\,{}^{G}A^{K}\, F_K] \tag{27}$$

Generalized Modal Force: The generalized modal force on a body due to applied point forces or point torques at P is obtained by projecting the load on the mode shapes.

As the applied force F_K and torque T_K are given with respect to marker K, they must first be transformed to the reference frame of the flexible body:

$$F_I = {}^{G}A^{B\,T}\,{}^{G}A^{K}\, F_K \tag{28}$$

$$T_I = {}^{G}A^{B\,T}\,{}^{G}A^{K}\, T_K \tag{29}$$

and then projected on the mode shapes. The force is projected on the translational mode shapes and the torque is projected on the angular mode shapes:

$$Q_M = \Phi_P^{T}\, F_I + \Phi_P^{*T}\, T_I \tag{30}$$

where Φ_p and Φ_P^{*} are the slices of the modal matrix corresponding to the translational and angular DOF of point P, as discussed in Flexible Marker Kinematics.

Note that since the modal matrix Φ is only defined at nodes, point forces and point torques can only be applied at nodes.

Distributed Loads

Although distributed loads can be generated in Adams as an array of point loads, this is rarely an efficient approach. As an alternative, distributed loads can be created in Adams using the MFORCE element. The MFORCE statement allows you to apply any distributed-load vector.

A discussion of distributed loads starts by examining the physical coordinate form of the equations of motion in the finite element modeling software.

$$M\ddot{x} + Kx = F \tag{31}$$

Here M and K are the FEM mass and stiffness matrices for the flexible component, and x and F are, respectively, the physical nodal DOF vector and the load vector.

Equation (31) is transformed into modal coordinates q using the modal matrix Φ:

$$\Phi^T M \Phi \ddot{q} + \Phi^T K \Phi q = \Phi^T F \tag{32}$$

This modal form of the equation simplifies to:

$$\hat{M}\ddot{q} + \hat{K}q = f \tag{33}$$

where $\hat{M}$ and $\hat{K}$ are the generalized mass and stiffness matrices and f is the modal load vector.

The applied force is likely to have a global resultant force and torque. These show up as loads on the rigid body modes and are treated in Adams as point forces and torques on the local reference frame, as covered in the previous section. The global resultant force and torque are not discussed further.

The projection of the nodal force vector on the modal coordinates:

$$f = \Phi^T F \tag{34}$$

is a computationally expensive operation, which poses a problem when F is a arbitrary function of time. Adams circumvents this problem by introducing the simplifying assumption that the spatial dependency and the time dependency can be separated, i.e., that the load can be viewed as a time varying linear combination of an arbitrary number of static load cases:

$$F(t) = s_1(t)F_1 + \ldots + s_n(t)F_n \tag{35}$$

Then the expensive projection of the load to modal coordinates can be performed once during the creation of the MNF, rather than repeatedly during the Adams simulation. Adams need only be aware of the modal form of the load:

$$f(t) = s_1(t)f_1 + \ldots + s_n(t)f_n \tag{36}$$

where the vectors f_1 to f_n are n different load case vectors. Each of the load case vectors contains one entry for each mode in the modal basis.

A more generous definition of f allows it to have an explicit dependency on system response, which we will denote as $f(q, t)$, where q now represents *all* the states of the system, not just those of the flexible body. The equation for the modal force can now be written as:

$$f(q, t) = s_1(q, t)f_1 + \ldots + s_n(q, t)f_n \tag{37}$$

Residual Forces and Residual Vectors

Implicit in the discussion in the previous sections is the assumption that the modal projection of the applied force:

$$f = \Phi^T F \tag{38}$$

is exhaustive. However, due to mode truncation, in practice this is not always the case. In some cases, some amount of force remains *unprojected*. We refer to this force as the residual force. One might think about this as the load that was projected on the neglected higher-order modes.

The value of the residual force could be evaluated as:

$$\Delta F = F - [\Phi^T]^{-1} f \tag{39}$$

Associated with a residual force is residual vector, which can be thought of as the deformed shape of the flexible body when the residual force is applied to it. This residual vector can be treated as a mode shape and added to the Craig-Bampton modal basis. This enhanced basis completely captures the applied load. Without this enhancement, the residual force is irretrievably lost.

There are two load cases where residual force is not a concern:

- Point forces or torques on Craig-Bampton boundary nodes. The nature of the Craig-Bampton modal basis is such that point loads on the boundary nodes project perfectly on the corresponding constraint modes.
- Uniform distributed loads. Uniform distributed loads project completely on the rigid body DOFs.

There is one special case of force truncation that deserves mention. This case is best illustrated by considering a FEM node with incomplete stiffness, as found on solid elements or shell elements. Applying a load to this node leads to a singularity in the FEM analysis. When Craig-Bampton modes are generated for this model, they will share a common attribute--the mode shape entry for this DOF is zero in all the modes. Consequently, any attempts in Adams to apply a load in this DOF will fail, because the load does not project on any of the modes and the structure will appear infinitely stiff. It is recommended that *no loads be applied in Adams that could not have been applied in the FEM software*.

Preloads

Adams supports preloaded flexible bodies. This allows Adams to support non-linear FEM analyses by accepting flexible bodies that have been linearized in a deformed state. These modes would not otherwise be considered candidates for a modal representation in Adams.

However, in certain Adams analyses the deformations of the non-linear component might safely be assumed to remain within a small range around a fixed operating point and a linearization of the body about this

operating point could yield a useful modal representation of the body. A non-linear finite element model of the body is brought to this operating point by applying some combination of loads. The body is linearized at the operating point and the modes are extracted and exported to Adams.

