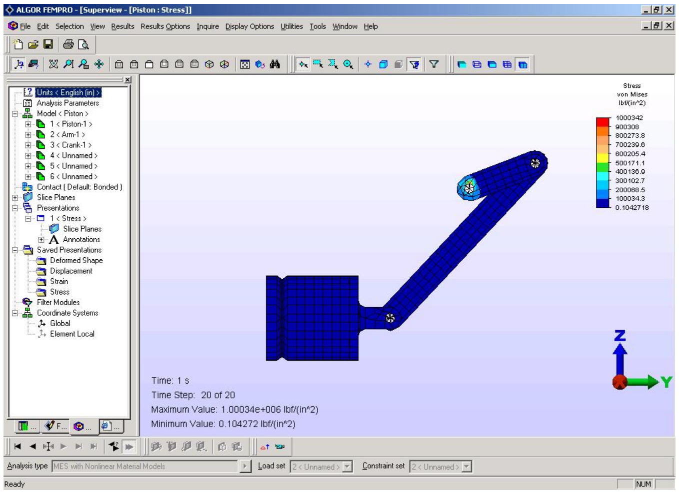
ALGOR®

Mechanical Event Simulation with Linear Material Models Tutorial



3-D Piston Assembly Model

Part Number 6700.525

Revision 1.10

August 2003

August 22, 2003

ALGOR, Inc.
150 Beta Drive
Pittsburgh, PA 15238-2932 USA
Phone: 1.412.967.2700
USA/Canada: 1.800.48.ALGOR
Fax: 1.412.967.2781
Product/Services e-mail:
info@algor.com
Technical Support e-mail:
service@algor.com
Internet Address:
www.ALGOR.com

Copyright © 2003 ALGOR, Inc.

All rights reserved. This publication may not be reproduced in any form, by any method, for any purpose, either in part or in its entirety, without the expressed written permission of ALGOR, Inc.

This publication describes the state of ALGOR software at the time of its printing and may not reflect the software at all times in the future. This publication may be changed without notice. This publication is not designed to transmit any engineering knowledge relating specifically to any company or individual engineering project. In providing this publication, ALGOR does not assume the role of engineering consultant to any user of this publication and hereby disclaims any and all responsibility for any errors or omissions arising out of any engineering activity in which this publication may be utilized.

This document has been designed to be printed on the customer's local computer and printer. ALGOR cannot be held responsible for any errors incurred in the printing of this document.

Files List for this Tutorial

In addition to this document file, the following files are referenced in the tutorial:

***Piston*.ach**
***MESresults*.ach**

These files are available in the `Tutorials\Models` subdirectory of the installation directory.

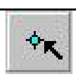
Tutorial Conventions

To make this tutorial easy to use, the following conventions will be employed. For the command conventions, the item (or an example of one) that you need to perform is noted in bold on the left. To the right of the item is a short description of the action and/or results of the action.

User Input Notation Conventions

`algframe`	Type **"algframe"** using the keyboard. Text that you need to type is noted in bold type using a Courier font.
<Esc>	Press the <Esc> key. Some of the other keys expressed in this manner are <Enter>, <Tab> and the function keys, for example <F9>.
<Ctrl>-c	Press <Ctrl> and the letter **"c"** simultaneously. Keys to be pressed at the same time are shown with a hyphen between them.
"Enclose"	Select the **"Enclose"** command. The names of pop-up menus, options and buttons are bold-faced, enclosed in quotation marks and shown as they are on the screen.
"Selection: Shape: Point"	Access the SELECTION pull-down menu and select the **"Shape"** pull-out menu. Select the **"Point"** command. Commands in sequences are separated by colons.
Mouse	Use the mouse to click on the specified location. FEMPRO is designed for a two-button mouse. Where "click" or "left-click" is used, you should press the left mouse button. "Right-click" means you should press the right mouse button. If you have a three-button mouse, you will not use your middle button for ALGOR software.

In the tables throughout this tutorial, input instructions for using toolbars and pull-down menus are in the two left columns. Descriptions or more detailed instructions are given in the right column. For example:

	**"Selection: Shape: Point"**	Access the SELECTION pull-down menu and select the **"Shape"** pull-out menu. Select the **"Point"** command to enter point selection mode.

Other Notation Conventions

sd3.dmit, an .esx file	Filenames and file extensions are lowercase with the filename in italic.
filename.doc	Filenames that are user-supplied are in bold, lowercase italics.
`\model` directory	Directory names will appear in Courier type and be followed by the term "directory". (The directory where the ALGOR software is stored is usually referred to as the installation directory).
FILE pull-down menu	Pull-down menu names are shown in uppercase characters.

3-D Piston Assembly Model

In this tutorial, we will introduce you to ALGOR software's capabilities for Mechanical Event Simulation (MES) with Linear Material Models. The example demonstrated in this tutorial shows how to set up and analyze a three-dimensional (3-D) model of a piston assembly.

> **Note:** The MES/NLM or Multiphysics software can also be used for this tutorial. However, if one of these other software packages is used, some of the keystrokes, text and figures in this tutorial will differ from what you see on your computer screen.

You will perform the following steps:

I. Setting up the Model – Retrieve a supplied model archive file, which contains a solid model of a piston assembly; add joints; create a mesh; specify data needed for MES/LM analysis including the analysis type, element type, element definition, material properties, boundary conditions, prescribed rotation and analysis parameters; check the model using the Superview IV Results environment.

II. Analyzing the Model – Analyze the model using the MES/LM processor.

> **Note:** Due to the extent of the calculations required for this mechanical event simulation, the actual analysis might take several hours depending on your computer hardware. For your convenience, the complete finite element model and MES/LM analysis results are available in a supplied model archive file named *MESresults*.ach. You can use the supplied archive files in the "Reviewing the Results" section without analyzing the model.

III. Reviewing the Results – Examine the stress results graphically with the Superview IV Results environment.

I. Preprocessing

In this phase, you will retrieve a supplied model archive file, named *Piston*.ach, which contains a solid model of a piston assembly. You will add joints and create a mesh. You will specify all data needed for a MES/LM analysis including the analysis type, element type, element definition, material properties, boundary conditions, prescribed rotation and global analysis parameters. Then, you will check the model geometry and finite element data using Superview to verify that the model is ready for analysis.

1. Problem Description

A piston assembly consists of a piston, an arm and a crank. Pin joints connect the piston to the arm, the arm to the crank and the crank to a mechanism that imparts rotation. The piston assembly is made of Aluminum (6061-T6).

