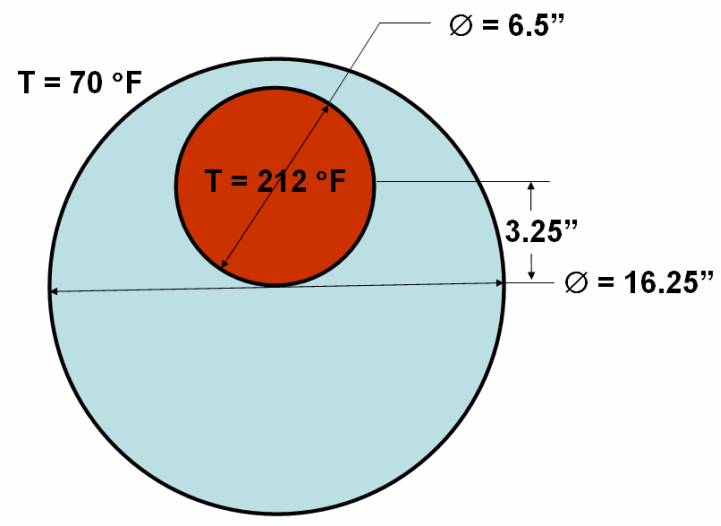
Steady-State Heat Loss of a Steam Pipe with Nonconcentric Insulation



2-D model with mapped mesh

Objective: Determine the temperature distribution.

Geometry: 2-D model with mapped mesh consists of the pipe located 3.25” off from the centerline of the insulation cylinder. The diameter of the pipe is 6.5”. The outside diameter of the insulation cylinder is 16.25”.

Loads: The temperature of the pipe’s outer wall is held at 212°F, and the temperature of the outside of the insulation is held at 70°F (air temperature).

Constraints: The temperature constraint.

Elements: 2-D

Thickness: Thickness = 1”

Material: Thermal conductivity; k = 0.0045 in*lbf/s/in°F

Solution

Creating the Sketches

Start FEMPRO from the Windows taskbar.

	"Start: Programs: ALGOR V16: FEMPRO"	Press the Windows **"Start"** button. Select the **"Programs"** pull-out menu and then select the **"ALGOR V16"** pull-out menu**.** Select the **"FEMPRO"** command.
	Mouse	Double click on the **"FEA Model"** icon in the **"New"** screen.
	CASE4	Type in a descriptive file name in the **"File name:"** field.
	"Save"	Press the **"Save"** button.
	"OK"	A dialog will appear asking you to choose the design scenario for this model. Select the **"Steady-State Heat Transfer"** in the **"Single analysis"** field.
	"OK"	Press the **"OK"** button to accept the default unit system of **"English (in)"**.
	**"New Sketch"**	Select the **"New Sketch"** command.
	"YZ"	Select the **"YZ"** option in the **"Plane"** drop-down box.
	"OK"	Press the **"OK"** button to create a sketch in the YZ plane at X=0.
	**"Geometry: Sketch Entities: Circle: Center and Point"**	Access the GEOMETRY pull-down menu and select the **"Sketch Entities"** pull-out menu. Select the **"Circle"** pull-out menu and select the **"Center and Point"** command.
	<Enter>	Press <Enter> to define the origin as the center of the circle.
	8.125 <Enter>	Type **"8.125"** in the **"Local X:"** field and press <Enter> to define the point (8.125,0) as a point on the circle.
	**"View: Enclose"**	Access the VIEW pull-down menu and select the **"Enclose"** command.
	3.25 <Enter>	Type **"3.25"** in the **"Local Y:"** field and press <Enter> to define the point (0,3.25) as the center of the circle.
	3.25 <Tab> 3.25 <Enter>	Type **"3.25"** in the **"Local X:"** field, press <Tab>, type **"3.25"** in the **"Local Y:"** field and press <Enter> to define the point (3.25,3.25) as a point on the circle.
	<Esc>	Press <Esc> to exit the circle command.
	**"Geometry: Sketch Entities: Center Line"**	Access the GEOMETRY pull-down menu and select the **"Sketch Entities"** pull-out menu. Select the **"Center Line"** command.
	9 <Enter>	Type **"9"** in the **"Local Y:"** field and press <Enter> to define the point (0,9) as the beginning of the center line.
	-9 <Enter>	Type **"-9"** in the **"Local Y:"** field and press <Enter> to define the point (0,-9) as the end of the center line.
	<Esc>	Press <Esc> to exit the center line command.
	**"Geometry: Tools: Trim"**	Access the GEOMETRY pull-down menu and select the **"Tools"** pull-out menu. Select the **"Trim"** command.