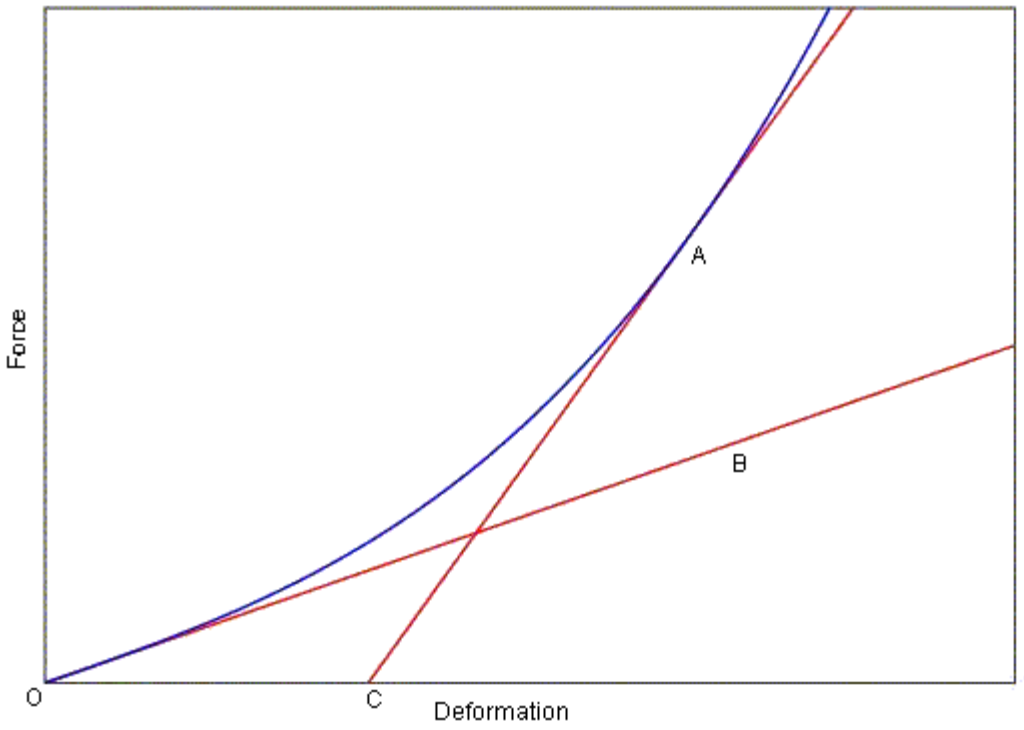


Figure 7 Force Deformation plot

Figure 7 illustrates the force-deformation relationship of the process described above. The undeformed state is defined by operating point O. As the body deforms, it is brought through a *non-linear* path to a deformed state A. A linear model of the body at O, such as might have been defined by an Adams flexible body, would incorrectly have predicted an operating point at B rather than at A. Note further, that if the body is linearized at A, and a modal description exported to Adams in the form of a preloaded flexible body, a limited range of validity must also be observed. Fully unloading the Adams flexible body would bring it to operating point C, which is not correct.

A preload is applied in Adams in the same way modal loads described in the previous section are applied, except that the preload is not under the user's control. The preload cannot be disabled or scaled because it is considered an immutable property of the flexible bodies with an associated deformed geometry. Only one preload can be defined for any given flexible body.

A preload is an *internal load* and as such only operates on the modal coordinates. There is no global resultant force. In other words, there is no load on the rigid body DOF. If this were otherwise, the flexible body would have a tendency to accelerate itself, which would be counterintuitive.

Unless the external load that gave rise to the preload is reapplied within Adams, the preloaded flexible body will recoil. If the flexible body originated from a linear finite element model, it will recoil to its undeformed shape. If the body came from a non-linear analysis, the effect will be more like that described in Figure 7. If the body is constrained to other bodies, this tendency to recoil will cause the body to push on the other bodies.

Flexible Body Equations of Motion

The governing equations for a flexible body are derived from Lagrange's equations of the form:

$$\frac{d}{dt}\left(\frac{\partial L}{\partial \dot{\xi}}\right) - \frac{\partial L}{\partial \xi} + \frac{\partial F}{\partial \dot{\xi}} + \left[\frac{\partial \Psi}{\partial \xi}\right]^T \lambda - Q = 0 \tag{40}$$

$$\Psi = 0 \tag{41}$$

where:

L	=	the Lagrangian, defined below.
F	=	an energy dissipation function, defined below.
Ψ	=	the constraint equations.
λ	=	the Lagrange multipliers for the constraints.
ξ	=	the generalized coordinates as defined in Equation (12).
Q	=	the generalized applied forces (the applied forces projected on ξ).

The Lagrangian is defined as:

$$L = T - V$$

where T and V denote kinetic and potential energy respectively.

The remainder of this section is devoted to the derivation of the contributions to Equation (41), in the following order:

- Kinetic energy and the mass matrix.
- Potential energy and the stiffness matrix.
- Dissipation and the damping matrix.
- Constraints.

Kinetic Energy and the Mass Matrix

The velocity from Equation (15) can be expressed in terms of the time derivative of the state vector ξ:

$$v_p = [\mathbf{I} \quad -{}^G A^B(\tilde{s_p} + \tilde{u_p})B \quad {}^G A^B \Phi_p]\dot{\xi} \tag{42}$$

We can now compute the kinetic energy. The kinetic energy for a flexible body is given as:

$$T = \frac{1}{2}\int_v \rho\, v^T v dV \approx \frac{1}{2}\sum_p m_p v_p^T v_p + {}^G\omega_P^{BT}\, \mathbf{I}_p\, {}^G\omega_P^B \tag{43}$$

where m_p and $\mathbf{I}_p$ are the nodal mass and nodal inertia tensor of node P, respectively. Note that $\mathbf{I}_p$ is often a negligible quantity which arises when reduced continuum descriptions, i.e. bars, beams, or shells, are employed in your flexible component model. Lumped masses and inertia may also contribute to this term.

Substituting for v and ω and simplifying yields an equation for the kinetic energy in Adams' generalized mass matrix and generalized coordinates.

$$T = \frac{1}{2}\dot{\xi}^T M(\xi)\, \dot{\xi} \tag{44}$$

For clarity of presentation we partition the mass matrix, $M(\xi)$, into a 3×3 block matrix:

$$M(\xi) = \begin{bmatrix} M_{tt} & M_{tr} & M_{tm} \\ M_{tr}^T & M_{rr} & M_{rm} \\ M_{tm}^T & M_{rm}^T & M_{mm} \end{bmatrix} \tag{45}$$

where the subscripts t, r, and M denote translational, rotational, and modal DOF respectively.

The expression for the mass matrix $M(\xi)$ simplifies to an expression in *nine* inertia invariants.

$$M_{tt} = I^1 \mathbf{I} \tag{46}$$

$$M_{tr} = -A[I^2 + I_j^3 \tilde{q_j}]B \tag{47}$$

$$M_{tm} = AI^3 \tag{48}$$

$$M_{rr} = B^T[I^7 - [I_j^8 + I_j^{8T}]q_j - I_{ij}^9 q_i q_j]B \tag{49}$$

$$M_{rm} = B^T[I^4 + I_j^5 q_j] \tag{50}$$

$$M_{mm} = I^6 \tag{51}$$

The explicit dependence of the mass matrix on the modal coordinates is evident. The dependence on orientation coordinates of the system comes about because of the transformation matrices A and B.

The inertia invariants are computed from the N nodes of the finite element model based on information about each node's mass, m_{pp}, its undeformed location s_p, and its participation in the component modes Φ_p. The discrete form of the inertia invariants are provided in Table 1.

Table 1 Discrete form of inertia invariants

$I^1 = \sum_{p=1}^{N} m_p$		(scalar)
$I^2 = \sum_{p=1}^{N} m_p s_p$		(3×1)
$I_j^3 = \sum_{p=1}^{N} m_p \Phi_p$	$j = 1, \ldots, M$	$(3 \times M)$
$I^4 = \sum_{p=1}^{N} m_p \tilde{s}_p \Phi_p + \mathbf{I}_p \Phi_p^*$		$(3 \times M)$
$I_j^5 = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \Phi_p$	$j = 1, \ldots, M$	$(3 \times M)$
$I^6 = \sum_{p=1}^{N} m_p \Phi_p^T \Phi_p + \Phi^{*T}_p \mathbf{I}_p \Phi_p^*$		$(M \times M)$

$I^7 = \sum_{p=1}^{N} m_p \tilde{s}_p^T \tilde{s}_p + \mathbf{I}_p$		(3×3)
$I_j^8 = \sum_{p=1}^{N} m_p \tilde{s}_p \tilde{\phi}_{pj}$	$j = 1, \ldots, M$	(3×3)
$I_{jk}^9 = \sum_{p=1}^{N} m_p \tilde{\phi}_{pj} \tilde{\phi}_{pk}$	$j, k = 1, \ldots, M$	(3×3)

Note: *phi-p-j* is mode shape vector {x, y, z} of node *p* of mode *j*, and *tilde-phipj* is skew matrix (3 X 3).