Boundary conditions are applied to:

- the skirt of the piston to constrain it except for translation in the Y direction
- surfaces of the crank to constrain them from translation in the X direction, and rotation about X and Y axes

Figure 1 shows a diagram of the piston assembly. The goal of the MES/LM analysis is to determine motion and stresses in the piston assembly due to the rotation of the crank.

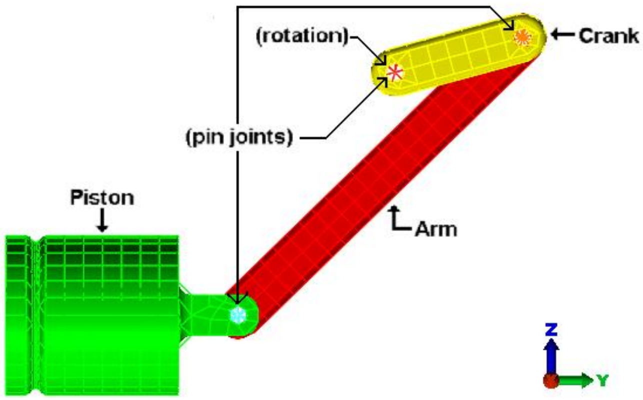


Figure 1: Diagram of the Piston Assembly

2. Retrieving the Supplied Model Archive File

In this section, you will use FEMPRO to retrieve the supplied model archive file, *Piston*.ach.

Starting FEMPRO

Start FEMPRO from the Windows taskbar.

	"Start: Programs: ALGOR V14: FEMPRO"	In the Windows taskbar, press the **"Start"** button. Select the **"Programs"** pull-out menu and select the **"ALGOR V14"** pull-out menu. Select the **"FEMPRO"** command.

FEMPRO will now appear with the **"Open"** screen active.

	"Cancel"	Press the **"Cancel"** button to close the **"Open"** screen.

In FEMPRO, you have a variety of tasks available to you. You can start a new model, choose an existing model and perform any complete engineering analysis. Help information is available by accessing the HELP pull-down menu and selecting the **"Contents"** command. This will access the *ALGOR User's Guide.*

Retrieving the Supplied Model Archive File

Retrieve the supplied model archive file from the `Tutorials\Models` directory. This archive file contains the model of the piston assembly.

	"File: Archive: Retrieve…"	Access the FILE pull-down menu and select the **"Archive"** pull-out menu. Select the **"Retrieve…"** command (see Figure 2). The **"Extract Archive"** screen will appear.

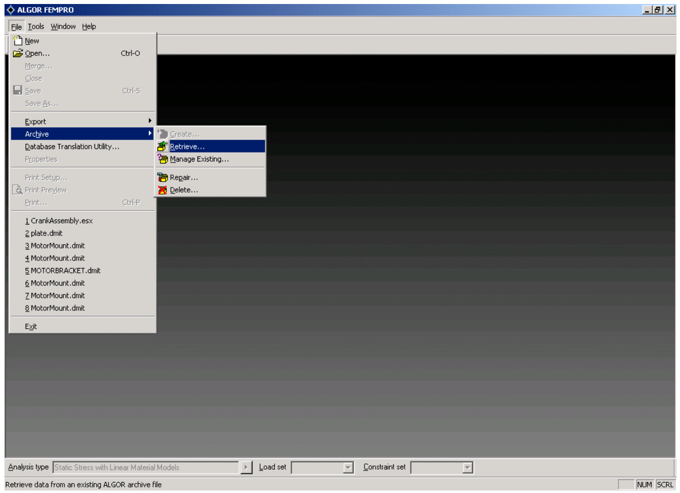


Figure 2: Retrieving the Archive File

	Mouse	Use the **"Look in:"** drop-down box to navigate to the `Tutorials\Models` directory, which is where the archive file is stored.
	Piston	Click on the *Piston*.ach file.
	"Open"	Press the **"Open"** button.
	Mouse	Specify the archive restore location in the **"Browse for Folder"** screen.
	"OK"	Press the **"OK"** button to accept the location.
	"MES with Linear Material Models"	Select the **"MES with Linear Material Models"** option in the **"Single analysis:"** drop-down box.
	"OK"	Press the **"OK"** button. The model will be loaded in the CAD Solid Model environment.

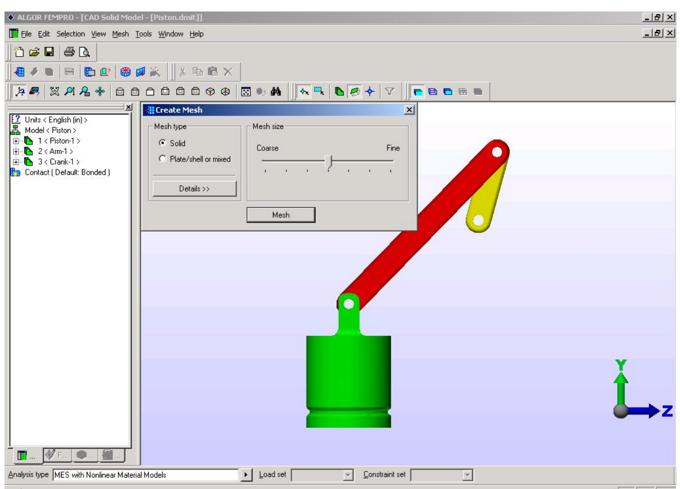


Figure 3: Meshed Model in the CAD Solid Model Environment

3. Meshing the Model in the CAD Solid Model Environment

Create a surface mesh and a solid mesh for the model.

	Mouse	In the **"Create Mesh"** screen, click and drag the slider to the left to specify a surface mesh size of **150%** (see Figure 4).

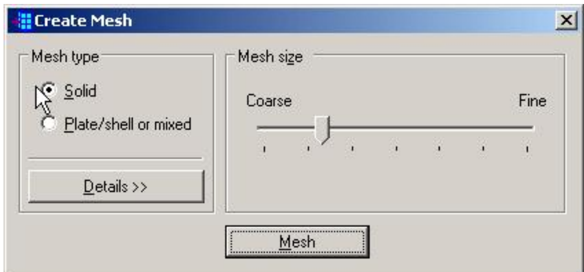


Figure 4: Specifying the Surface Mesh Size

	"Mesh"	Press the **"Mesh"** button. A surface mesh will be created. After meshing is completed, the model with the mesh will be displayed (see Figure 5).
	Mouse	Press the **"X"** to close the **"Create Mesh"** screen.

