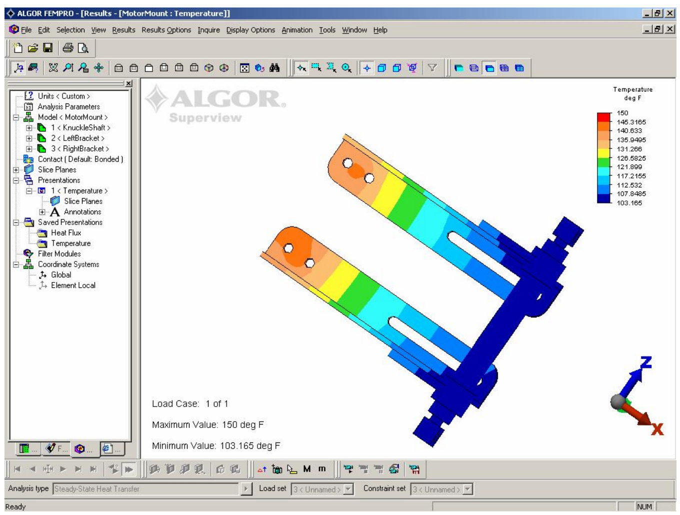
ALGOR®

Steady-State Heat Transfer Tutorial



3-D Motor Mount Assembly Model

Part Number 7000.510

Revision 16.000

September 2004

September 29, 2004

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

Files List for this Tutorial

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

***RefinedMeshMotorMount*.ach**

This file is 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>.

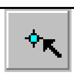
<Ctrl>-c Press <Ctrl> and the letter **"c"** simultaneously. Keys to be pressed at the same time are shown with a hyphen between them.

"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 Motor Mount Assembly Model

In this tutorial, we will introduce you to FEMPRO's FEA Editor environment and steady-state heat transfer analysis capabilities by demonstrating how to set up and analyze a three-dimensional (3-D) model of a motor mount assembly.

You will perform the following steps:

I. Setting up the Model – Retrieve a supplied model archive file, which contains a meshed model; specify data needed for analysis including the analysis type, element type, element definition, loads and analysis parameters; check the model using the Results environment.

II. Analyzing the Model – Analyze the model using the Steady-State Heat Transfer processor.

III. Reviewing the Results – Examine the temperature results graphically with the Results environment.

I. Setting up the Model

In this phase, you will retrieve a supplied model archive file, named *RefinedMeshMotorMount*.ach, which contains a meshed model. You will specify all data needed for analysis including the analysis type, element type, element definition, loads and analysis parameters. Then, you will check the model geometry and finite element data to verify that it is ready for analysis.

1. Problem Description

A motor mount assembly is designed to hold a 15-pound motor. The assembly consists of a left bracket, a right bracket and a knuckle shaft. The brackets are made of Aluminum (6061-T6) and the shaft is made of Steel (ASTM-A36). The motor is mounted to the brackets with bolts. The assembly has a tensile force due to the output of the motor.

For the steady-state heat transfer analysis, there is a convection load applied to the surfaces of the assembly that are exposed to the surrounding air. Temperature loads of 150° Fahrenheit are applied to the brackets where the motor would be in contact with them. Use the following properties for the air surrounding the brackets:

Ambient Temperature	= 50° Fahrenheit
Air Speed	= 17.6 in /sec
Mass Density	= 1.167E-7 $lbf*s^2/in/in^3$
Dynamic Viscosity	= 2.56E-9 $lbf*s/in^2$
Thermal Conductivity	= 3.50E-7 BTU/(s*in*°F)
Specific Heat	= 92.8 BTU/($lb*s^2$/in*°F)

Figure 1 shows a diagram of the motor mount assembly:

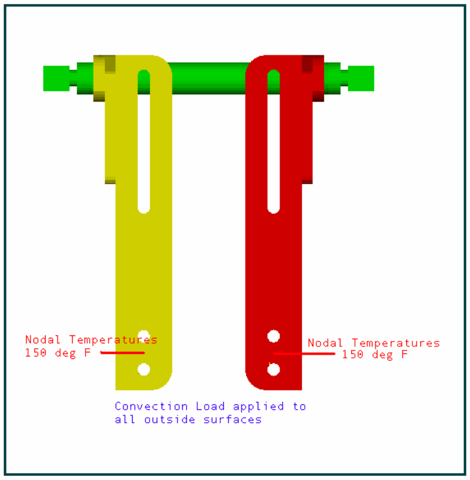


Figure 1: Diagram of the Motor Mount Assembly

2. Retrieving the Supplied Model Archive File

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

Starting FEMPRO

Start FEMPRO from the Windows taskbar.

	"Start: Programs: ALGOR V16: FEMPRO"	In the Windows taskbar, press the **"Start"** button. Select the **"Programs"** pull-out menu and select the **"ALGOR V16"** pull-out menu. Select the **"FEMPRO"** command.

FEMPRO will now appear with the **"New"** dialog active.

	"Cancel"	Press the **"Cancel"** button to close the **"New"** dialog.

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 a meshed model of the motor mount. (To learn more about meshing, see the InCAD tutorials on the motor bracket model).

	"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"** dialog will appear.

