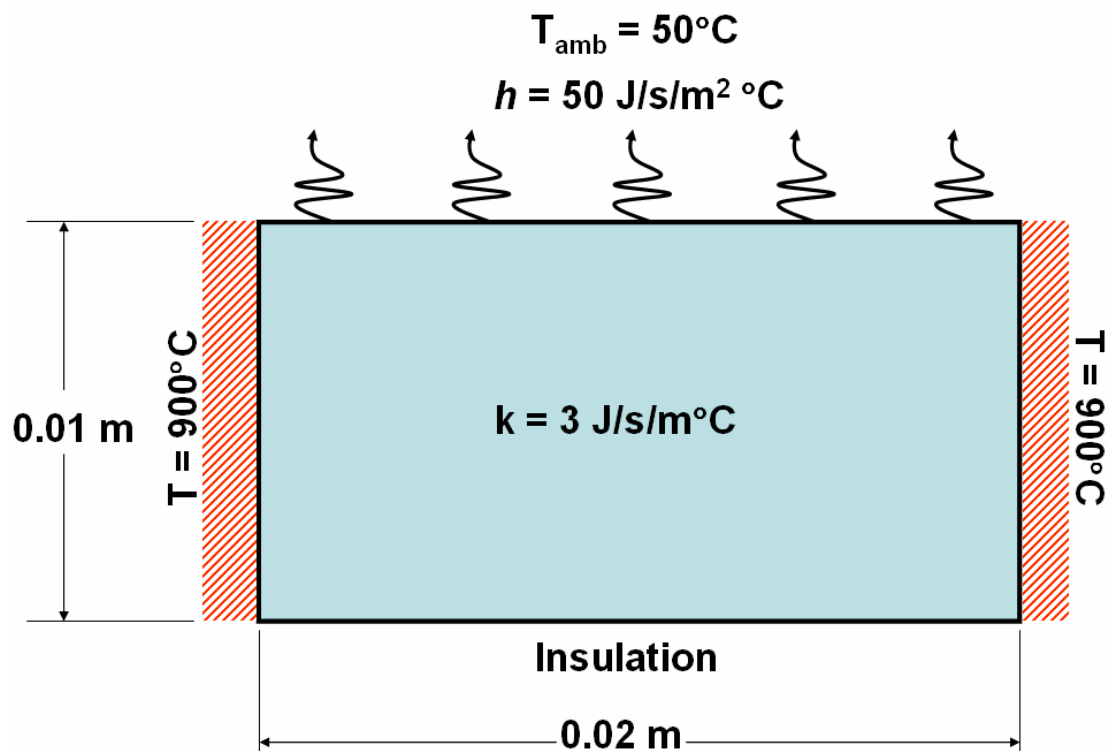


Ceramic Strip with Convection



Ceramic Strip with Convection

- Objective:** Determine the temperature distribution.
- Geometry:** The test problem is a 0.02 meter wide, by 0.01 meter high ceramic strip.
- Loads:** The bottom surface of the strip is perfectly insulated. The sides are held at a constant 900°C . The top is exposed to a convection environment with the following conditions: $T = 50^\circ\text{C}$ ambient temperature, $h = 50 \text{ J/s/m}^2\text{ }^\circ\text{C}$
- Constraints:** The temperature constraint.
- Elements:** 2-D
- Thickness:** Thickness = 1 m.
- Material:** Thermal conductivity; $k = 3 \text{ J/s/m}^\circ\text{C}$