	Mouse	Click on the center line as the boundary.
	Mouse	Click on the big circle.
	Mouse	Click on the right hand side of the big circle. It will be removed from the sketch.
	Mouse	Click on the center line as the boundary.
	Mouse	Click on the small circle.
	Mouse	Click on the right hand side of the small circle. It will be removed from the sketch.
	<Esc>	Press <Esc> to exit the trim command.
	Mouse	Click on the center line.
	<Ctrl> Mouse	Holding down the <Ctrl> key, click on the small and big arc.
	**"Geometry: Tools: Mirror"**	Access the GEOMETRY pull-down menu and select the **"Tools"** pull-out menu. Select the **"Mirror"** command.
	Mouse	Click on the center line to select. It will be the magenta.
	**"Edit: Delete"**	Access the EDIT pull-down menu and select the **"Delete"** command. The selected entities will be deleted from the sketch**.**
	**"Geometry: Sketch: Finish Sketch"**	Access the GEOMETRY pull-down menu and select the **"Sketch"** pull-out menu. Select the **"Finish Sketch"** command.
	Mouse	Right click on the heading for Part 1 in the tree view.
	"Mesh: Between Two Objects…"	Select the **"Mesh"** pull-out menu and select the **"Between Two Objects…"** command.
	"60"	Type **"60"** in the "AA':" field in the "Divisions" section.
	"20"	Type **"20"** in the "AB:" field in the "Divisions" section.
	Mouse	Click on the left half of the small circle.
	Mouse	Click on the left half of the big circle.
	"Apply"	Press the **"Apply"** button.
	Mouse	Click on the right half of the small circle.
	Mouse	Click on the right half of the big circle.
	"Apply"	Press the **"Apply"** button. The mesh of this model is shown in Figure1.

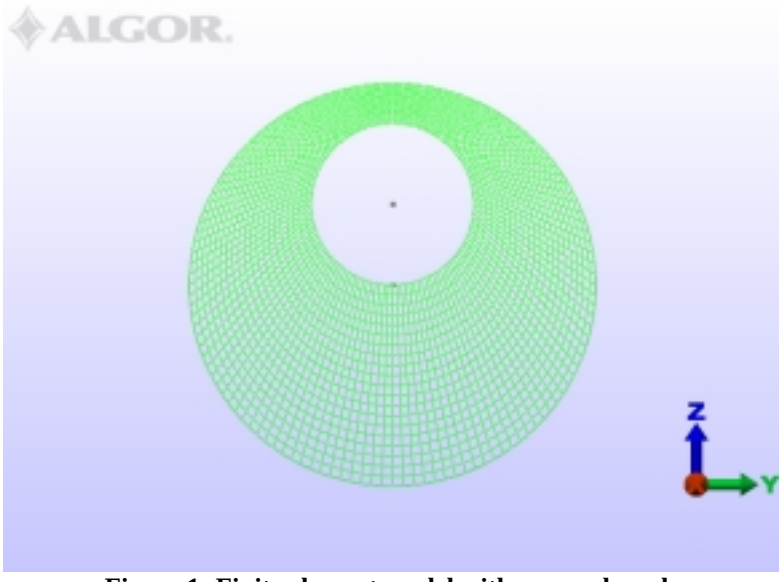


Figure 1: Finite element model with mapped mesh

Specifying the Element Information

	Mouse	Right click on the **"Element Type"** heading for Part 1 in the tree view.
	"2-D"	Select the "**2-D**" command.
	Mouse	In the tree view, click on the **"Element Definition"** heading for Part 1.
	"Modify Element Definition…"	Select the **"Modify Element Definition…"** command.
	"1"	Type **"1"** in the **"Thickness"** field.
	"OK"	Press the **"OK"** button.