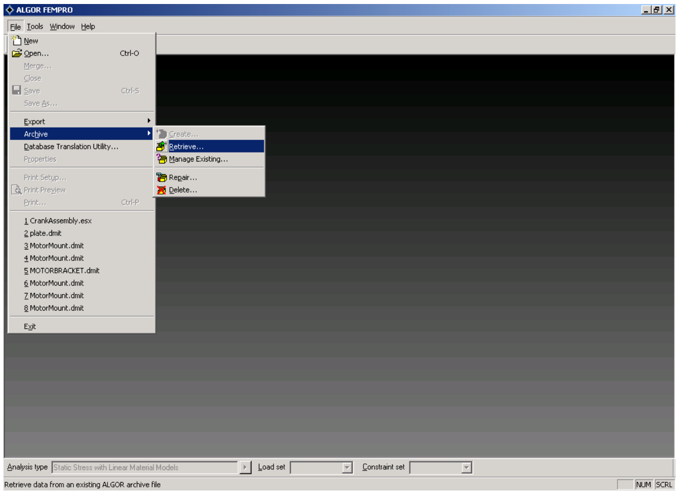


Figure 2: Retrieving the Archive File

	Mouse	Use the **"Look in:"** drop-down box to navigate to the `Tutorials\Models` directory, which is where the archive file is stored.
	RefinedMeshMotorMount	Click on the *RefinedMeshMotorMount*.ach file.
	"Open"	Press the **"Open"** button.
	Mouse	Specify the archive restore location in the **"Browse for Folder"** dialog.
	"OK"	Press the **"OK"** button to accept the location. The meshed model will be displayed in the FEA Editor environment (see Figure 3).
	"Custom"	Select the **"Custom"** option in the **"Unit System"** drop-down box.
	"British thermal unit (Btu)"	Select the **"British thermal unit (Btu)"** option in the **"Energy"** drop-down box. See Figure 4.
	"OK"	Press the **"OK"** button to accept the units definition choice

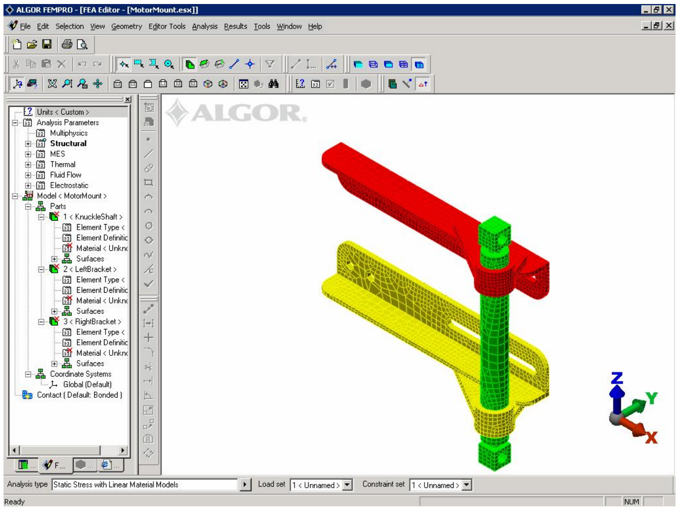


Figure 3: Meshed Model in the FEA Editor Environment

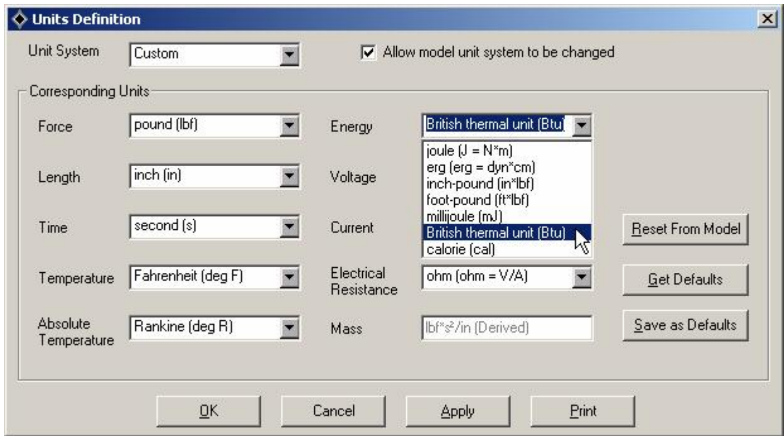


Figure 4: Select BTU as the Energy Unit

3. Specifying Model Data in the FEA Editor Environment

In this section, you will specify all data needed for analysis including the analysis type, element type, element definition, applied temperatures, a convection load and global analysis parameters.

Setting the Analysis Type

Specify the analysis type so that the appropriate loads and constraints can be applied for processing.

	"Steady-State Heat Transfer"	Select the **"Steady-State Heat Transfer"** option in the **"Analysis type"** drop-down box in the lower left corner of the screen.

Specifying Element and Material Information

By default, the mesh engine set the element type to brick elements. The default values for element type and element definition are sufficient for the motor mount assembly model.

Specify the material properties for Part 1, the knuckle shaft (the green part).

