Introduction Example

Motor Bracket Assembly Model

Brick Elements

Objective: To perform an analysis on a motor bracket that is loaded with pressure.

Geometry: Use the file *MotorMount.*igs located in the "introduction example\input file" directory as the input file for this exercise.

Loads: Pressure loads of 20 psi will be applied to the top of each bracket.

Constraints: The two holes on the shaft will be fully constrained.

Elements: Brick

Material: Steel (ASTM-A36)

Solution

Meshing the Model

Start FEMPRO from the Windows taskbar.

	"Start: Programs: ALGOR V17: FEMPRO"	Press the Windows **"Start"** button. Select the **"Programs"** pull-out menu and then select the **"ALGOR V17"** pull-out menu. Select the **"FEMPRO"** command.
	**"Open"**	Select the **"Open"** icon at the left side of the dialog.
	"IGES (*.igs, *.iges)"	Select the **"IGES (*.igs, *.iges)"** option in the CAD Files section of the **"Files of type"** drop-down box. Navigate to the directory where the model is located.
	***MotorMount*.IGS**	Select the *MotorMount*.IGS file in the "introduction example\input file" directory.
	"Open"	Press the **"Open"** button.
	"OK"	A dialog will appear asking you to choose the design scenario for this model. Press the **"OK"** button to accept the default of **"Static Stress with Linear Material Models"** in the **"Single analysis"** field.

When the file is opened, ALGOR will create a Direct Memory Image Transfer File (*filename*.dmit) from the IGES file. Once the file is directly opened in FEMPRO, the model is displayed in the CAD Solid Model environment. The solid model should be displayed as shown in Figure 1.

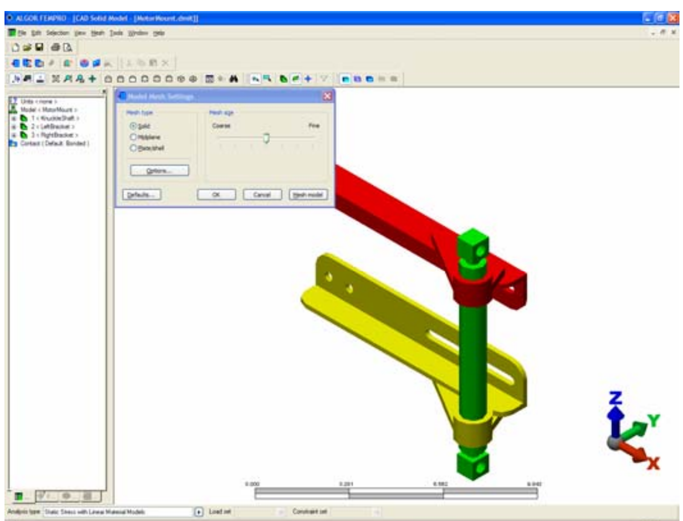


Figure 1: Model in the CAD Solid Model Environment

If the **"Model Mesh Settings"** dialog does not appear, access the MESH pull-down menu and select the **"Model Mesh Settings…"** command.

	"Mesh model"	Press the **"Mesh model"** button in the **"Model Mesh Settings"** dialog.
	"No"	Press the **"No"** button when asked to view the mesh results. A mesh will be displayed on the model at this time (see Figure 2).

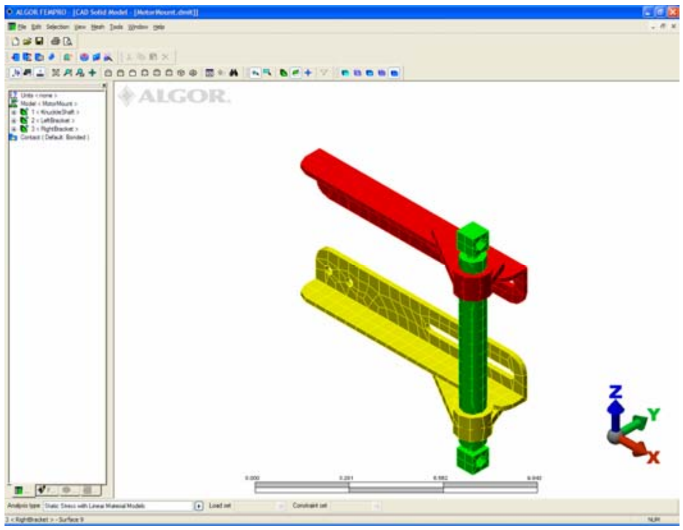


Figure 2: Meshed Model

	"Tools: FEA Editor"	Access the TOOLS pull-down menu and select the **"FEA Editor"** command to move to the FEA Editor environment where we will load and constrain the model.
	"OK"	When prompted to assign a unit system, press the **"OK"** button to accept the default **"English (in)"** system.

Defining the Material Data

	Mouse	Click on the **"Material"** heading for Part 1 in the tree view.
	<Ctrl>Mouse	Holding down the <Ctrl> key, click on the **"Material"** heading for Part 2 in the tree view.
	<Ctrl>Mouse	Holding down the <Ctrl> key, click on the **"Material"** heading for Part 3 in the tree view.
	Mouse	Right click on one of the selected headings.
	"Modify Material…"	Select the **"Modify Material…"** command (see Figure 3).