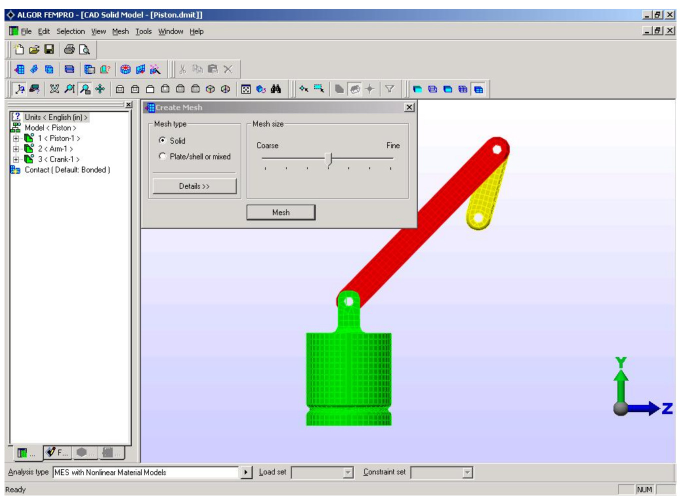


Figure 5: Completed Surface Mesh

4. Creating Joints in the CAD Solid Model Environment

Create joints at the piston and the arm, the arm and the crank, and the end of the crank. Use the tree view to select surfaces for creating joints. (Alternatively, surfaces can be selected by clicking on the model display in the working area.)

First, select the surfaces for creating a joint at the piston and the arm. From examining the surfaces of the model, we know that the surfaces we want to use are surfaces 1, 2, 32, and 35 of Part 1 and surfaces 4 and 12 of Part 2. You can find the surface numbers by hovering the mouse over the surfaces and looking at the status bar.

	**"Selection: Select: Surfaces"**	Access the SELECTION pull-down menu and select the **"Select"** pull-out menu. Select the **"Surfaces"** command.
	**"View: Zoom Area"**	Access the VIEW pull-down menu and select the **"Zoom Area"** command.
	Mouse	Using the mouse, draw a box around the joint with the piston (see Figure 6).
	Mouse	Click on the **"Surface 1"** heading in the tree view for Part 1. The selected surface will be highlighted in the display area.
	<Ctrl>- Mouse	Holding down the <Ctrl> key, click on the **"Surface 2"** heading for Part 1.
	<Ctrl>- Mouse	Holding down the <Ctrl> key, click on the **"Surface 32"** heading for Part 1.
	<Ctrl>- Mouse	Holding down the <Ctrl> key, click on the **"Surface 35"** heading for Part 1.
	<Ctrl>- Mouse	Holding down the <Ctrl> key, click on the **"Surface 4"** heading for Part 2.
	<Ctrl>- Mouse	Holding down the <Ctrl> key, click on the **"Surface 12"** heading for Part 2.
	**"View: Rotate"**	Access the VIEW pull-down menu and select the **"Rotate"** command.
	Mouse	Click and drag the mouse to rotate the model to verify that all 6 of the surfaces of the holes are selected (see Figure 6).

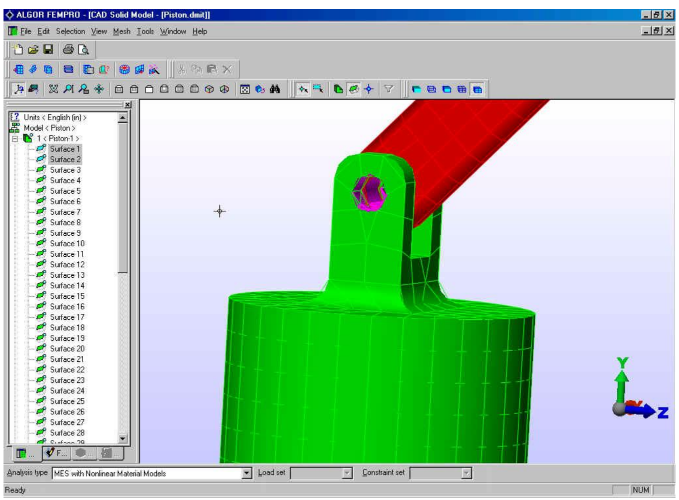


Figure 6: Joint Surfaces Selected and Highlighted

Create a joint for the selected surfaces.

	Mouse	Right-click on the heading for one of the selected surfaces.
	"Create Joint"	Select the **"Create Joint"** command. The **"Create Joint"** screen will appear (see Figure 7).

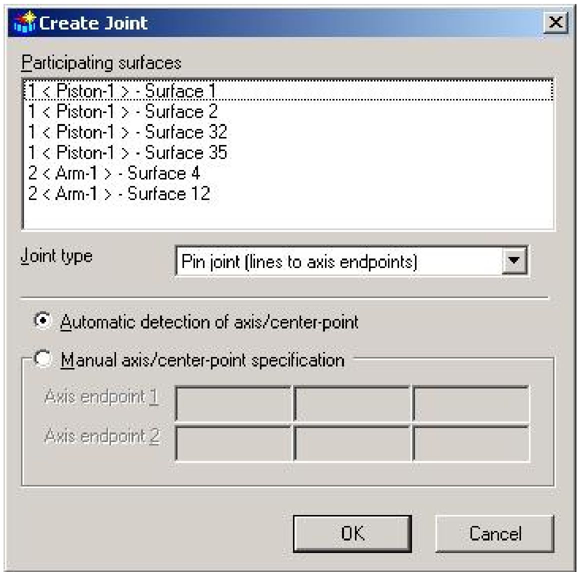


Figure 7: The Create Joint Screen

	"OK"	Press the **"OK"** button to accept the specified data and create a pin joint at the selected surfaces. The tree view will be updated with an additional group. Also, the joint will be shown in the display area.

Next, select the surfaces for creating a joint at the arm (Part 2) and the crank (Part 3). From examining the surfaces of the model, we know that the surfaces we want to use are surfaces 3 and 13 of Part 2 and surfaces 4 and 12 of Part 3.

	Mouse	Click on the **"Surface 3"** heading for Part 2.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the **"Surface 13"** heading for Part 2.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the **"Surface 4"** heading for Part 3.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the **"Surface 12"** heading for Part 3.

Create a joint for the selected surfaces.