	Mouse	In the tree view, right-click on the **"Material <Unknown>"** field for Part 1.
	"Modify Material…"	Select the **"Modify Material…"** command
	"Steel (ASTM-A36)"	Highlight the **"Steel (ASTM-A36)"** item in the **"Select Material"** section. (See Figure 5.)
	"OK"	Press the **"OK"** button to accept the material property choice. The selected material will be shown in the tree view.

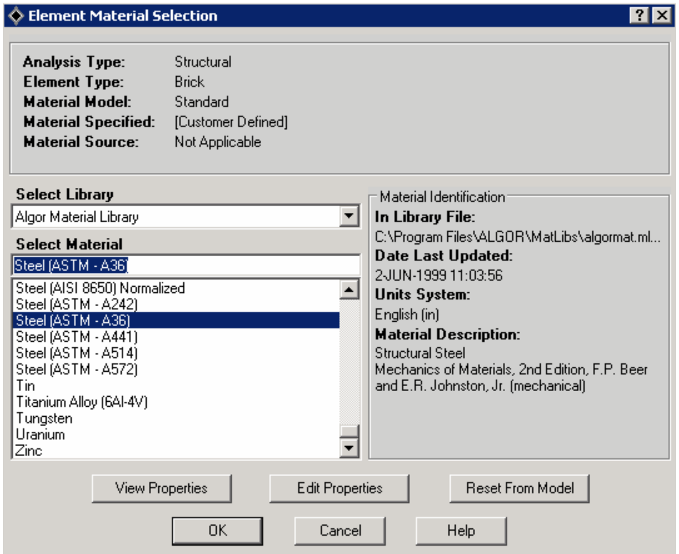


Figure 5: Specifying Material Properties for Part 1

Specify material properties for Part 2, the left bracket (red) and Part 3, the right bracket (yellow). The material properties can be specified for both parts at once because they share the same material, the same element type and the same values for the element definition.

	Mouse	Click on the **"Material <Unknown>"** heading for Part 2 in the tree view.
	<Ctrl>-Mouse	Holding down the **<Ctrl>** key, right-click on the **"Material <Unknown>"** heading for Part 3. (See Figure 6).

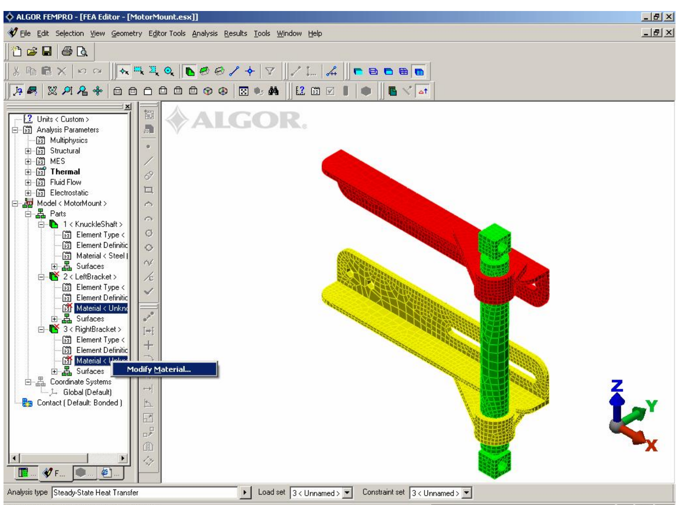


Figure 6: Material Properties for Parts 2 and 3

	"Modify Material…"	Select the **"Modify Material…"** command.
	"Aluminum (6061-T6)"	Highlight the "**Aluminum (6061-T6)**" item in the **"Select Material"** section. (See Figure 7.)

	"OK"	Click **"OK"** to accept the material property choice for Parts 2 and 3.

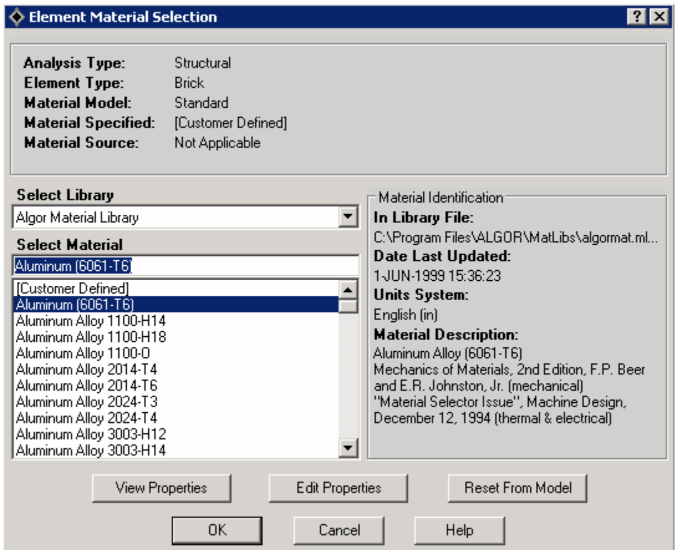


Figure 7: Specifying Material Properties for Parts 2 and 3

Adding Applied Temperatures

Add applied temperatures of 150° Fahrenheit to the brackets of the motor mount. These temperatures simulate the motor heating the mount. The location of the applied temperatures will be between the holes of the brackets because that is where bolts would hold the motor closest to the mount.

