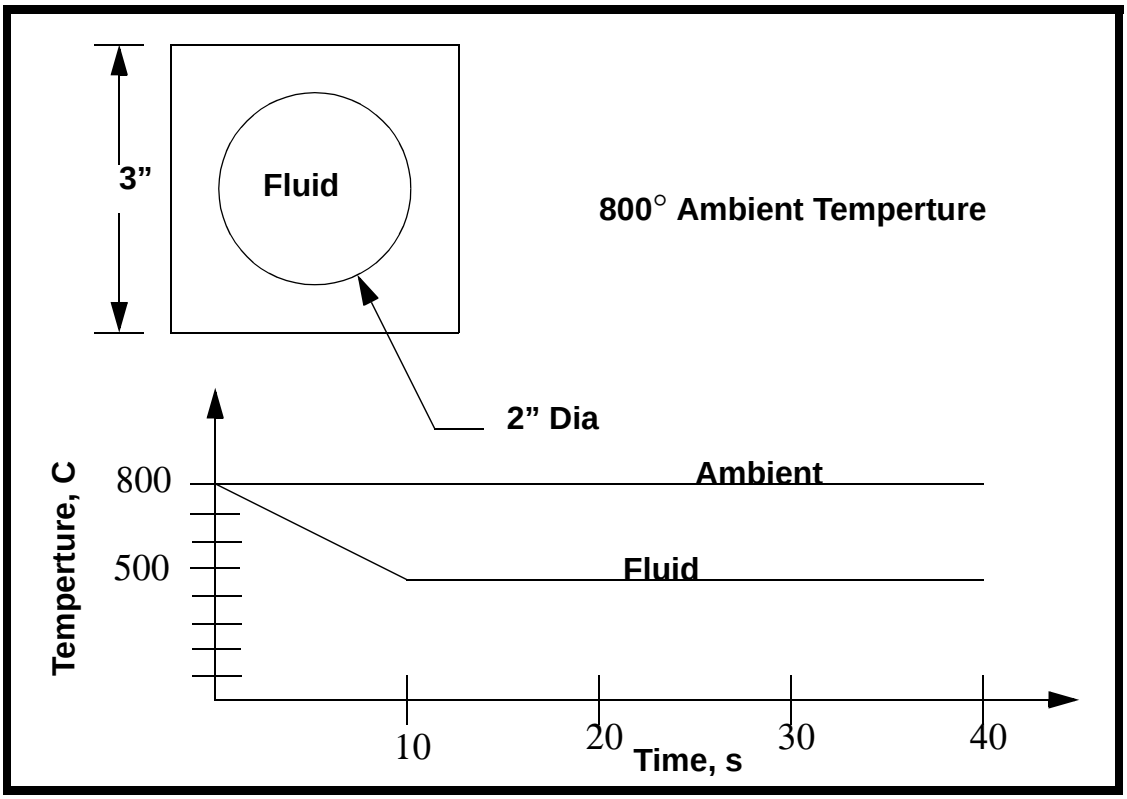
LESSON 25

Heat Transfer Analysis of a Pipe



Objectives:

- Transient Heat Transfer Analysis
- Model Convection, Conduction

Model Description:

In this exercise, you will be modelling a 2-Dimensional cross section of a container holding a fluid. Initially, the outside and inside of the container are at 800°. The temperature of the inner fluid in the model drops from 800° to 500° in a period of 10 seconds. The variation of temperature will be calculated as a function of time.

Suggested Exercise Steps:

- Create the Geometry shown in the previous shape.
- Mesh the model with Quad8 elements using a 4x4 mesh.
- Specify material properties as:

  conductivity = 4.85E-4 BTU/s-°F-in,

  Specific Heat = 0.116 BTU/lbm-°F, and

  Density = 0.283 lb/in^3.

- Submit the analysis and post process the results.

Exercise Procedure:

1. Create a new database named **thermal_flow.db**.

File/New ...