	Mouse	Right-click on the heading for one of the selected surfaces.
	"Create Joint"	Select the **"Create Joint"** command.
	"OK"	Press the **"OK"** button.

Select the surfaces for creating a joint at the end of the crank (Part 3). By examining the surfaces of the model, we can determine that the surfaces we want to use are surfaces 3 and 13 in Part 3.

	Mouse	Click on the **"Surface 3"** heading for Part 3.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the **"Surface 13"** heading for Part 3.

Create a joint for the selected surfaces.

	Mouse	Right-click on the heading for one of the selected surfaces.
	"Create Joint"	Select the **"Create Joint"** command.
	"OK"	Press the **"OK"** button.
	**"View: Orientation: YZ Right"**	Access the VIEW pull-down menu and select the **"Orientation"** pull-out menu. Select the **"YZ Right"** command (see Figure 8).

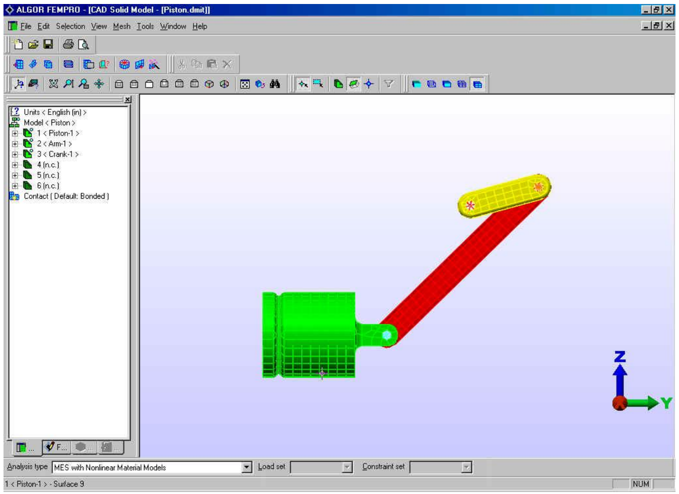


Figure 8: Model with Mesh and Joints

5. Specifying Model Data in the FEA Editor Environment

In this section, you will specify all data needed for an MES/LM analysis including the analysis type, element type, element definition, material properties, boundary conditions, prescribed rotation and analysis parameters.

Transferring the Model to the FEA Editor Environment

The FEA Editor environment is where Mechanical Event Simulation analysis parameters are defined and where the analysis will be performed. Once the model is in the FEA Editor environment, the parameters that need to be specified will be marked with a red "X". By following these fields which have the red "X" overlay in the tree view, the analysis can be set up.

	"Tools: FEA Editor"	Access the TOOLS pull-down menu and select the **"FEA Editor"** command.

After the mesh is loaded, the model will be displayed in the FEA Editor environment.

Element Definition

By default, the solid meshing engine has created brick elements for the crank, arm, and piston parts. When transferring from the CAD Solid Model environment to the FEA Editor environment, the element definition is set for brick elements. The options used by default are acceptable for this tutorial. Therefore, it is not necessary to define element data for parts 1, 2 and 3. For more information about setting meshing parameters and element data information, see the *ALGOR User's Guide*.

The pin joints that you created in the CAD Solid Model environment were defined as truss elements by default. Define or change element data for the pin joint parts (Parts 4, 5 and 6).

First, change the element definition for Parts 4 and 5, the pin joints on each end of the arm.

	Mouse	Click on the **"Element Definition"** heading for Part 4 in the tree view.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, right click on the **"Element Definition"** heading for Part 5.
	"Modify Element Definition…"	Select the **"Modify Element Definition…"** command
	Mouse	Highlight the **"Cross-Sectional Area"** field. Notice that there is a red dot highlighting the entry blank for the missing information.
	1	Type **"1"** in the **"Cross-Sectional Area"** field in the **"General Settings"** section of the **"General"** tab of the **"Element Definition"** screen (see Figure 9).

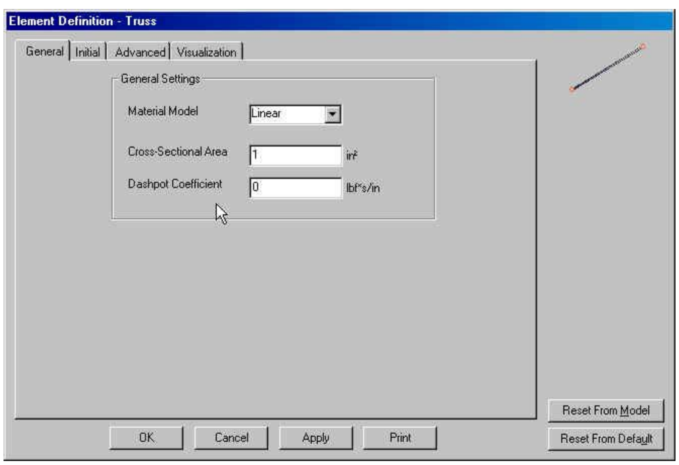


Figure 9: Element Definition for Parts 4 and 5

	"OK"	Press the **"OK"** button to accept the specified element and close the **"Element Definition"** screen.

Part 6, the pin joint at the end of the crank, is a special case. As described in the problem description, the end of the crank rotates, which will be simulated by a prescribed rotation. Hence, Part 6 will need to have both translational and rotational degrees of freedom; however, truss elements have only translational degrees of freedom. Therefore, the element of Part 6 will need to be changed to beam element.

	Mouse	Right-click on the **"Element Type <Truss>"** heading for Part 6.
	"Beam"	Select the **"Beam"** command. Note that because this part is made of linear elements, only applicable element types are available.

Define element data for the joint (Part 6) elements.

	Mouse	Right-click on the **"Element Definition"** heading for Part 6.
	"Modify Element Definition…"	Select the **"Modify Element Definition…"** command.
	"Round"	Select the **"Round"** option in the **"Section Type"** drop-down box (see Figure 10).