	"Selection: Select: Vertices"	Access the SELECTION pull-down menu and select the **"Select"** pull-out menu. Select the **"Vertices"** command.
	"View: Orientation: XZ Back"	Access the VIEW pull-down menu and select the **"Orientation"** command. Select the **"XZ Back"** command.
	"View: Zoom Area"	Access the VIEW pull-down menu and select the **"Zoom Area"** command.
	Mouse	Draw a rectangle around the bracket holes as shown in Figure 8 by clicking first at the top left corner then at the lower right corner of the desired area. The view will zoom in on the selected area.
	<Esc>	Press <Esc> to change the selection mode back to vertex selection.

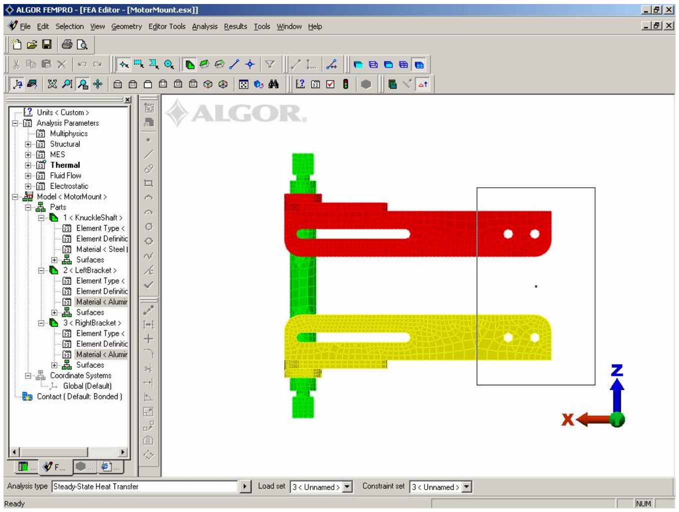


Figure 8: Zooming In on the Bracket Holes

	Mouse	Click on a node between the holes of the upper bracket. A selection mark will appear on the node.
	<Ctrl>-Mouse	Holding down the <Ctrl> key, click on a node between the holes of the lower bracket. The second node will be added to the selection set (see Figure 9).

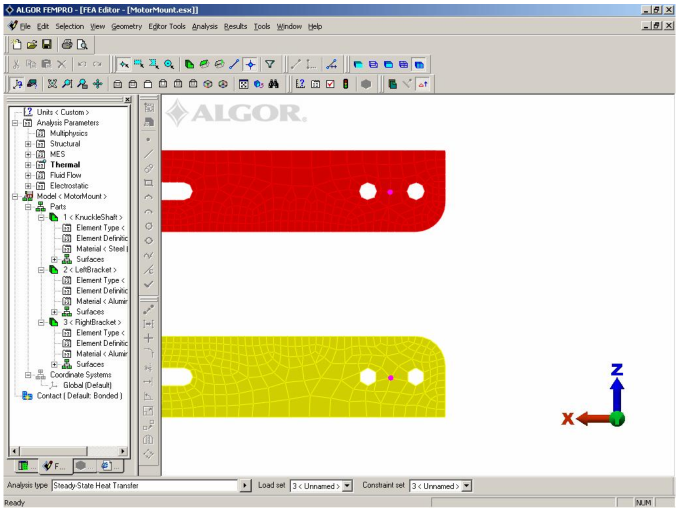


Figure 9: Selecting Nodes for Applied Temperatures

	Mouse	Right-click anywhere in the display area.
	"Add: Applied Temperatures…"	Choose **"Add: Applied Temperatures…"** (see Figure 10). The **"Creating 2 Nodal Applied Temperature Objects"** dialog will appear.

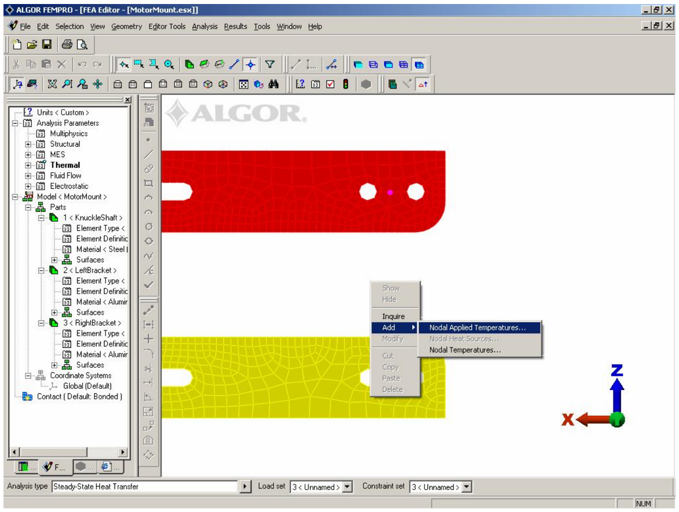


Figure 10: Adding Applied Temperatures

	150	Type **"150"** in the **"Magnitude"** field.
	1e6	Type **"1e6"** in the **"Stiffness"** field (see Figure 11). Stiffness controls the energy available to maintain the prescribed temperature. If enough heat is available, then the calculated temperature of the specified nodes will approach the applied temperature.
	"OK"	Press the **"OK"** button to accept the specified applied temperature values. Graphical symbols will appear on the selected nodes indicating that applied temperatures have been added.