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.
	CASE5	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.
	"Unit System: Metric mks (SI)"	Select the "Unit System" pull-out menu and select the "Metric mks (SI)" option.
	"OK"	Press the "OK" button to accept the Metric mks (SI) unit system.
	"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: Rectangle"	Access the GEOMETRY pull-down menu and select the "Sketch Entities" pull-out menu. Select the "Rectangle" command.
	<Enter>	Press <Enter> to define the first corner of a rectangle.
	0.02 <Tab> 0.01 <Enter>	Type "0.02" in the "Local X:" field, press <Tab>, and type "0.01" in the "Local Y:" field and press <Enter> to define the point (0.02,0.01) as the opposite corner.
	<Esc>	Press <Esc> to exit the rectangle command.
	"View: Enclose"	Access the VIEW pull-down menu and select the "Enclose" command.
	"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.
	"30"	Type "30" in the "AB:" field in the "Divisions" section.
	Mouse	Click on the left of the top line.
	Mouse	Click on the left of the bottom line.
	"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.
	"3"	Type "3" 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: 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 box around the left of the rectangle.
	<Ctrl> Mouse	Draw a box around the right of the rectangle.
	Mouse	Right click in the display area.
	"Add: Nodal Apply Temperature..."	Select the "Add" pull-out menu and select the "Nodal Apply Temperature..." command.
	"900"	Type "900" in the "Magnitude" field.
	"1e6"	Type "1e6" in the "Stiffness" field. 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: Select: Lines"	Access the SELECTION pull-down menu and select the "Select" pull-out menu. Select the "Lines" command.
	Mouse	Draw a box around the top of rectangle.
	Mouse	Right click in the display area.
	"Modify Attributes"	Access the GEOMETRY pull-down menu and select the "Tools" pull-out menu. Select the "Modify Attributes..." command.
	2	Type "2" in the "Surface:" field.
	"OK"	Press the "OK" button.

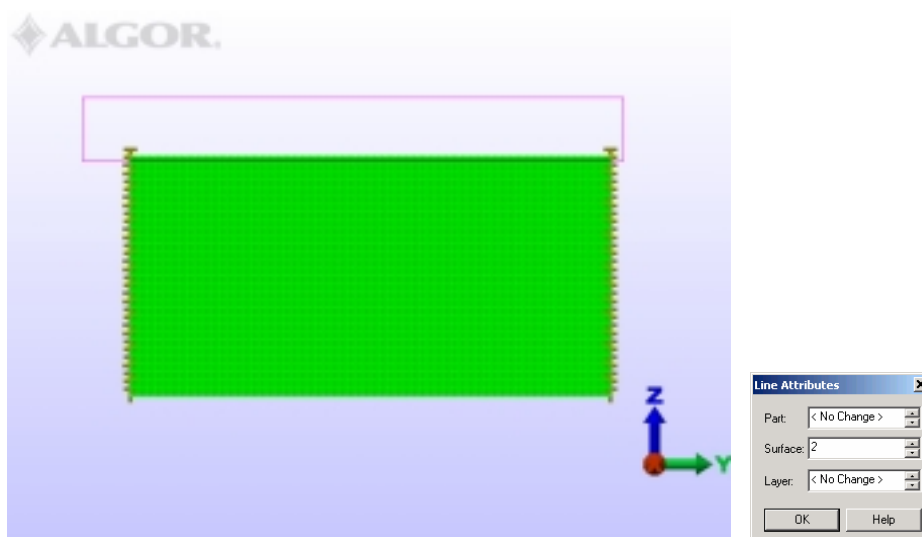




Figure 2: Line Attributes Dialog

	"Selection: Shape: Point"	Access the SELECTION pull-down menu and select the "Shape" pull-out menu. Select the "Point" command.
	"Selection: Select: Surfaces"	Access the SELECTION pull-down menu and choose the "Select" pull-out menu. Select the "Surfaces" command. This will allow you to select the surfaces.
	Mouse	Click on the top of the rectangle.
	Mouse	Right click in the display area.
	"Add: Surface Load (Thermal)"	Select the "Add" pull-out menu and select the "Surface Load (Thermal)..." command.
	50	Type "50" in the "Convection Coefficient" field (see figure 3).
	50	Type "50" in the "Ambient Temperature" field.
	"OK"	Press the "OK" button to apply these surface forces.