Database Name: **thermal_flow.db**

OK

2. Change the preference type to **MSC.Marc**.

Analysis Code: **MSC.Marc**

Analysis Type: **Thermal**

OK

3. Create the geometry for the model.

■ **Geometry**

Action: **Create**

Object: **Curve**

Method: **Revolve**

Total Angle: **45**

Point List: **[1, 0, 0]**

Apply

Create a second curve using the options that follow:

Action: **Create**

Object: **Curve**

Method: **XYZ**

Vector Coordinates List: **<0, 1.5, 0>**

Origin Coordinates List: **[1.5, 0, 0]**

Apply

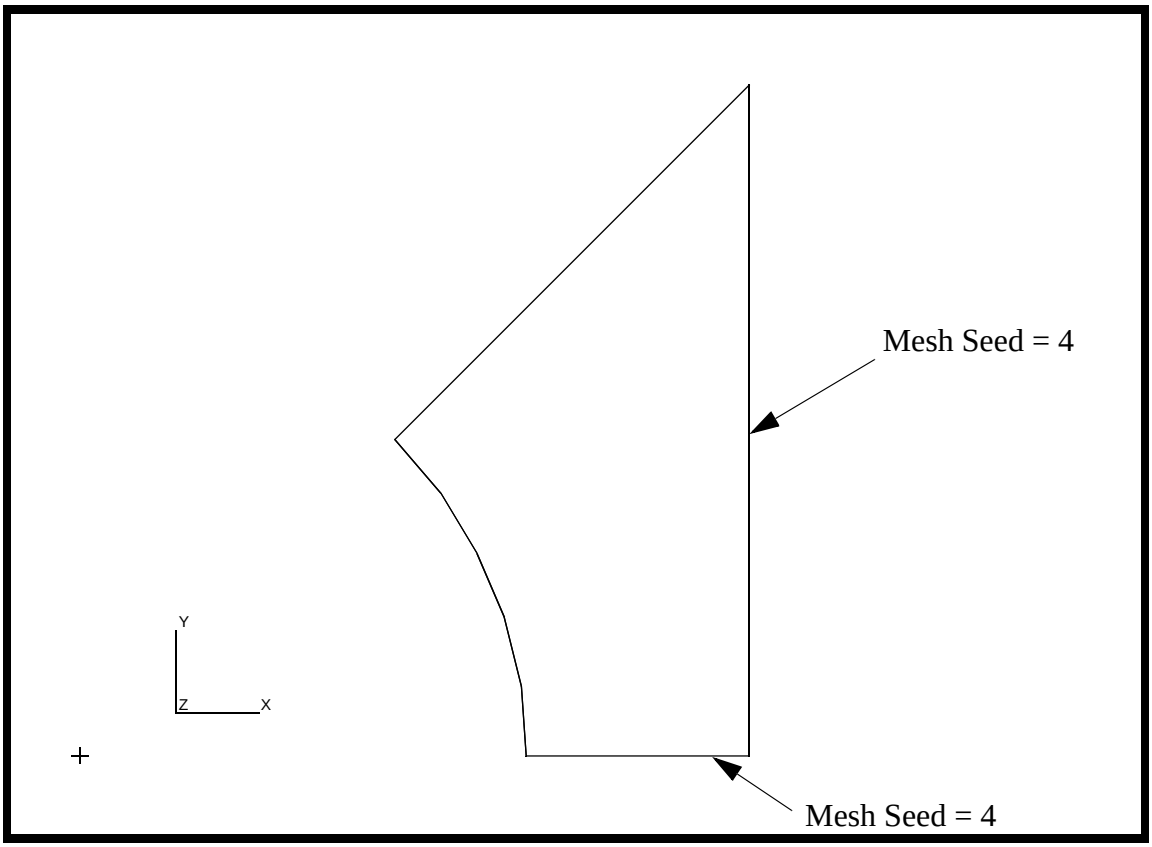
Next, create a surface between the two curves.

Action:	**Create**
Object:	**Surface**
Method:	**Curve**
Curve Option:	**2 Curve**
Starting Curve List:	**Curve 2**
Ending Curve List:	**Curve 1**

Apply

Your model should now look like the one shown in Figure 25.1:

Figure 25.1 - 1/8 symmetry model of container holding fluid



4. Create two mesh seeds on the newly created surface. Use a Mesh Seed of 4 on the vertical and the horizontal edges.

■ **Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**Uniform**

● **Number of Elements**

Number:	**4**
Curve List:	see Figure 25.1

The bottom edge is **Surface 1.1**.

The next edge to be seeded is the right side, **Curve 2**, It will also have 4 elements.