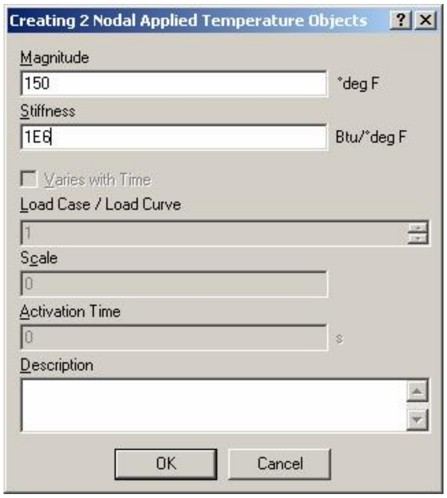


Figure 11: Defining Applied Temperatures

Applying a Convection Load

Apply a convection load to the outside surfaces of the model to simulate heat transfer to the surrounding air. The convection coefficient will be calculated using the "Film/Convection Coefficient Calculator".

	"View: Orientation: Isometric"	Access the VIEW pull-down menu and select the **"Orientation"** pull-out menu. Select the **"Isometric"** command.
	"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.

Apply the convection load to surfaces of Part 1, the knuckle shaft (green). For this model, you will add the convection load to the long cylindrical surfaces of Part 1.

	Mouse	Move the cursor over one of the two long, cylindrical surfaces of the knuckle (Part 1). If the cursor remains still, or hovers, over this surface, the tool-tips window will pop up, identifying the surface. See Figure 12. The surface will be identified as Part 1; Surface 15.

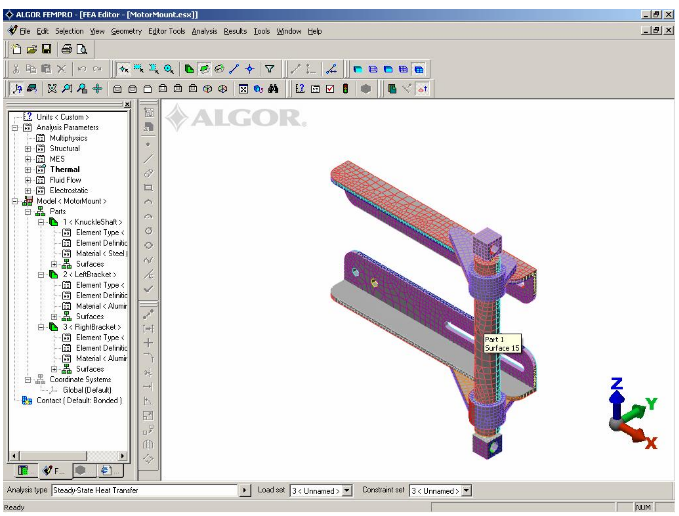


Figure 12: Tool-tips Window Identifies the Surface as Part 1; Surface 15

	Mouse	Move the cursor, and hover over the back cylindrical surface (shown in cyan) of the knuckle part. This surface will be identified by tool-tips as Part 1; Surface 4.

Having identified the surfaces to which we will apply convective loads, we are now ready to apply those loads.

	Mouse	In the tree view, click on the **"+"** sign to the left of the **"Surfaces"** heading for Part 1 to expand the list of surfaces.
	Mouse	Click on the **"Surface 4"** heading. The selected surface will be highlighted. Alternatively, you could select the surface on the 3D view of the model, and its name would be highlighted in the tree view.
	<Ctrl>-Mouse	Holding down the **<Ctrl>** key, right-click on the **"Surface 15"** heading to add it to the selection set

	"Add: Surface Loads…"	Select the **"Add"** pull-out menu and select the **"Surface Loads…"** command (see Figure 13).

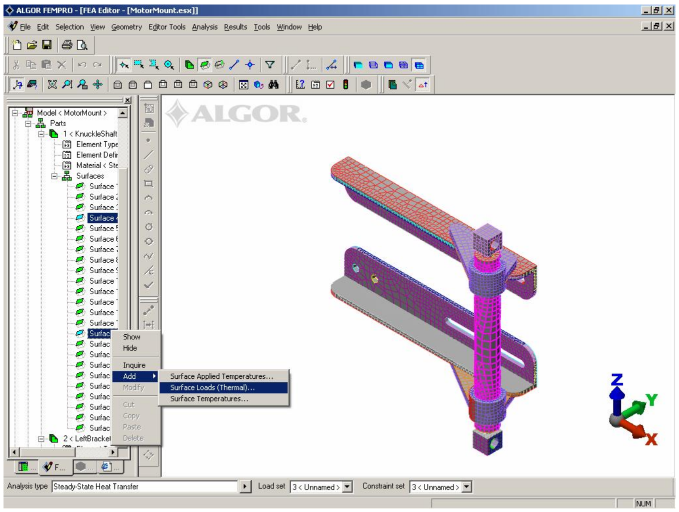


Figure 13: Adding Surface Loads to Surfaces 4 and 15 of Part 1

	"Calculate convection coefficient…"	Press the **"Calculate convection coefficient…"** button (see Figure 14).