Potential Energy and the Stiffness Matrix

Frequently, the potential energy consists of contributions from gravity and elasticity in the quadratic form:

$$V = V_g(\xi) + \frac{1}{2}\xi^T K \xi \tag{52}$$

In the elastic energy term, K is the generalized stiffness matrix which is, in general, constant. Only the modal coordinates, q, contribute to the elastic energy. Therefore, the form of K is:

$$K = \begin{bmatrix} K_{tt} & K_{tr} & K_{tm} \\ K_{tr}^T & K_{rr} & K_{rm} \\ K_{tm}^T & K_{rm}^T & K_{mm} \end{bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & K_{mm} \end{bmatrix} \tag{53}$$

where K_{mm} is the *generalized* stiffness matrix of the structural component with respect to the modal coordinates, q. It is not the full structural stiffness matrix of the component.

V_g is the gravitational potential energy:

$$V_g = \int_v \rho \vec{r}_p \cdot \vec{g} dV = \int_v \rho [x + A(s_p + \Phi(P)q)]^T g dv \tag{54}$$

where g denotes the gravitational acceleration vector. The resulting gravitational force, f_g is:

$$f_g = \frac{\partial V_g}{\partial \xi} = \begin{bmatrix} \left[\int_v \rho dV\right] g \\ \left[\int_v \rho (s_p + \Phi(P)q)^T dV\right] \frac{\partial A^T}{\partial \psi} g \\ \left[\int_v \rho \Phi^T(P) dV\right] A^T g \end{bmatrix} \tag{55}$$

Dissipation and the Damping Matrix

The damping forces depend on the generalized modal velocities and are assumed to be derivable from the quadratic form:

$$F = \frac{1}{2} \dot{q}^T D \dot{q} \tag{56}$$

which is known as Rayleigh's dissipation function. The matrix D contains the damping coefficients, d_{ij}, and is generally constant and symmetric.

In the case of orthogonal mode shapes, the damping matrix can be effectively defined using a diagonal matrix of modal damping ratios, c_i. This damping ratio could be different for each of the orthogonal modes and can be conveniently defined as a ratio of the critical damping for the mode, c_i^{cr}. Recall that the critical damping ratio is defined as the level of damping that eliminates harmonic response as seen in the following derivation. Consider the simple harmonic oscillator defined by uncoupled mode i.

$$m_i \ddot{q}_i + c_i \dot{q}_i + k_i q_i = 0 \tag{57}$$

where m_i, k_i, and c_i denote, respectively, the generalized mass, the generalized stiffness, and the modal damping corresponding to mode i. Assuming the solution $q_i = e^{\lambda t}$, leads to a characteristic equation:

$$m_i \lambda^2 + c_i \lambda + k_i = 0 \tag{58}$$

which has the solution:

$$\lambda = \frac{-c_i \pm j\sqrt{4m_i k_i - c_i^2}}{2m_i} \tag{59}$$

The critical damping of mode i, is the one that eliminates the imaginary part of λ:

$$c_i^{cr} = 2\sqrt{k_i m_i} \tag{60}$$

Defining c_i as a ratio of critical damping introduces the modal damping ratio, η_i, which is referred to as CRATIO in the Adams dataset:

$$c_i = \eta_i c_i^{cr} \tag{61}$$

The solution to Equation (57) is:

$$q_i = e^{-\eta_i \omega_i t}\, e^{j\sqrt{(1-\eta^2)}\omega_i t} \tag{62}$$

where $\omega_i = \sqrt{k_i / m_i}$ is the natural frequency of the undamped system. This solution ceases to be harmonic when $\eta_i = 1$, which corresponds to 100% of critical damping

Constraints

Adams satisfies position and orientation constraints for flexible body markers by using the marker kinematics properties presented in Flexible Marker Kinematics. A more complete presentation of Adams joints is beyond the scope of this article.

Governing Differential Equation of Motion--Final Form

The final form of the governing differential equation of motion, in terms of the generalized coordinates is

$$M\ddot{\xi} + \dot{M}\dot{\xi} - \frac{1}{2}\left[\frac{\partial M}{\partial \xi}\dot{\xi}\right]^T \dot{\xi} + K\xi + f_g + D\dot{\xi} + \left[\frac{\partial \Psi}{\partial \xi}\right]^T \lambda = Q \tag{63}$$

The entries in Equation (63) are:

$\xi, \dot{\xi}, \ddot{\xi}$	=	the flexible body generalized coordinates and their time derivatives.
M	=	the flexible body mass matrix in Equation (45).
$\dot{M}$	=	the time derivative of the flexible body mass matrix.
$\frac{\partial M}{\partial \xi}$	=	the partial derivative of the mass matrix with respect to the flexible body generalized coordinates. This is a $(M+6)\times(M+6)\times(M+6)$ tensor, where M is the number of modes.

K	=	the generalized stiffness matrix.
f_g	=	the generalized gravitational force.
D	=	the modal damping matrix.
Ψ	=	the algebraic constraint equations.
λ	=	Lagrange multipliers for the constraints.
Q	=	generalized applied forces.

Dynamic Limit

Dynamic limit is a feature in Adams Solver (C++) to simplify the equations of motion of high frequency modes by ignoring the inertia terms while keeping the stiffness terms. This will potentially reduce the simulation time, when a significant number of high frequency modes are participating in the solution. Please refer to the DYNAMIC_LIMIT and STABILITY_FACTOR arguments of FLEX_BODY statement in Adams Solver (C++) documentation for detailed information on this feature.

Examples of Using Adams Flex

Examples

(Some examples may require internet access)

Tutorials of overall product use:

- Getting Started Using Adams Flex

Examples of Adams Flex features:

- Adams Linear to Verify Flexible Bodies
- Modal Loads
- Nonlinear Deformations
- Preloaded Flexible Body
- More examples of Adams Flex features
- Adams Verification Guide

Example of Modal Loads

Overview

In this tutorial, you'll explore how a steel plate, shown in Figure 1, responds to the distributed loads listed below. The steel plate is 1 meter square and 10 mm thick. It is supported vertically at all four corners and restrained against rotation in its plane.

- **Uniform pressure** - An overpressure of one atmosphere (atu) uniformly applied on the top surface.
- **Centered pressure** - An overpressure of one atu applied to a .4 M by .4 M square on the plate center.
- **Parabolic pressure** - An overpressure parabolically varying in the normal direction only, from a value of zero at the edge to one atu in the center.