Defining the Material Data

	Mouse	Right click on the **"Material"** heading for Part 1 in the tree view.
	"Modify: Material…"	Select the **"Modify Material…"** command.
	"Edit Properties"	Highlight the **"Customer Defined"** item from the list of available materials in the **"Select Material"** section.
	"0.0045"	Type **"0.0045"** in the **"Thermal Conductivity"** field.
	"OK"	Press the **"OK"** button to create the material.
	"OK"	Press the **"OK"** button to accept the selected material.

Adding Loads and Constraints

	**"View: Orientation: YZ Right"**	Access the VIEW pull-down menu and select the **"Orientation"** pull-out menu. Select the **"YZ Right"** command.
	**"Selection: Shape: Circle"**	Access the SELECTION pull-down menu and select the **"Shape"** pull-out menu. Select the **"Circle"** command.
	**"Selection: Select: Vertices"**	Access the SELECTION pull-down menu and select the **"Select"** pull-out menu. Select the **"Vertices"** command.
	Mouse	Draw a circle cover the small circle.
	Mouse	Right click in the display area.
	"Add: Nodal Apply Temperature…"	Select the **"Add"** pull-out menu and select the **"Nodal Apply Temperature…"** command.
	"212"	Type **"212"** in the **"Magnitude"** field.
	"1e6"	Type **"1e6"** in the **"Stiffness"** field (see figure 2). Stiffness controls the energy available to maintain the prescribe temperature. If enough heat is available, then the calculated temperature of the specific nodes will approach the applied temperature.
	"OK"	Press the **"OK"** button to accept the specified applied temperature value. Graphical symbol will appear on the selected nodes indicating that applied temperatures have been added.
	**"Selection: Shape: Point"**	Access the SELECTION pull-down menu and select the **"Shape"** pull-out menu. Select the **"Point"** command.

	Mouse	Click 120 nodes around the periphery of the big circle.
	Mouse	Right click in the display area.
	"Add: Nodal Apply Temperature…"	Select the **"Add"** pull-out menu and select the **"Nodal Apply Temperature…"** command.
	"70"	Type **"70"** in the **"Magnitude"** field.
	"1e6"	Type **"1e6"** in the **"Stiffness"** field (see figure 2). Stiffness controls the energy available to maintain the prescribe temperature. If enough heat is available, then the calculated temperature of the specific nodes will approach the applied temperature.
	"OK"	Press the **"OK"** button to accept the specified applied temperature value. Graphical symbol will appear on the selected nodes indicating that applied temperatures have been added.