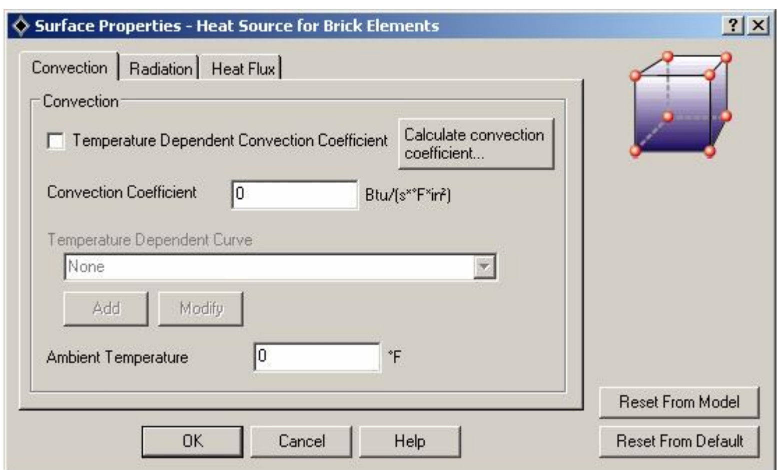


Figure 14: Accessing the Calculate Convection Coefficient Dialog

Enter data for the **"Film/Convection Coefficient Calculator"**.

	`17.6`	Type **"17.6"** in the **"Flow speed"** field in the **"Additional Parameters"** section of the **"General"** tab (see Figure 15). The convection coefficient will not be calculated without entering a value for this field. Other fields in the **"General"** tab can be left at the default values.

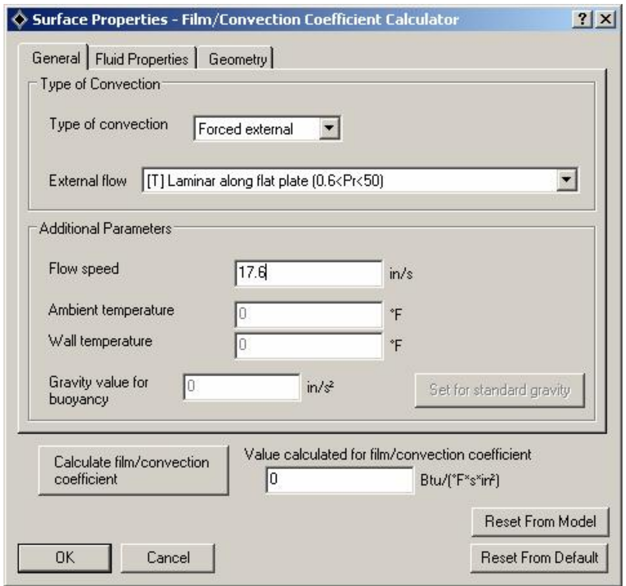


Figure 15: Specifying the Flow Speed

	Mouse	Click on the **"Fluid Properties"** tab.
	`1.167e-7`	Type **"1.167e-7"** in the **"Mass Density"** field. Remember that English (in) units are being used.
	<Tab>`2.56e-9`	Press <Tab> to advance the cursor to the **"Dynamic viscosity"** field and type **"2.56e-9"**.
	<Tab>3.5e-7	Press <Tab> to advance the cursor to the **"Thermal conductivity"** field, and type **"3.5e-7"**.
	<Tab>92.8	Press <Tab> to advance the cursor to the **"Specific heat"** field and type **"92.8"**. Make sure that your dialog looks like Figure 16.

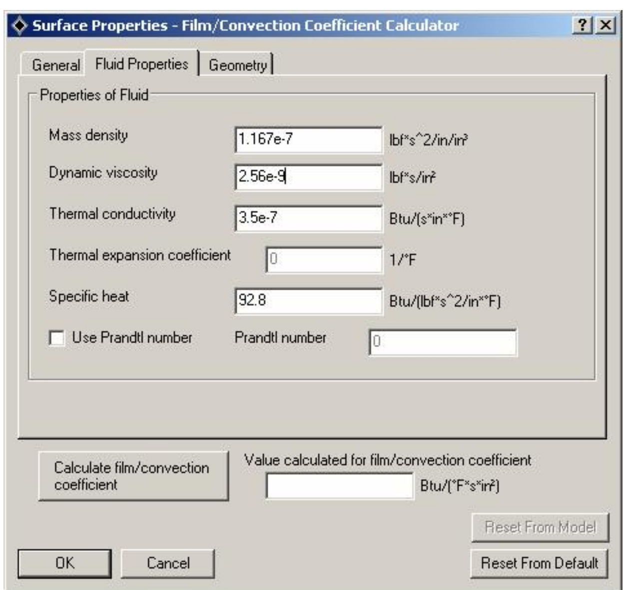


Figure 16: Specifying Fluid Properties

	Mouse	Click on the **"Geometry"** tab.
	10	Type **"10"** in the **"Length"** field (see Figure 17).