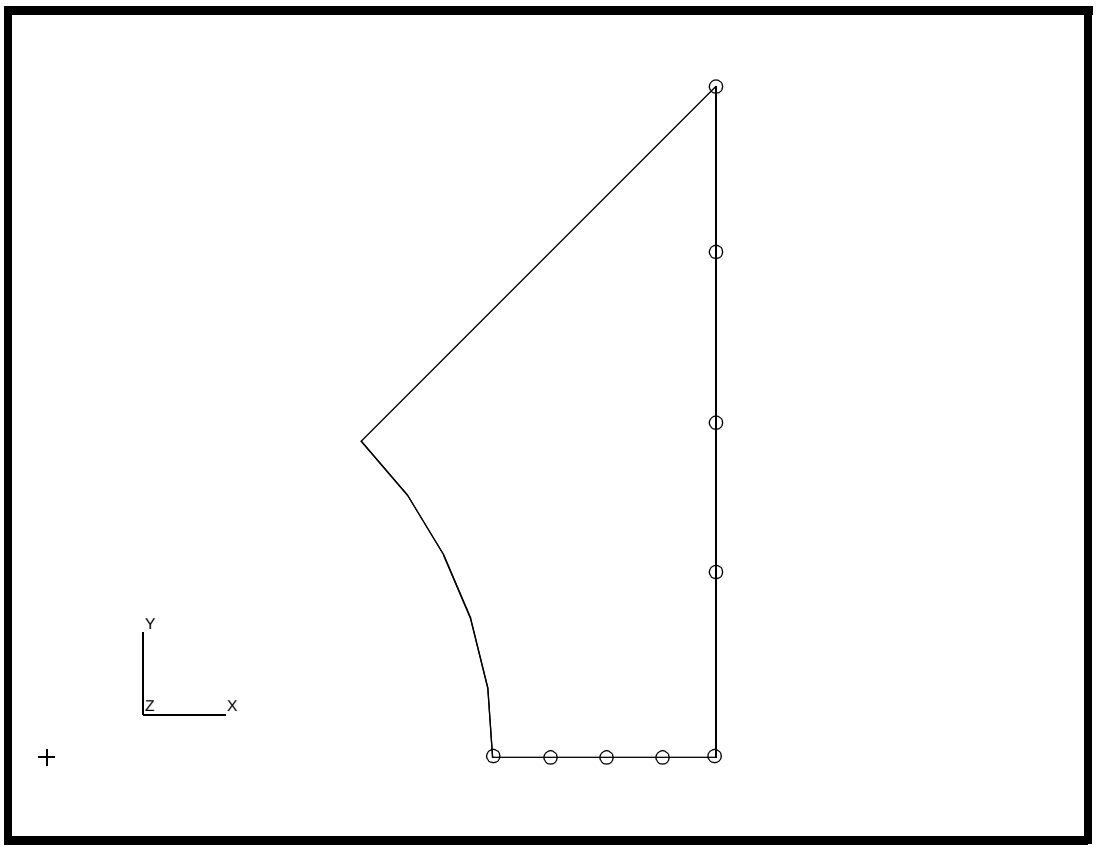
Action:	**Create**
Object:	**Mesh Seed**
Method:	**Uniform**

● **Number of Elements**

Number:	**4**
Curve List:	see Figure 25.1

Your model should now appear as shown in Figure 25.2:

Figure 25.2 - Model with mesh seeds



5. Create a group **fem** and make it current. This group will contain the finite elements

Group/Create ...