You'll apply the loads by creating a modal force in Adams View. You'll then run a dynamic simulation to see how the first loadcase (uniform pressure) alters the plate over time. You'll then see the effects of disabling some of the modes and find the static equilibrium of the forces and plate flexibility. Finally, you'll run simulations using the other loadcases.

Each time you run a simulation, you'll review the results in Adams PostProcessor to see how they compare to similar data obtained from MSC.Nastran.

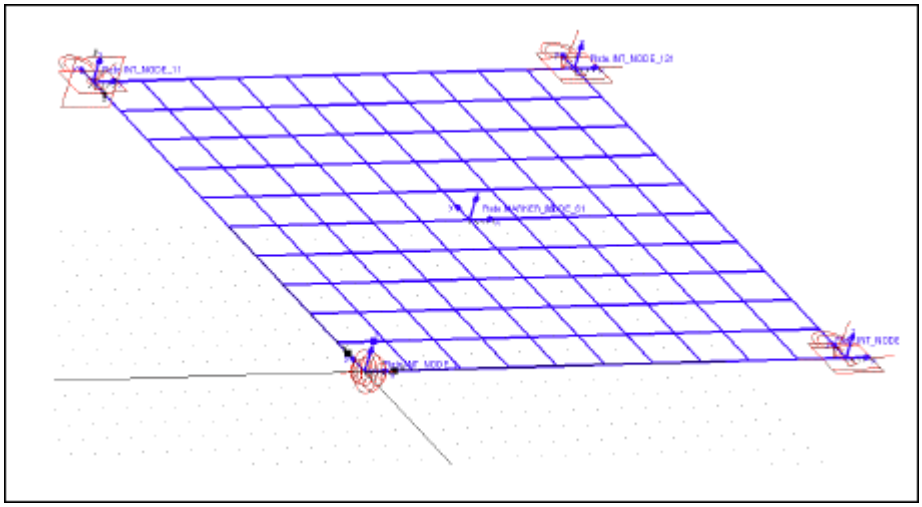


Figure 1 Steel Plate

Correlation with MSC.Nastran Data

For each of the applied loads listed on page 1, we ran a MSC.Nastran static solution. Table 1 compares the MSC.Nastran and Adams static results for the plate center deflections and corner reaction loads. During the tutorial, you'll compare the results that you obtain, with the results in the table.

Table 1 MSC.Nastran and Adams Results

Loadcase:	Static center deflection (M):		Corner reaction (N):	
	MSC.Nastran	Adams	MSC.Nastran	Adams
Uniform pressure-	.12533	-.12535	+25335	+25335
Centered pressure	-.02769	-.02835	+4053	+4054
Parabolic pressure	-.09372	-.09371	+17125	+17126

Note that Adams Flex has solved the problem using a reduced set of degrees of freedom (DOF). The Adams Flex solutions, therefore, do not match the MSC.Nastran solution exactly.

About the Tutorial Files

We've included three files to define the plate and the loads applied to it:

- plate.cmd - This file contains Adams View commands that create a model called Simple and integrate into the model a flexible body called Plate. The commands also constrain the plate at all four corners using one spherical joint, two inplane joint primitives, and one inline joint primitive. Finally, they create a marker at the center of the plate from which to measure displacements.

- plate_load.mnf - This file defines the flexible body and associates the loadcases in plate.loads with the flexible body. The file plate.cmd reads in this file.
- Before giving you this file, we used the tool mnfload to associate a loadcase file, plate.loads, with the flexible body. This loadcase file defines the three sets of loadcases for the plate. It is an ASCII file that lists the loads to be applied at each node. A comment that begins with a %C separates the different loadcases in the file. Adams View stores the comments as titles for the loadcases so you can select the ones you want to apply.

Starting Adams View

You'll start the tutorial by running Adams View and importing the command file, plate.cmd, to create a model called Simple.

To start Adams View and create a model:

1. Copy the file **plate.cmd** and **plate_load.mnf** to your local directory. This file is located in *install_dir*/flex/examples/mnfload, where *install_dir* is the directory where the Adams software is installed.
2. Start Adams View and import the command file **plate.cmd**.

   Adams View starts and the steel plate appears, as shown in Figure 1.
3. Rotate the plate so you can see it more easily (that is, not from directly above).

Creating a Measure

In this section, you create a point-to-point measure that measures the plate deformation at the center of the plate.

To create the measure:

1. Click the Design Exploration tab. From the **Measures** container, select the **Point-to-Point** tool  .
2. In the container on the Main toolbox, set **Component** to **Global Z**.
3. Following the instructions in the status bar, select **marker_22** as the location from which to measure.
4. Select **Plate.MARKER_NODE_61** as the location to which to measure.

Creating a Modal Force to Apply the Distributed Loads

Now you'll create a modal force (MFORCE) in Adams View to apply the distributed loads to the plate. You'll apply the load in the first loadcase, called one atu overpressure, which applies a uniform pressure on the plate.

To create a modal force:

1. Click the Forces tab. From the **Special Forces** container, select the **Modal Force** tool  .

   The Create Modal Force dialog box appears. Notice that the loadcase information does not appear in the dialog box. This is because you have not specified a flexible body yet.
2. In the **Flexible Body** text box, enter **Plate**.

   Adams View automatically enters the names of the loadcases associated with the flexible body Plate into the Load Case option menu.
3. Set **Load Case** to **one atu overpressure**.
4. In the **Scale Function** text box, enter a function to vary the pressure over time. In this case, enter the function:

   sin(PI*time)
5. Select **OK**.

   Adams View displays a MFORCE icon at the lower left corner (LPRF) of the plate.

Simulating the Distributed Loads

Now you'll run a dynamic simulation to see how Adams Flex applies the loads over time.

To run a dynamic simulation:

1. Click the Simulation tab. From the **Simulate** container, select the **Simulate** tool  .
2. Set **End Time** to **1.0** and **Steps** to **50**.
3. Select the **Simulation Start** tool. 

   You see the plate deform as Adams Flex applies the loads.

Reviewing the Results of the Simulation

You'll now review the results of the simulation, which include:

- Point-to-point measure to see how the location of the center of the plate (Node 61) changed relative to marker_22.
- Reaction forces (FZ) at the corners of the plate.
- Modal forces of the various modes (FQ). The sum of all the modes equals the applied force.

To review the results of the measure:

1. From the **Review** menu, select **PostProcessing**.

Adams PostProcessor appears.

2. Set **Source** to **Measures**.
3. From the **Measure** list, select **MEA_PT2PT_1**, and then select **Add Curves**.

   The plot shows the time history of the plate center deflection.
4. Compare the maximum deflection that Adams Flex returned, with the maximum deflection from MSC.Nastran, listed in Figure 1. Notice that the Adams Flex deflection is slightly larger in magnitude because it contains a small dynamic component that is not present in the MSC.Nastran solution. You will compute the static solution later in the tutorial (Running a Time-Independent (Static) Simulation.)