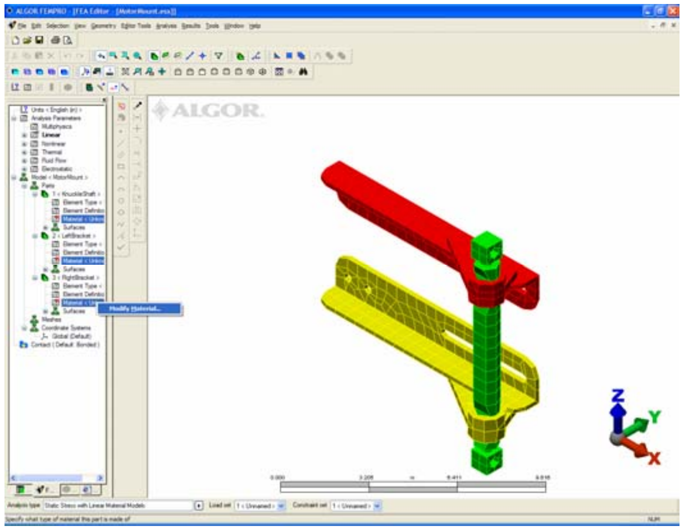


Figure 3: Defining the Materials

	"Steel (ASTM-A36)"	Select the **"Steel (ASTM-A36)"** item in the **"Select Material"** section of the **"Element Material Selection"** dialog.
	"OK"	Press the **"OK"** button to accept this material for all three parts.

Adding Loads and Constraints

	**"Selection: Select: Surfaces"**	Access the SELECTION pull-down menu and choose the **"Select"** pull-out menu. Select the **"Surfaces"** command.
	Mouse	Click on the top surface of one of the brackets.
	<Ctrl>Mouse	Holding down the <Ctrl> key, click on the top surface of the other bracket.
	Mouse	Right click in the display area.
	"Add: Surface Pressure/Tractions…"	Select the **"Add"** pull-out menu and select the **"Surface Pressure/Tractions…"** command (see Figure 4).

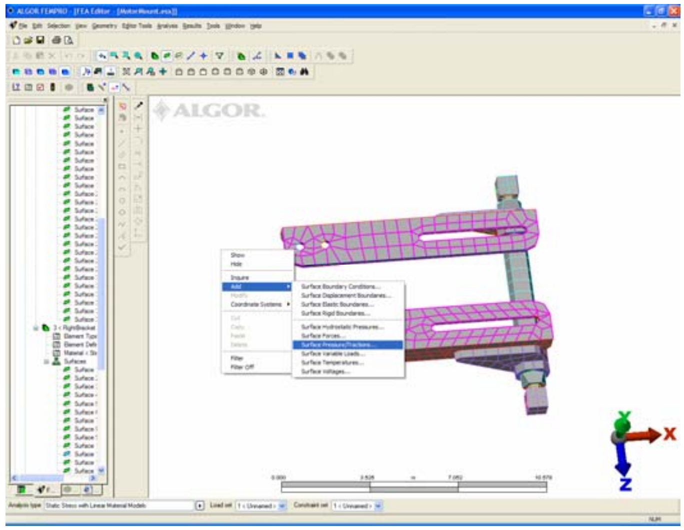
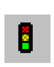


Figure 4: Applying Surface Loads

	20	Type **"20"** in the **"Magnitude"** field.
	"OK"	Press the **"OK"** button to accept this load.
	Mouse	Click on one of the inner surfaces of one of the holes in the shaft.
	<Ctrl>Mouse	Holding down the <Ctrl> key, click on the remaining three inner surfaces of the holes.
	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"	Press the **"Fixed"** button in the **"Predefined"** section.
	"OK"	Press **"OK"** to accept these surface boundary conditions.

Running the Analysis

	**"Analysis: Perform Analysis…"**	Access the ANALYSIS pull-down menu and select the **"Perform Analysis…"** command to analyze the model. The model will then be analyzed. The model will then be displayed in the Results environment to review the results.

Viewing the Results

The stress contour on the displaced shape will appear as shown in Figure 5.

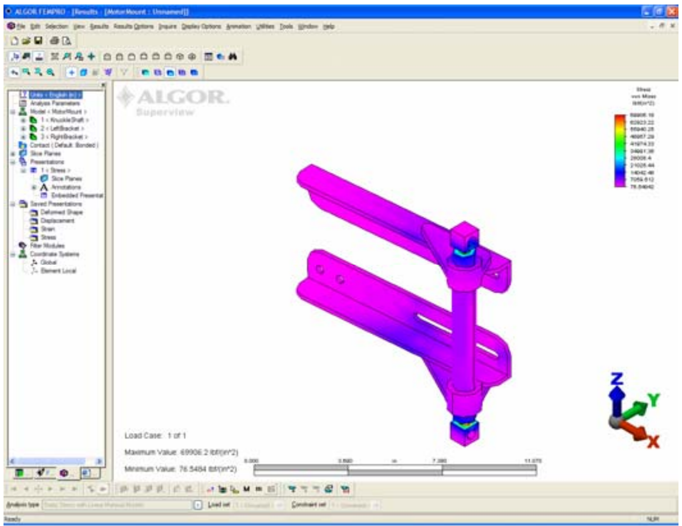


Figure 5: Stress Results Contour

To review a completed archive of this exercise, refer to the file *MotorMount*.ach in the "introduction example\results archive" directory.