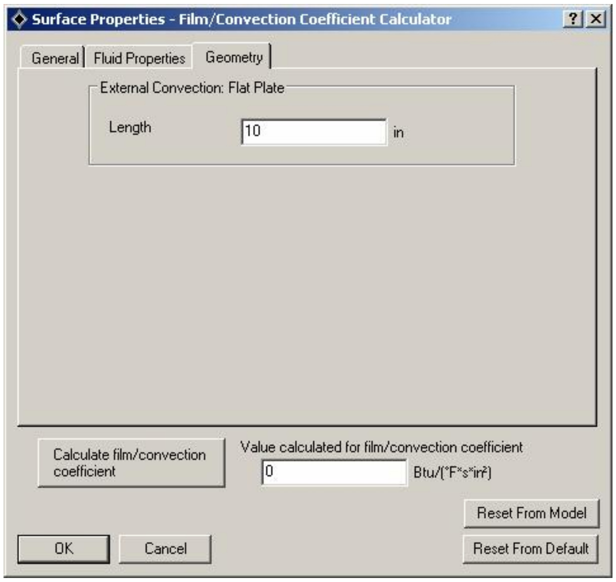


Figure 17: Specifying the Length

	"Calculate film/convection coefficient"	Press the **"Calculate film/convection coefficient"** button. A pop-up message will appear indicating that the resulting coefficient is 1.829e–6 BTU/(°F*s*in^2).
	"OK"	Press the **"OK"** button to close the pop-up message.
	"OK"	Press the **"OK"** button to close the **"Film/Convection Coefficient Calculator"** dialog. You will return to the **"Surface Properties"** dialog. A **"Film/Convection Coefficient"** dialog will appear asking, "Do you want to update the Convection Coefficient to the value calculated on the other screen?"
	"Yes"	Press the **"Yes"** button. The calculated value will be shown in the **"Convection Coefficient"** field.
	50	Type **"50"** in the **"Ambient Temperature"** field in the **"Convection"** section (see Figure 18).
	"OK"	Press the **"OK"** to apply the convection load. Graphical symbols will appear on the model display indicating that a convection load has been added to the selected surfaces of Part 1.

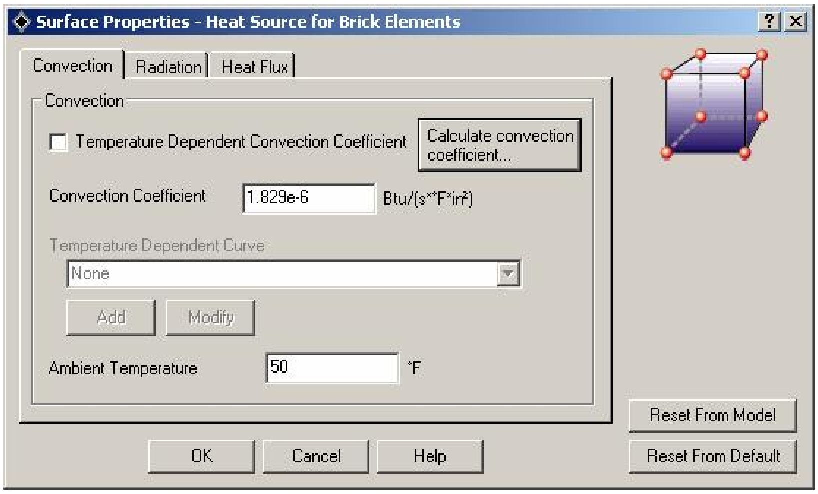


Figure 18: Specifying Convection Data for the Selected Surfaces of Part 1

Add the convection load to Part 2, the left bracket (red). For this model, you will add the convection load to all surfaces of Part 2 except for Surfaces 1 and 31. These are the surfaces that are in contact with Part 1, the knuckle shaft; so air does not flow across these surfaces.

	Mouse	In the model tree, click on the **"+"** sign to the left of **"Surfaces"** for Part 2. The list of surfaces will be displayed.
	"Surface 1"	Click on the **"Surface 1"** heading in the tree view. The selected surface will be highlighted.
	<Shift>-"Surface 35"	Holding down the <Shift> key, click on the **"Surface 35"** heading. All surfaces for Part 2 will be highlighted.
	<Ctrl>-"Surface 1"	Holding down the <Ctrl> key, click on the **"Surface 1"** heading to remove it from the selection set.
	<Ctrl>-"Surface 31"	Holding down the <Ctrl> key, click on the **"Surface 31"** heading to remove it from the selection set. All surfaces of Part 2 should be selected except for Surfaces 1 and 31.
	Mouse	Right-click on one of the highlighted surfaces.
	"Add: Surface Loads…"	Select the **"Add"** pull-down menu and select the **"Surface Loads…"** command (see Figure 19).