To review the results of the reaction forces:

1. From the dashboard in Adams PostProcessor, set **Source** to **Objects**.
2. From the **Filter** list, select **constraint**.
3. From the **Object** list, select one of the joints that holds the plate in place. For example, select **Joint_4**, the spherical joint in the lower left corner of the plate.
4. From the **Characteristic** list, select **Element_Force**.
5. From the **Component** list, select **Z**.
6. Select **Surf**.
7. Continue selecting joints and their reaction force. You will notice that the reaction forces are the same for all joints.
8. Compare the reaction forces with the forces obtained from MSC.Nastran shown in Table 1. Again, the dynamic results differ from the static results.

To review the modal forces:

1. From the dashboard in Adams PostProcessor, set **Source** to **Result Sets**.
2. From the **Result Set** list, select **MFORCE_1**.
3. From the **Component** list, select any modal force component. The modal force components are listed as FQn, where n is the number of the mode. Note that most of the modes have zero contributions, whereas a few of the modes (9, 14, 20, 25, and 36) have non-zero contributions.

   For Adams Flex to accurately model this particular distributed load on the plate, each of these modes needs to be enabled. If one of these modes is disabled, its contribution to the modal force is not included in the simulation. You'll see the effects of this in the next section.
4. When you are done, from the **File** menu, select **Close Plot Window** to return to the Adams View main window.

Viewing Effect of Disabling Modes

Now you'll disable modes 20 to 48. You'll then run another dynamic simulation to see the effects of disabling modes on the maximum deflection of the plate center and to see how the reaction forces at the joints compare to the previous simulation. The point-to-point measure calculates the plate center deflection.

To disable modes:

1. In the Main Menu, select the **Select** tool  to reset the model.
2. Display the Flexible Body Modify dialog box.
3. Select **range**.
4. In the Enable or Disable a Range of Modes dialog box, set the options to disable all modes above 19 based on mode number.
5. Select **OK**.

To run a simulation:

- Run the simulation using the same values as before (**Dynamic**, **one second**, and **50 steps**).

To review the results:

1. Follow the instructions in Reviewing the Results of the Simulation on page 6, to review the point-to-point measure and joint forces.
2. As you review them, notice that the maximum deflection of the plate center is significantly smaller (approximately 8 to 10% smaller), now that you've disabled the modes. Note, however, that the sum of the reaction forces remains the same.

   In the previous section of this tutorial, you found that modes 20, 25, and 36 contributed significantly to the associated distributed load. Because you disabled these modes in this section, modal load truncation occurred, resulting in a smaller deflection of the plate.

   The resulting reaction forces on the joints were barely affected by the modal truncation because Adams Flex derives them from the forces and torques associated with flexible body's six rigid body degrees of freedom. These six forces and torques represent the external resultant of the distributed loadcase projected by all of the modes during the MNF to matrix translation.

Running a Time-Independent (Static) Simulation

In this section, you'll modify the modal force so it is constant over time and perform a static equilibrium simulation, comparing the results with MSC.Nastran results.

To modify the MFORCE:

1. Right-click the MFORCE icon. From the shortcut menu that appears, point to the modal forces, and then select **Modify**.

   The Modify Modal Force dialog box appears.
2. In the **Scale Function** text box, remove the function and enter **1.0**.
3. Select **OK**.

Enable the modes:

1. Display the Flexible Body Modify dialog box.
2. Select **range** and enable all the non-rigid body modes (7 through 48).

To run a static equilibrium:

1. Click the Simulation tab. From the **Simulate** container, select the **Simulate** tool  .
2. Select the **Static Equilibrium** tool. 

To review the results:

1. Review the strip chart of the measure by noting the value in the legend.
2. Compare the static displacement of the plate center with those obtained from MSC.Nastran, shown in Table 1.

Simulating Remaining Loadcases

Now go ahead and run Adams simulations of the remaining two loadcases:

- **Centered pressure** - An overpressure of one atu applied to a .4 M by .4 M square on the plate center.
- **Parabolic pressure** - An overpressure parabolically varying in the x-direction only, from a value of zero at the edge to 1 atu in the center.

To run the simulations:

1. Modify the modal force so it refers to one of the loadcases listed above.
2. Perform a static simulation.
3. Compare with the values in Table 1.

Example of Modeling Nonlinear Deformations

Overview

Adams Flex models deformations that are always linear. You can easily model nonlinear deformations, however, by piecing together smaller elements that each deform linearly but together represent nonlinear deformations. This example steps you through how you would model nonlinear deformations.

- About the Example
- Starting Adams View and Integrating the MNF
- Integrating the Short Flexible Beam
- Assembling a Sectioned Beam

- Building a Single Flexible Body Beam
- Adding a Spinning Shaft
- Simulating the Beams
- Animating the Results from a Top View

About the Example

To illustrate how you would model nonlinear deformations, we've provided an example based on the problem of the spin-up cantilever beam ("*Dynamics of a Cantilever Beam Attached to a Moving Base*," Kane, T.R., Ryan, R.R. and Banarjee, A.K., 1987, Journal of Guidance, Control and Dynamics, Vol.13, No.2). The cantilever beam is shown in Figure 2.

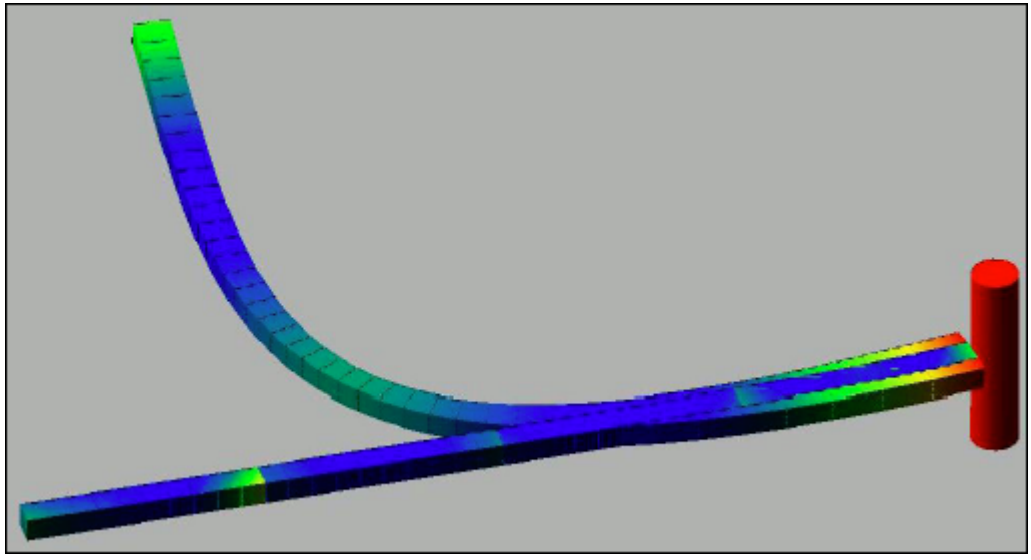


Figure 2 Cantilever Beam

The modal formulation of a flexible body is inherently linear. This example demonstrates how this premature linearization assumption prevents Adams from recognizing the stiffening of the beam due to centrifugal forces. When the nonlinear effects are not modeled, Adams can only predict that the beam will collapse.