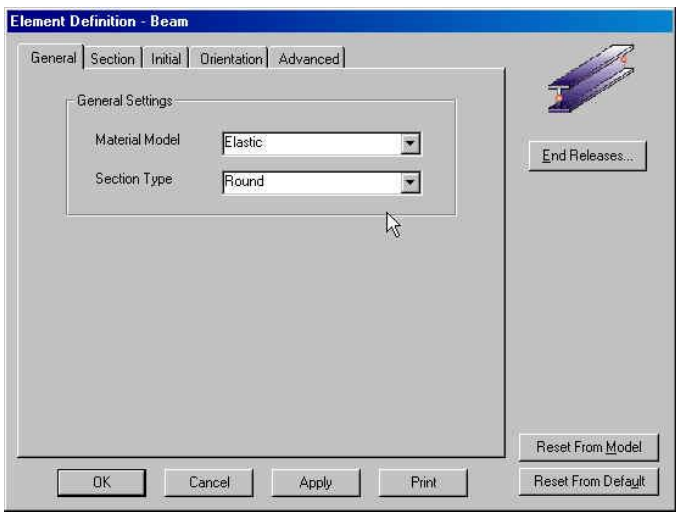


Figure 10: Specifying the Section Type for Part 6 Beam Elements

	Mouse	Click on the **"Section"** tab.
	1	Type **"1"** in the **"Diameter"** field.
	"OK"	Press the **"OK"** button to accept the specified data and close the **"Element Definition"** screen.

Defining Material Properties

Define material properties for the parts of the model, which all are made of the same material, Aluminum (6061-T6). You can define material properties for the piston, arm and crank at once because they are the same element type - bricks. Likewise, material properties can be defined at once for the two pin joint parts that are trusses. The material for the beam element pin joint must be defined separately.

First, define material properties for Parts 1, 2 and 3 (the piston, arm and crank).

	Mouse	Click on the **"Material <Unknown>"** heading for Part 1 in the tree view.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the **"Material <Unknown>"** for Part 2 to add it to the selection set.

	<Ctrl>-Mouse	Holding down the <Ctrl> key, right-click on the **"Material <Unknown>"** for Part 3.
	"Modify Material…"	Select the **"Modify Material…"** command.
	"Aluminum (6061-T6)"	Highlight the **"Aluminum (6061-T6)"** option in the **"Select Material"** section (see Figure 11).

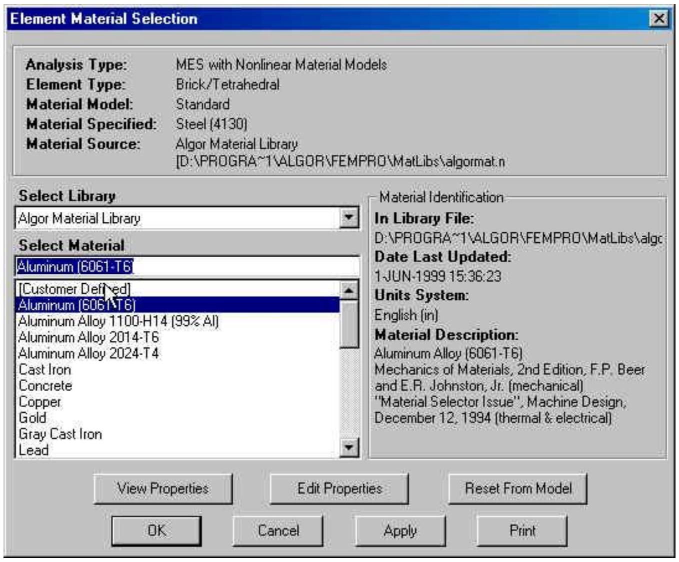


Figure 11: Specifying Material Properties for Parts 1, 2 and 3

	"OK"	Press the **"OK"** button to accept the selected material.

Define material properties for Parts 4 and 5 (the two truss element pin joints).

	Mouse	Click on the **"Material <Unknown>"** heading for Part 4 in the tree view.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, right-click on the **"Material <Unknown>"** for Part 5.
	"Modify Material…"	Select the **"Modify Material…"** command.
	"Aluminum (6061-T6)"	Highlight the **"Aluminum (6061-T6)"** option in the **"Select Material"** section.
	"OK"	Press the **"OK"** button to accept the selected material.

Define material properties for Part 6 (the beam element pin joint).

	Mouse	Right-click on the **"Material <Unknown>"** for Part 6.
	"Modify Material…"	Select the **"Modify Material…"** command.
	"Aluminum (6061-T6)"	Highlight the **"Aluminum (6061-T6)"** option in the **"Select Material"** section.
	"OK"	Press the **"OK"** button to accept the selected material.

Adding Boundary Conditions

Add boundary conditions to:

- the skirt of the piston to constrain it except for translation in the Y direction
- selected surfaces of the arm and crank to constrain them from translation in the X direction

First, add boundary conditions to the bottom surface of the piston skirt.

	**"Selection:Select:Surfaces"**	Access the SELECTION pull-down menu and select the **"Select"** pull-out menu. Select the **"Surfaces"** command. The model will be redisplayed with coloring according to surface number.
	Mouse	Click on one of the 4 surfaces that comprise the outside of the piston (see Figure 12).
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on one of the remaining 3 surfaces that comprise the outside of the piston.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on one of the remaining 2 surfaces that comprise the outside of the piston.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on the remaining surface that comprise the outside of the piston.

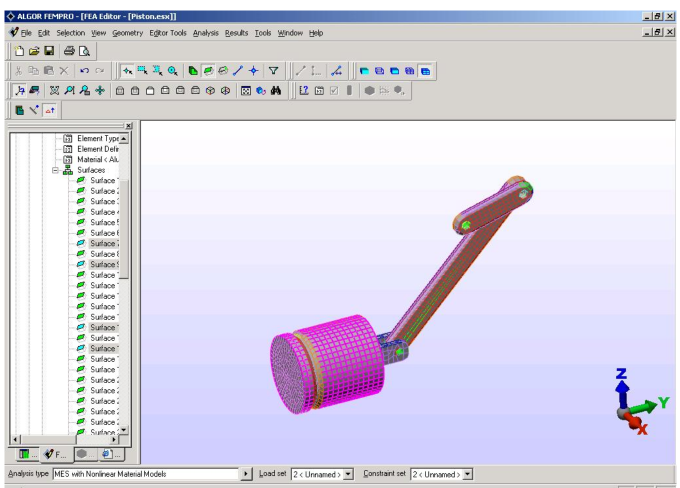


Figure 12: Selecting the Piston Surfaces for Boundary Conditions

	Mouse	Right-click in the display area.
	"Add:Surface Boundary Conditions…"	Select the **"Add"** pull-out menu and select the **"Surface Boundary Conditions…"** command.
	"Fixed"	In the **"Predefined"** section, press the **"Fixed"** button. All of the checkboxes in the **"Constrained DOFs"** section will be activated.
	"Ty "	Deactivate the **"Ty"** checkbox.
	"Ry"	Deactivate the **"Ry"** checkbox (see Figure 13).
	"OK"	Press the **"OK"** button to accept the specified surface boundary conditions for the piston skirt. The model will be redisplayed with red circles on the partially constrained nodes.