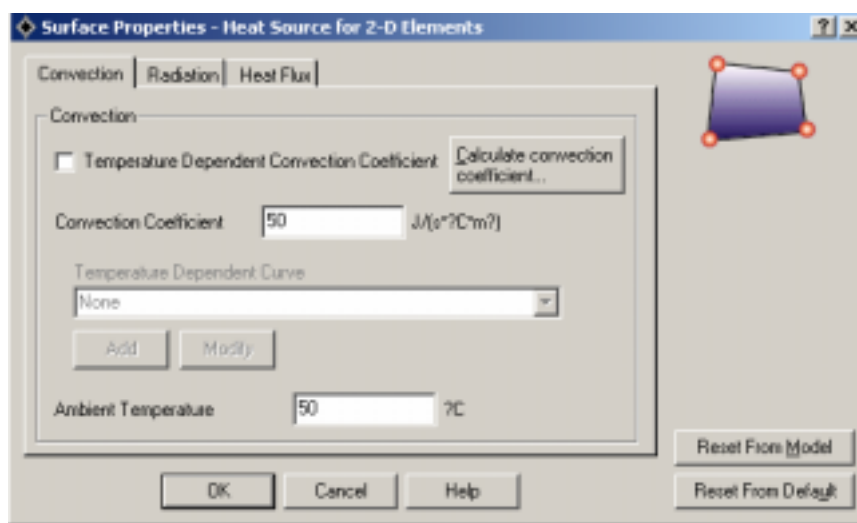



Figure 3: Heat Source Dialog

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.

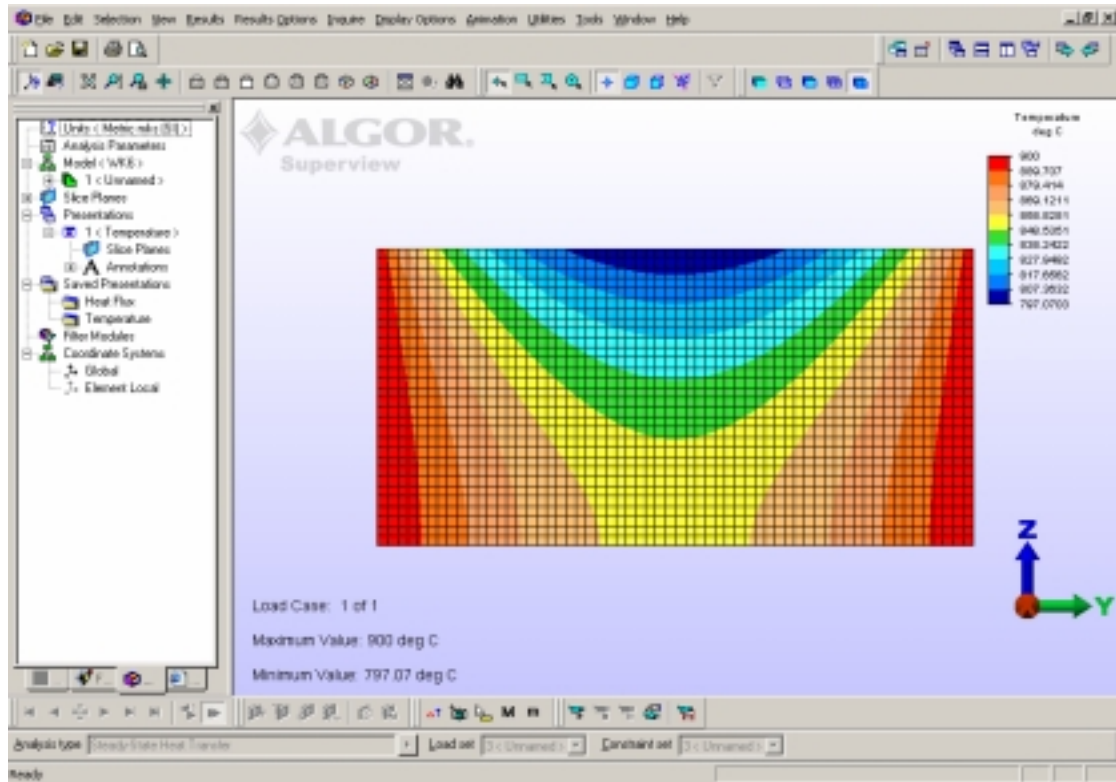


Figure 4: Superview display of temperature distribution.