To show you how you can avoid this problem, we'll have you create two beams. One beam you'll create by piecing together smaller beams and the second beam you'll create using a single, continuous beam part. You'll connect both beams to a spinning shaft. You'll then run a simulation to compare the results for the two types of beams. During the simulation, you'll discover that the single, continuous beam fails due to divergence.

Starting Adams View and Integrating the MNF

You'll start the example by running Adams View and creating a new model. You'll then integrate the MNF into the new model.

To start Adams View and create a database for the flexible body:

1. Copy the file **quarter_beam.mnf** and **full_beam.mnf** to your local directory. This file is located in *install_dir*/flex/examples/mnf, where *install_dir* is the directory where the Adams software is installed.
2. Start Adams View and create a new model named spin that has no gravity. Use the default units.

Integrating the Short Flexible Beam

Now you'll integrate the MNF containing a definition of a short flexible beam to be used as the base for the sectioned beam. You'll also disable several of its modes. You'll disable its modes before you copy the beam so that each copy has the appropriate modes disabled.

To import the beam:

1. Import **quarter_beam.mnf** into your model using the default damping ratio. Name the flexible body **SHORT_1**.

   A 500 mm solid element beam appears.
2. To see the flexible body more clearly, color it yellow. To color it, select the flexible body, and then, from the **Entity Color** tool stack in the Main toolbox, select the color yellow.
3. Display the Flexible Body Modify dialog box.
4. Select **Modal ICs**.

   The Modify Modal ICs dialog box appears as shown below.
5. Disable all modes **except** 8, 11, 14, 15, 19, 23, 27, 29, 30, and 32.
6. Select **Close**.

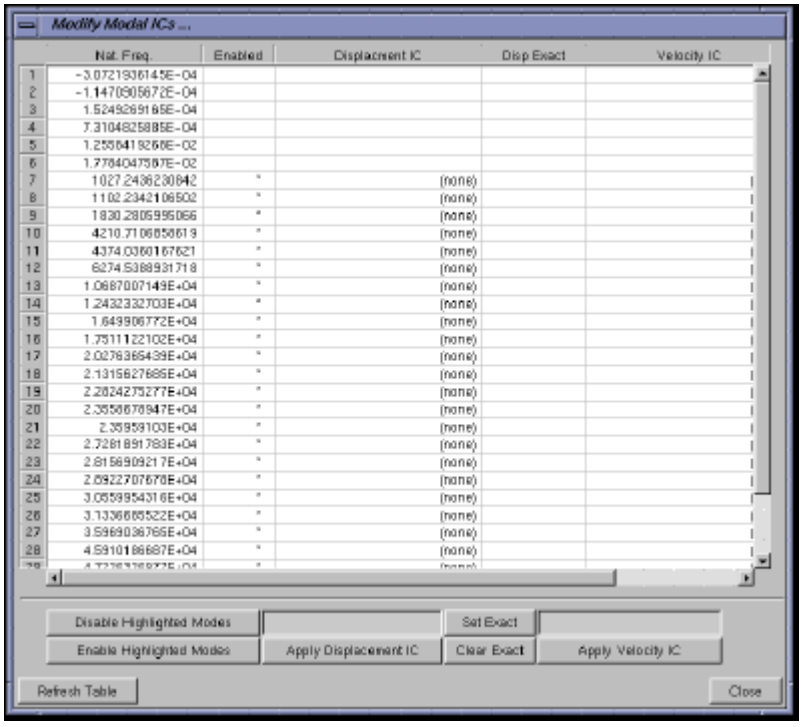


Figure 3 Modify Modal ICs Dialog Box

Assembling a Sectioned Beam

Now you'll copy the beam three times, adding each copy to the end of the next. Finally, to assemble a sectioned beam, you'll attach the beams to each other using fixed joints.

1. Select the **SHORT_1** geometry.
2. From the Main Menu → Object Manipulation Strip, select the **Point-to-Point** tool  .
3. In the Move container, select the **Selected**, **Copy**, and **From To** options.
4. Following the instructions in the status bar, select Node 1000 at the left end of the flexible body for the point to move from and Node 1001 at the right end as the point to move to.

   A second flexible body appears at the right end of the first flexible beam.
5. Repeat Steps 2 through 4 two more times until you have four flexible bodies arranged end-on-end to form a 2000 mm beam. You may want to zoom out on the view or translate the view to see all the beams.

6. To ensure that the segments are correctly copied, select the last one and select the **Info** tool  on the Status Toolbar. Verify that the list of selected modes includes only 8, 11, 14, 15, 19, 23, 27, 29, 30, and 32.
7. Create three fixed joints using the **1 Location** construction method and attach the flexible bodies to each other at Nodes 1000 and 1001 as shown in Figure 3 on page 6.

Tip:	It is easier to see the nodes on the flexible bodies if you set each beam to a different color.

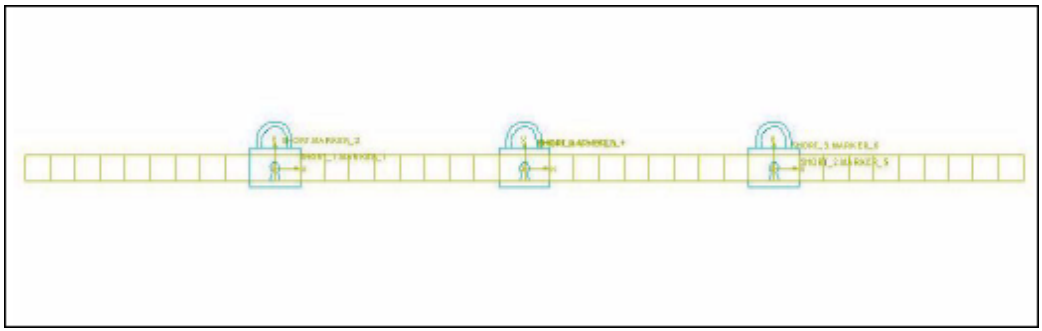


Figure 4 Four Flexible Bodies Arranged End-on-End

Building a Single Flexible Body Beam

Now, you'll integrate a single flexible beam, called LONG, into your model and disable some of its modes.

To integrate the beam into Adams View:

1. Integrate the file full_beam.mnf and rename it LONG_1 using the default damping ratio. The new beam should be superimposed over the sectioned beam. You can use the **Select List** command on the **Edit** menu to select LONG_1 if you cannot select it in the window.
2. Color the geometry **LONG_1** red so that it contrasts well with the sectioned beam.
3. Disable all modes on the geometry LONG_1 **except** 8, 10, 12, 15, 18, 20, 22, 24, 28, 29, and 31.

Adding a Spinning Shaft

In this section, you'll create a spinning shaft from cylinder geometry and create a motion generator on the shaft.

To create a spinning shaft:

1. Click the **Bodies** tab. From the **Solids** container, select the **Cylinder** tool  . Create a cylinder approximately 500 mm long, 50 mm in diameter, and aligned with the global y-axis. Position it flush with the left end of the two beams as shown in the figure below.