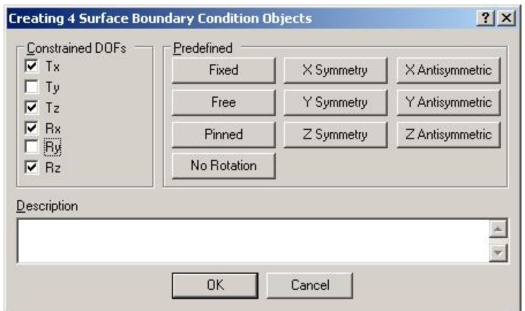
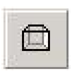


Figure 13: Specifying Surface Boundary Conditions for the Piston Skirt

Add boundary conditions to selected surfaces of the arm and crank.

	**"View:Orientation:YZ Right"**	Access the VIEW pull-down menu and select the **"Orientation"** pull-out menu. Select the **"YZ Right"** command. This orientation will make it is easier to confirm the selection of surfaces on the crank for adding boundary conditions (see Figure 14).

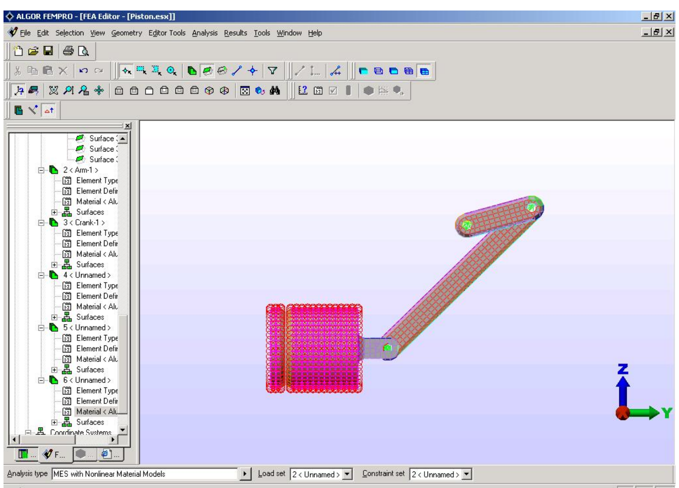


Figure 14: Preparing to Select Surfaces on the Crank

	Mouse	Click on the **"Surface 1"** heading for Part 3 in the tree view.
	<Shift>-Mouse	Holding down the <Shift> key, click on the **"Surface 21"** heading for Part 3. In the working area, the whole part should be highlighted as shown below in Figure 15. Also, the tree view will show all of the surfaces as selected.

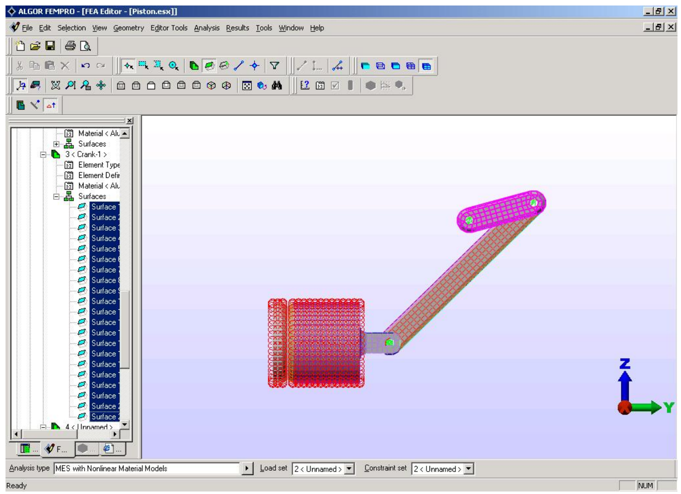


Figure 15: Adding Surface Boundary Conditions to the Crank

	Mouse	Right-click in the display area.
	"Add: Surface Boundary Conditions…"	Select the **"Add"** pull-out menu and select the **"Surface Boundary Conditions…"** command.
	"Tx"	In the **"Constrained DOFs"** section, activate the **"Tx"** checkbox.
	"Ry"	Activate the **"Ry"** checkbox.
	"Rz"	Activate the **"Rz"** checkbox (see Figure 16).
	"OK"	Press the **"OK"** button to accept the specified boundary conditions for the arm and crank. The model will be redisplayed with red circles on the partially constrained nodes.

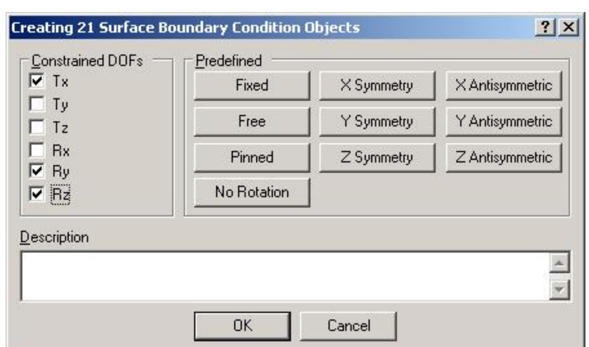
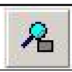


Figure 16: Boundary Conditions for Arm and Crank Surfaces

Adding a Prescribed Rotation and Boundary Condition to Joint

Add a prescribed rotation to the pin joint at the end of the crank (Part 6).

	**"View:Zoom Area"**	Access the VIEW pull-down menu and select the **"Zoom Area"** command.
	Mouse	Draw a zoom rectangle around Part 6, the joint at the end of the crank by first clicking above and to the left and then below and to the right of the joint (see Figure 17).