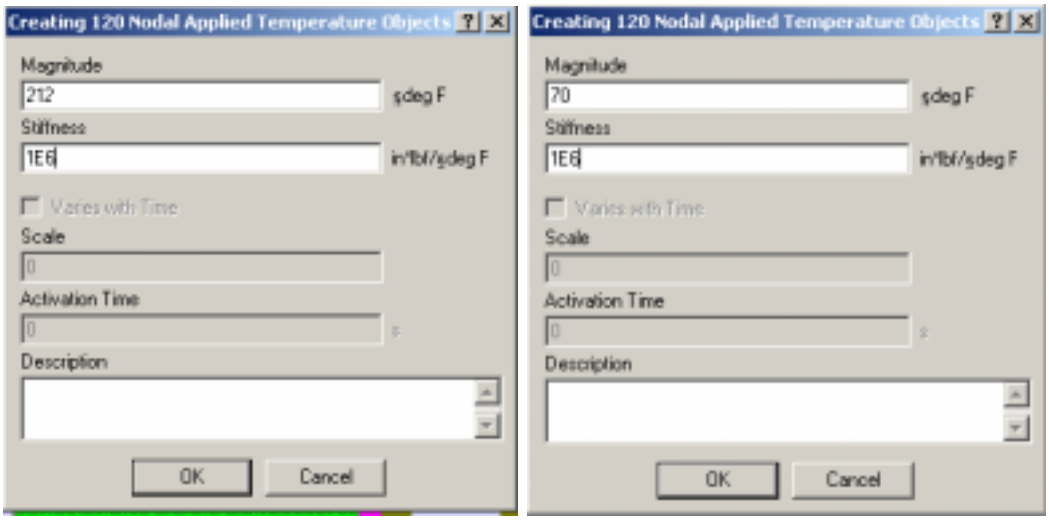


Figure 2: Defining apply temperature

Analysis

	**"Analysis: Perform Analysis…"**	Access the ANALYSIS pull-down menu and select the **"Perform Analysis…"** command to run the analysis. At the completion of the analysis, FEMPRO will automatically transfer to the Results environment.

Viewing the Results

When the analysis is finished, Superview will start automatically, and the temperature distribution will be displayed.

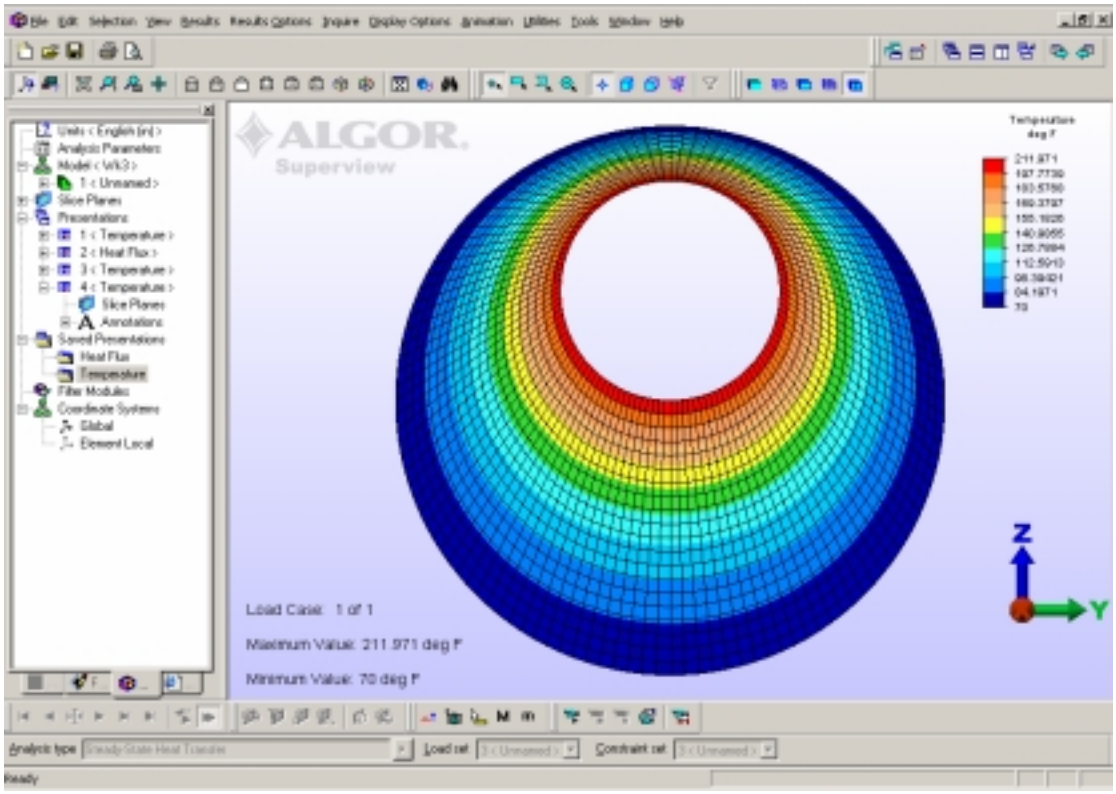


Figure 3: Superview display of temperature distribution.