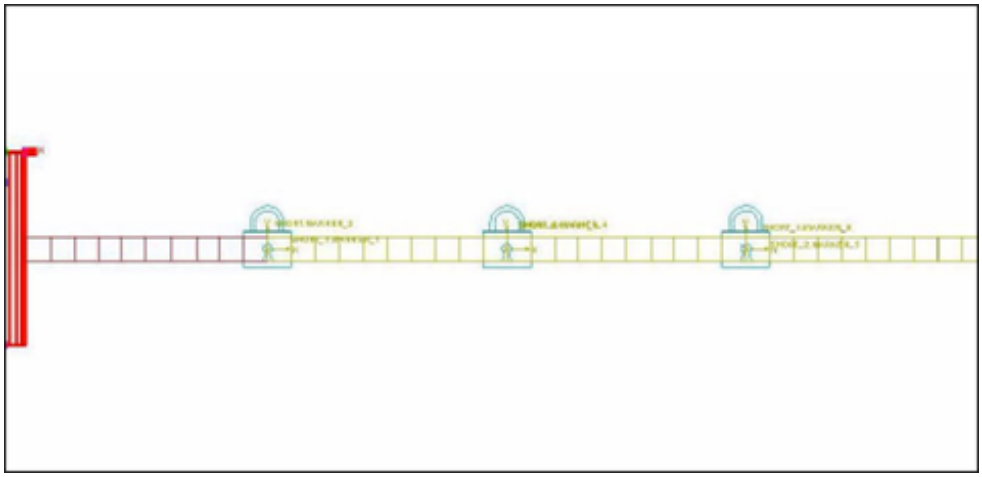


Figure 5 Spinning Shaft

2. Fasten the two beams to the cylinder using two fixed joints. Use the **2 Bod - 1 Loc** construction method.
3. View your model from the top by pressing T.
4. Click the **Connectors** tab. Select **Joints** container and select the **Revolute** tool  . Create a revolute joint between the cylinder center of mass and ground. Before creating the revolute joint, turn off the grid to ensure that the revolute joint aligns normal to the view.
5. Right-click the revolute joint. From the pop-up menu that appears, point to the joint corresponding to the revolute joint, and then select **Modify**.
6. In the Modify Joint dialog box, select **Impose Motion(s)**.
7. In the Impose Motion dialog box, from the pull-down menu next to **Rot Z**, select **acce(time)**.
8. In the text box that appears directly to the right of the pull-down menu, enter:
   6/15*(1-COS(2*PI*TIME/15))
9. Be sure that the displacement and velocity initial conditions (**Disp. IC** and **Velo. IC**) are zero (the default).
10. Select **OK**.

11. Select **OK**.

Simulating the Beams

In this last step, you'll simulate the two beams to see how they respond.

To simulate the beams:

1. Center the cylinder in the view and zoom out to allow the beam to stay in the view as it spins. You may also want to switch to a top view so that you can see it spin better.
2. Simulate for **15** seconds with **1000** output steps.

   This simulation will take approximately 15 minutes, depending on your hardware. Since the movement of the beam starts gradually, you will not notice any motion in the first few seconds of the simulation. Also note that the simulation fails at approximately 10 seconds due to divergence of the single, continuous beam.

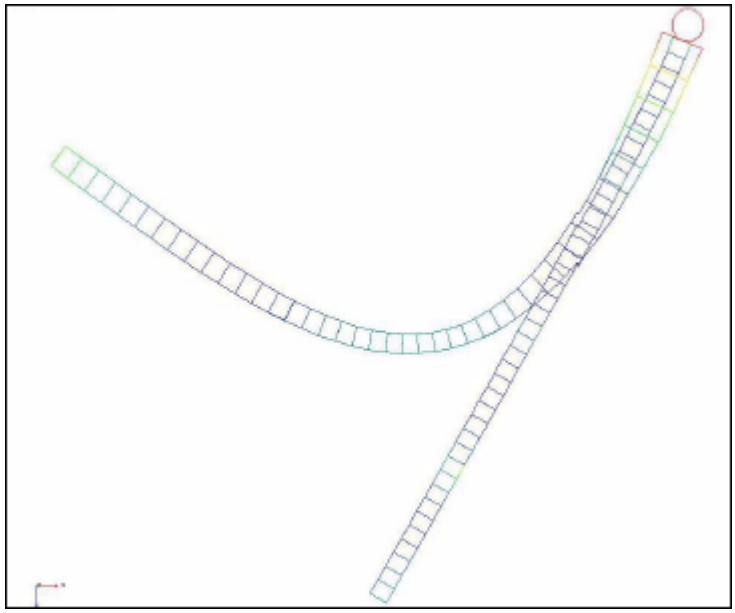


Figure 6 Results of Simulation

Animating the Results from a Top View

Now, to see the cantilever beam spin, you'll run an animation of the results from a top view.

To animate from top view:

1. In the main window, press **T**.
2. Click **Animation** tool. 
3. Select the **Play** tool. 

You will notice that the single, long beam deforms unrealistically because Adams assumes linear deformations. If you want to simulate nonlinear behavior, you need to break a flexible body into segments that behave linearly as you have done here with the sectioned beam.

Example of Working with a Preloaded Flexible Body

Overview

In this example, you'll explore how the behavior of flexible bodies changes when they correspond to a deformed configuration. The example does not address how you would create such a flexible body description because this varies depending on the FEA program with which you are working.

This example illustrates the behavior of preloaded flexible bodies using two links shown in Figure 7. Both links correspond to the same physical component, but one has been stretched by 1E+7N axial force. During the simulations, you'll discover how the behavior of the stretched link is different from the undeformed link. In addition, you'll discover how the behavior of the link changes when it is constrained.

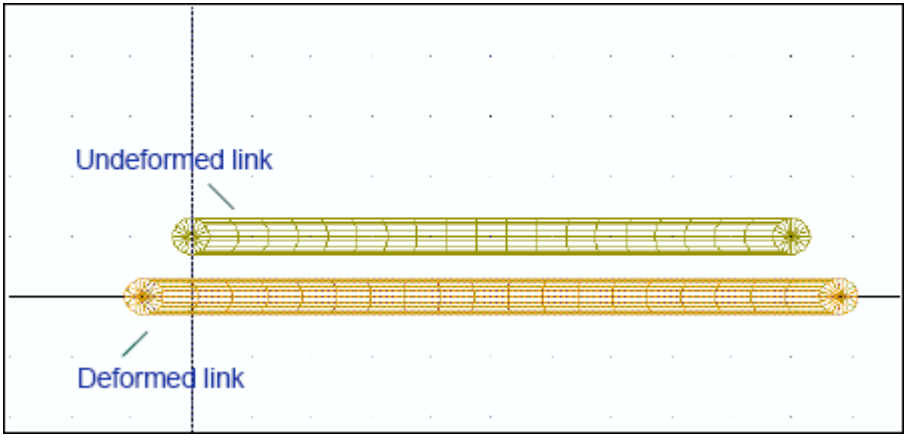


Figure 7 Undeformed and Deformed Links

Starting Adams View

You'll start the example by running Adams View and creating a new model.