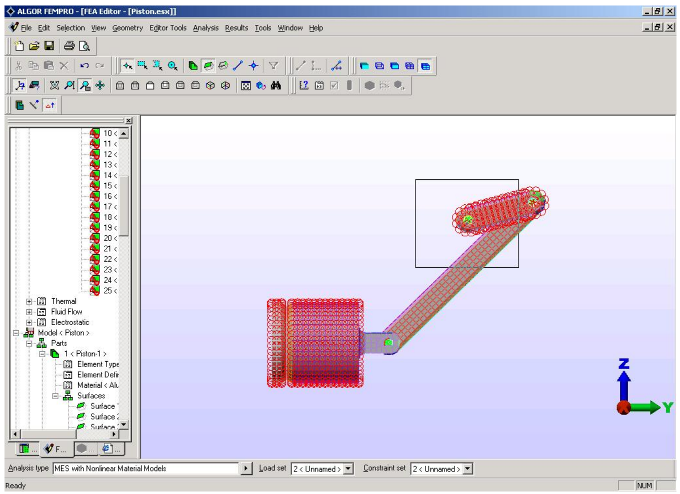
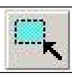


Figure 17: Drawing a Zoom Rectangle around Part 6

	**"Selection: Shape: Rectangle"**	Access the SELECTION pull-down menu and select the **"Shape"** pull-out menu. Select the **"Rectangle"** command.
	**"Selection: Select: Vertices"**	Access the SELECTION pull-down menu and select the **"Select"** pull-out menu. Select the **"Vertices"** command
	Mouse	Draw a selection rectangle around the center of the joint (see Figure 18). The selected nodes will be highlighted.

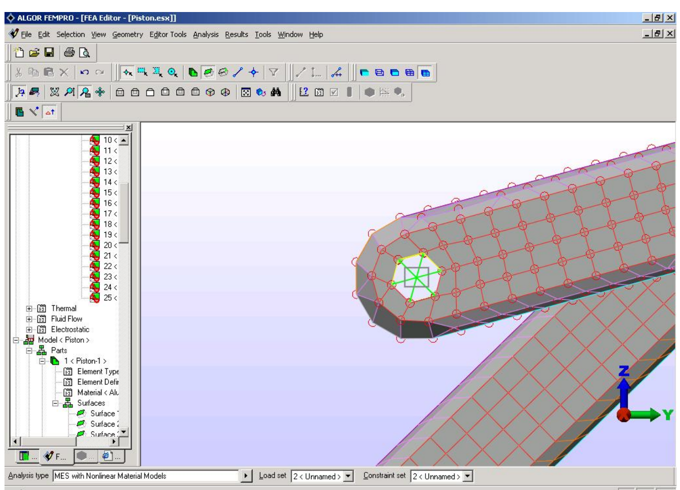


Figure 18: Drawing a Selection Rectangle around the Center of Part 6

	Mouse	Right-click in the working area.
	"Add: Nodal Prescribed Displacements…"	Select the **"Add"** pull-out menu and select the **"Nodal Prescribed Displacements…"** command.
	"Rotation"	Select the **"Rotation"** radio button in the **"Type"** section.
	1	Type **"1"** in the **"Magnitude"** field.
	"…"	Press the **"…"** button to the right of the **"Active Range"** field.
	10	Type **"10"** in the **"Death Time"** column of the **"Birth and Death Times"** table. This will ensure that the rotation of the joint will continue throughout the duration of the analysis (see Figure 19).
	"OK"	Press the **"OK"** button to accept the specified data.

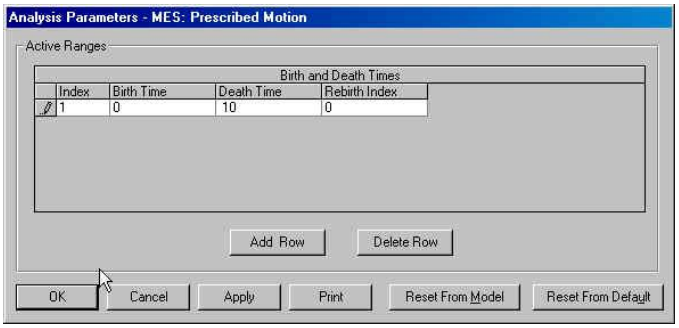


Figure 19: Specifying the Birth and Death Time for the Prescribed Rotation

	"OK"	Press the **"OK"** button. A blue, counter-clockwise, semi-circular arrow will appear on the selected nodes indicating the prescribed rotation.

Add a boundary condition to the pin joint at the end of the crank (Part 6).

	Mouse	With the two nodes still selected, right-click in the display area.
	"Add: Nodal Boundary Conditions…"	Select the **"Add"** pull-out menu and select the **"Nodal Boundary Conditions…"** command.
	"Fixed"	In the **"Predefined"** section, press the **"Fixed"** button.
	"Rx"	Deactivate the **"Rx"** checkbox. This will constrain the piston except for rotation about the X direction (see Figure 20).
	"OK"	Press the **"OK"** button to accept the specified surface boundary conditions for the piston skirt. The model will be redisplayed with red circles on the partially constrained nodes.

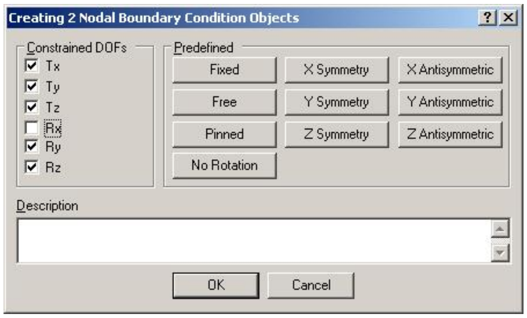


Figure 20: Boundary Conditions for Joint

Specify analysis parameters for MES with Linear Material Models.

	Mouse	Right-click on the **"Analysis Parameters"** heading in the tree view.
	"Modify Analysis Parameters…"	Select the **"Modify Analysis Parameters…"** command.
	1	Type **"1"** in the **"Duration"** field in the **"Event "** section.
	20	Type **"20"** in the **"Capture Rate"** field in the **"Event "** section.
	"Add Row"	In the **"Data for Selected Load Curve"** section, press the **"Add Row"** button. Row 2 will be added to the load curve.
	1	Type **"1"** in the second row of the **"Time"** column.
	<Tab>1	Press the <Tab> key to advance the cursor to row 2 of the **"Multiplier"** column. Type **"1"** (see Figure 21).