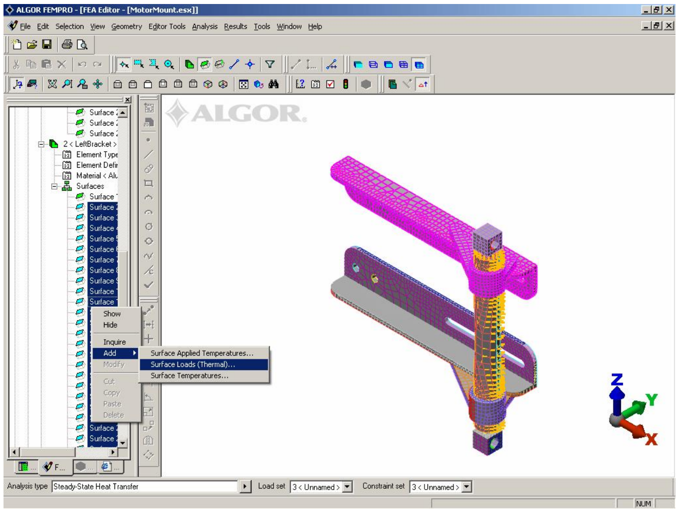


Figure 19: Adding Surface Loads to Selected Surfaces for Part 2

	`1.829e-6`	Type **"1.829e-6"**in the **"Convection Coefficient"** field to specify the same value that was calculated for Part 1.
	`50`	Type **"50"** in the **"Ambient Temperature"** field in the **"Convection"** section (see Figure 18).
	"OK"	Press the **"OK"** button to apply the convection load to the selected surfaces of Part 2.

On your own, add the convection load to Part 3, the right bracket (yellow), by following the same steps as for Part 2. That is, add the convection load to all surfaces of Part 3 except for Surfaces 1 and 31. When you are finished, your model should look like Figure 20.

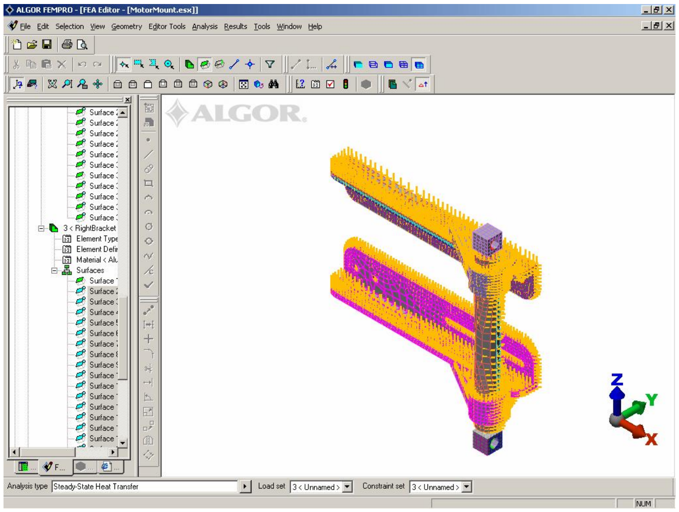


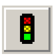
Figure 20: Model with Convection Load Added to Parts 1, 2 and 3

II. Analyzing the Model

In this phase, you will analyze the motor mount assembly model with the Steady-State Heat Transfer processor.

1. Analyzing the Model with the Steady-State Heat Transfer Processor

Analyze the model using the Steady-State Heat Transfer processor.

	**"Analysis: Perform Analysis…"**	Access the ANALYSIS pull-down menu and select the **"Perform Analysis…"** command. The **"Thermal"** analysis dialog will appear. The software will verify the geometry and finite element data and then the analysis will begin to run automatically. If you want to view the progress of these processes, you can press the **"Details >>"** button.

III. Reviewing the Results

In this phase, you will use the Results environment to view the analysis results.

1. Using the Results environment to View Analysis Results

Now you will use the Results environment again, this time to look at the results obtained from the Steady-State Heat Transfer analysis. You will view temperature results and rotate the model. Then, on your own, you can experiment with other capabilities.

Examining Results in the Results environment

When the analysis is finished, the Results environment will start automatically, and the temperature distribution on the model will be displayed.

Rotate the model to examine the temperature distribution.

	"View: Rotate"	Access the VIEW pull-down menu and select the **"Rotate"** command.
	Mouse	Click and drag the mouse to rotate the model.
	"View: Display: Shaded with Features"	Access the VIEW pull-down menu and select the **"Display"** pull-out menu. Select the **"Shaded with Features"** command (See Figure 21).

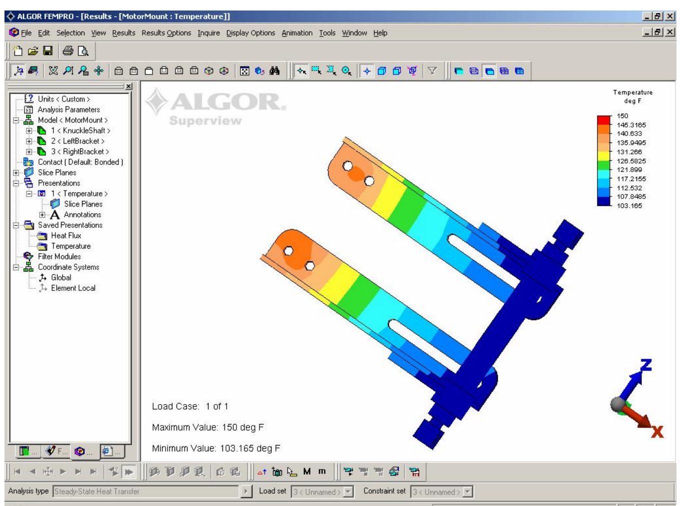


Figure 21: Temperature Results in Superview

Note: The exact temperature distribution, as well as the minimum and maximum temperatures shown in the legend, may vary depending on exactly which nodes were chosen when adding the temperature loads.

***Congratulations!* You have completed this Steady-State Heat Transfer Tutorial.**