To start Adams View and create a database for the flexible body:

1. Copy the file **link.mnf** and **deflink.mnf** to your local directory. This file is located in *install_dir*/flex/examples/mnf, where *install_dir* is the directory where the Adams software is installed.
2. Start Adams View and create a new model, named links, that has no gravity. Use the default units.

Integrating the Links

Now you'll integrate the MNFs containing the links.

To import the links:

1. Import **link.mnf** into your model using the default damping ratio. Name the flexible body **UNDEFORMED**.

   A link appears.
2. Turn on the working grid.
3. Move **UNDEFORMED** up by one grid space.
4. Import **deflink.mnf** into your model using the default damping ratio. Name the flexible body **DEFORMED**.

   You've now created two flexible bodies. Both bodies correspond to the same physical component, but one has been stretched by a 1E-7N axial force.

Simulating the Preloaded Flexible Bodies

You'll now perform a static equilibrium simulation on the two bodies. A static simulation attempts to find a configuration for the bodies for which all the forces balance.

To perform a static equilibrium simulation:

1. Click the **Simulation** tab. From the **Simulate** container, select Simulate  .
2. Select the **Static Equilibrium** tool.  .

   As you would expect, the stretched flexible body recoils to its undeformed configuration because the load that caused the body to be deformed is no longer present.

   You also see color contours on DEFORMED. They show how the body deforms relative to its input configuration.

Adding Joints to Deformed Body

You'll now add two revolute joints to DEFORMED. During a simulation, the load built into DEFORMED is transferred to these joints.

To add joints:

1. Return the model to its input configuration.
2. Add revolute joints to nodes 1000 and 1001 of DEFORMED. Node 1000 is at the left end of the flexible body and Node 1001 is at the right end.

Tip:	If you have trouble selecting the nodes, try zooming in on the model by pressing the letter **w** and drawing a zoom window. Then right-click to display a list of nodes near your cursor.

Performing Simulation and Investigating Results

You'll now perform a dynamic simulation and investigate the results of the simulation in Adams PostProcessor to see that the 1E-7N load that was built into DEFORMED has been transferred to the joints you created.

To perform a simulation and investigate results:

1. Perform a dynamic simulation for 1 second with 50 steps.
2. From the **Review** menu, select **PostProcessing**.

   Adams PostProcessor appears.
3. From the **Filter** list, select **constraint**.
4. From the **Object** list, select both joints.
5. From the **Characteristic** list, select **Element_Force**.
6. From the **Component** list, select X, and then select **Add Curves**.

   Note that although the color contours indicate a deformation, they are displaying numerical noise.

Example of Using Adams Linear to Verify Flexible Bodies

Overview

The following example uses a model of a slender, 2-meter long beam that was modeled in an FEA product with attachment points at the two ends and exported to Adams View. A separate finite element analysis for this beam, with one end fixed and the other end pinned, yielded the following natural frequencies for the first 3 bending modes in the XY plane:

1. 0.938486 Hz

2. 3.028538 Hz
3. 6.278326 Hz

Figure 8 shows a simplified view of the beam.

Figure 8 Beam with Fixed-Pinned Boundary Conditions

In this example, you'll import the beam into Adams View. You'll then fix the beam at one end and pin it to the other and verify the flexible body to ensure that its results in Adams Linear match the three bending modes listed above.

The sections of the tutorial are:

- Starting Adams View and Importing the MNF
- Familiarizing Yourself with the Beam
- Fixing the Beam
- Running a Simulation
- Running Adams Linear

Starting Adams View and Importing the MNF

You'll start the tutorial by running Adams View and creating a new model. You'll then integrate the MNF file into the new model.

To start Adams View and create a database for the flexible body:

1. Copy the file **full_beam.mnf** to your local directory. This file is located in *install_dir*/flex/examples/mnf, where *install_dir* is the directory where the Adams software is installed.
2. Start Adams View and create a new model that has no gravity. Use the default units.
3. Import **full_beam.mnf** into your model using a damping ratio of 0.

Familiarizing Yourself with the Beam

In this section, you'll look at each of the modes in the beam to familiarize yourself with the modal content of the flexible beam.

To familiarize yourself with the beam modes:

1. Double-click the flexible body in Adams View to display the Flexible Body Modify dialog box.

2. From the Flexible Body Modify dialog box, select the **Next Mode** tool  to view the next mode.
3. Continue selecting the **Next Mode** tool  until you've viewed most of the modes.
4. Close the Flexible Body Modify dialog box by selecting **Cancel**.

Fixing the Beam

Now you'll fix one end of the beam to ground using a fixed joint and pin the beam on the other end using a revolute joint.

To fix the ends of the beam:

1. Click the **Connectors** tab. Select **Joints** container and select the **Fixed Joint** tool. 
2. In the settings container of the Main toolbox, select the construction method **1 Location**.
3. Place the cursor at the left end of the beam near the origin. Node numbers flash on the screen as you move the cursor. Select node 1000 for the location of the fixed joint by clicking the left mouse button when the cursor is over node 1000.
4. Click the **Connectors** tab. Select **Joints** container and select the **Revolute** tool 
5. In the settings container of the Main toolbox, select the construction method **1 Location**.
6. Place the cursor at the right end of the beam. Select node 1001 for the location of the revolute joint by clicking the left mouse button when the cursor is over node 1001.

Running a Simulation

After fixing the beam, you can run a static simulation of the beam to establish an operating point for the linearization.

To run a simulation:

1. Click the **Simulation** tab. From the **Simulate** container, select Simulate tool  .
2. Click the **Simulation Settings** from the bottom of the Simulation Control dbox. Check the **More** box.
3. The **Solver Settings** dialog box appears.

In the Simulation Control dialog box, select the **Static Equilibrium** tool  to perform a single static simulation at time zero. An alert box appears informing you that the equilibrium operation was successful.

Running Adams Linear

Now that you've found the static equilibrium of the beam, you can use Adams Linear to view the linear mode frequencies that Adams Linear has computed.

To run Adams Linear:

1. Click the **Simulation** tab. From the **Simulate** container, select Simulate tool  .
2. Select the **Linear** tool. 

   A message appears, informing you that the linear modes are done and asking you if you want to review the results now.
3. Click **Animate**.

   The **Linear Modes Controls** dialog box appears.
4. Using the Linear Modes Controls tool, display mode 1.
5. In the **Max Translation** text box, enter a value to scale the mode.

   Note that Adams Linear has accurately computed a natural frequency of 0.9385. Note also that the corresponding mode shape is unlike any of the component mode shapes for this flexible body. Adams Linear has synthesized this system mode using a combination of component modes.
6. Verify that the other modes in the XY plane, modes 3 and 5, have acceptable results.
7. Close the Linear Modes Controls dialog box and display the Flexible Body Modify dialog box.
8. Modify the flexible body by disabling mode 9.
9. Recompute the static equilibrium and the linear solution and note how the system mode solution deteriorated when this important mode shape was disabled.

If either the attachment points or units were incorrectly specified when generating the MNF, your Adams Flex body would fail this simple validation test. For more on specifying attachment points and units, see FE Model Requirements in the Adams Flex online help (under the **Translate** tab).