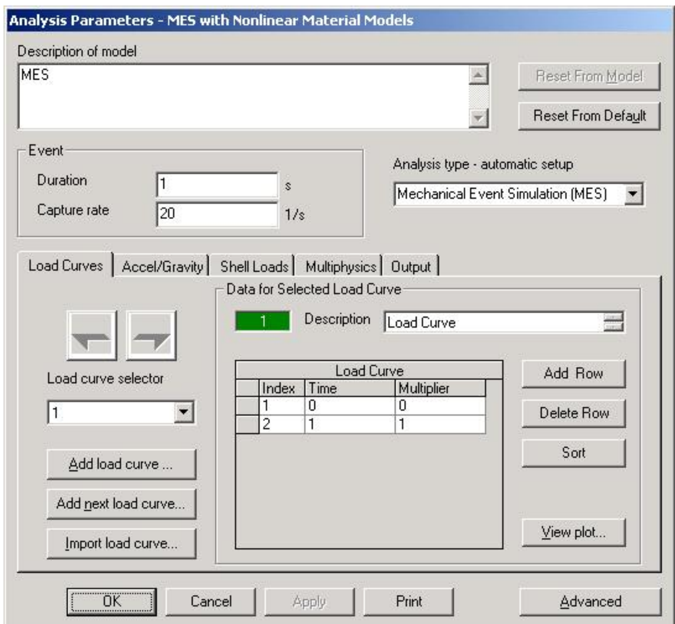


Figure 21: Specifying the Load Curve

	"OK"	Press the **"OK"** button to accept the specified data and close the **"Analysis Parameters"** screen.

6. Checking the Model

Now that all model data and analysis parameters have been specified, the model can be checked to determine whether the geometry and finite element information is valid and ready for analysis.

	**"Analysis:Check Model"**	Access the ANALYSIS pull-down menu and select the **"Check Model"** command. A **"Model Validation"** screen will appear indicating that the software is verifying the geometry and finite element data. Once the check has been completed, the model will be loaded in the Superview IV Results environment. On your own, you can use the features of the Superview IV Results environment to further check the model by examining the nodes, elements, loads and boundary conditions. (For more information about using the Superview IV Results environment to examine models, see the "In-depth Results Evaluation" or "Presentation of Results" tutorial).
	"Tools: FEA Editor"	After you are finished checking the model, access the TOOLS pull-down menu and select the **"FEA Editor"** command to return to the FEA Editor environment.

The model is now ready to be analyzed with the MES/LM processor.

II. Analyzing the Model

In the processing phase, you will analyze the piston assembly model with the MES/LM processor.

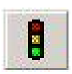
Note: MES/NLM or Multiphysics software can also be used, which include the capabilities of MES/LM.

Due to the extent of the calculations required for this mechanical event simulation, processing may take several hours depending on your computer hardware. For your convenience, the complete finite element model and analysis results are available in a supplied model archive file named *MESResults*.ach. You can use the "File:Archive:Retrieve…" command sequence to retrieve the archived model in *MESResults*.ach. Then, you can skip past the rest of this section and use the supplied results files in the next section.

If you prefer, you can analyze the model that you have built in the tutorial thus far. The choice is yours.

1. Analyzing the Model with the MES/LM Processor

Analyze the model using the MES/LM processor.

	**"Analysis:Perform Analysis…"**	Access the ANALYSIS pull-down menu and select the **"Perform Analysis…"** command. The **"MES with Linear Material Models"** screen will appear. The software will generate a solid mesh, verify the geometry and finite element data and then the analysis will begin to run automatically
	"OK"	Press the **"OK"** button to dismiss the message informing you that the current sessions of the Superview IV Results environment will be closed.

This analysis has 100 time steps, and the number of iterations necessary to achieve a convergent nonlinear solution can take a significant amount of time depending on your computer resources. (On a Pentium III, 900-MHz computer, the analysis took approximately 90 minutes.) You might want to run this analysis overnight or on another computer.

After the analysis is completed, the model will be loaded into the Superview IV Results envirnoment.

III. Reviewing the Results

In this phase, you will use the Superview IV Results environment to view the MES/LM analysis results.

1. Using the Superview IV Results environment to View Analysis Results

Previously, you used the Superview IV Results environment to check the piston assembly model. Now you will use the Superview IV Results environment to look at the results obtained from the MES/LM analysis processor. You will view stress results and create and view an analysis replay. Then, on your own, you can experiment with other Superview IV Results environment capabilities.

> **Note:** If you retrieved the supplied *MESResults*.ach file, then you can use the supplied results files for reviewing the results. In the FEA Editor environment, access the TOOLS pull-down menu and select the **"Superview"** command.

Select the Time Step to View the Stress Results

Examine the stress at the middle time step.

	**"Results Options: LoadCase: Middle"**	Access the RESULTS OPTIONS pull-down menu and select the **"Load Case"** pull-out menu. Select the **"Middle"** command.

Displaying Stress Results for Selective Parts

Choose Part 1(Skirt) to hide with during selection.

	Mouse	Click on the **"Part 1"** heading in the tree view.
	Mouse	Right-click in the display area.
	"Hide Unselected Elements"	Select the **"Hide Unselected Elements"** command.
	"View: Orientation: YZ Right"	Access the VIEW pull-down menu and select the **"Orientation"** command. Select the **"YZ Right"** command.
	**"View: Mouse Zoom"**	Access the VIEW pull-down menu and select the **"Mouse Zoom"** command.
	Mouse	Drag the mouse so that all of group one can be seen in the viewing area (see Figure 22).

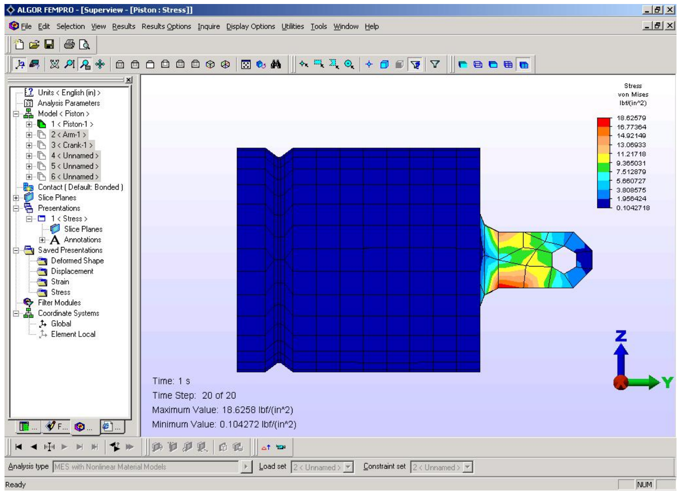


Figure 22: Piston with von Mises Stress Results

On your own, you can continue to use Superview IV Results environment capabilities to further examine the analysis results.

Congratulations! **You have completed this Mechanical Event Simulation with Linear Material Models Tutorial.**