New Group Name: **fem**

■ **Make Current**

Apply

Cancel

6. Mesh the surface using **Quad8's**.

■ **Elements**

Action: **Create**

Object: **Mesh**

Type: **Surface**

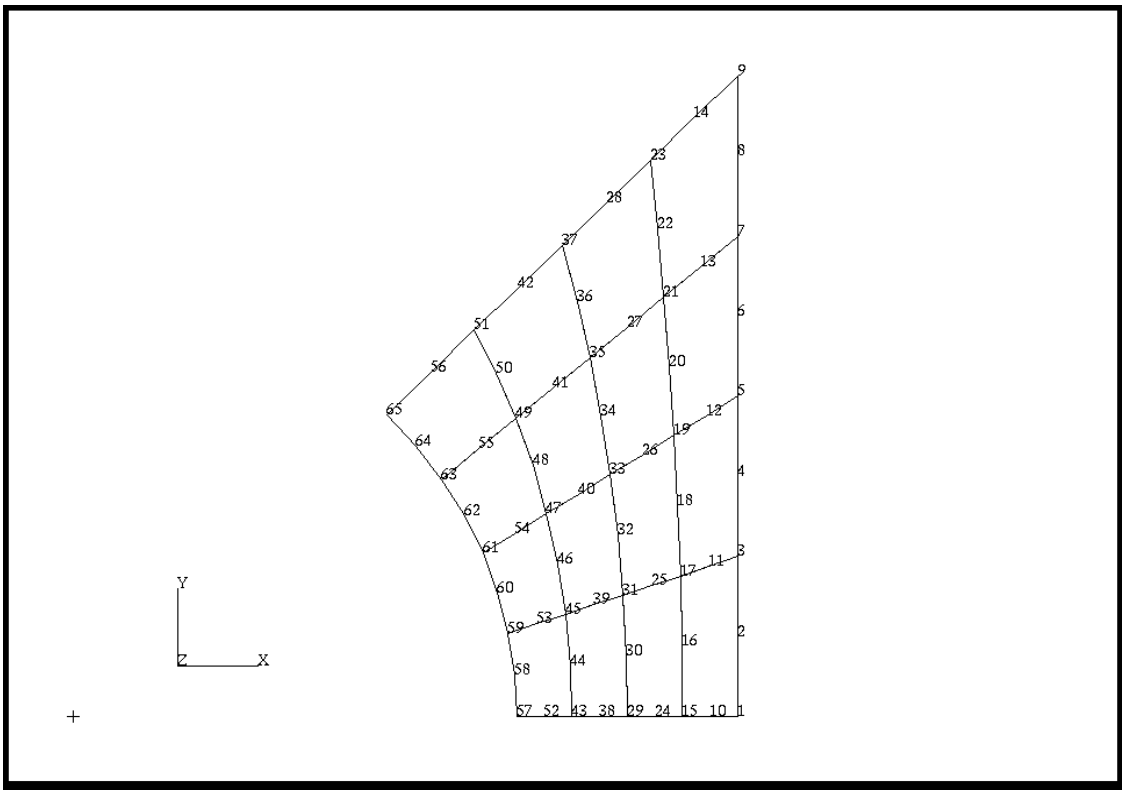
Element Topology: **Quad8**

Surface List: **Surface 1**

Apply

Your model should now appear as shown in Figure 25.3:

Figure 25.3 - Meshed model



7. Create the material **steel**, with thermal properties.

■ **Materials**

Action: **Create**

Type: **Isotropic**

Method: **Manual Input**

Material Name : **steel**

Input Properties...

Constitutive Model: **Thermal**

Conductivity:	**0.000485**
Specific Heat:	**0.116**
Density:	**0.283**

OK

Apply

8. Create the element properties, applying the steel material data set to all the elements.

■ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**prop1**
Options:	**Planar**
	Standard Formulation

Input Properties...

Material Name :	**steel**
Thickness:	**1.0**

OK

Select Members:	**Surface 1**

Add

Apply

9. Create a time dependent field, which will be applied to the boundary conditions.

■ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**

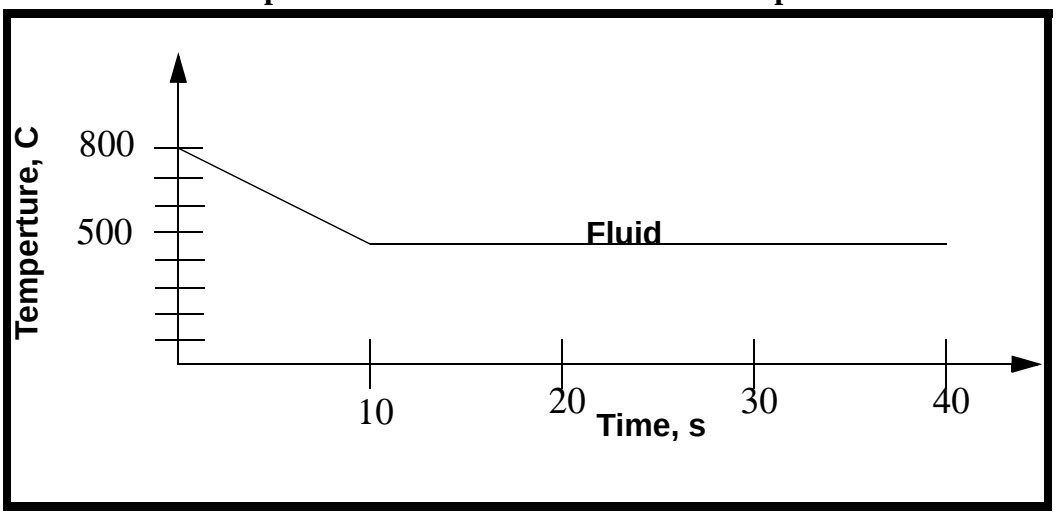
Field Name: **inner_temp**

Active Independent Variable: ■ **Time (t)**

Input Data...

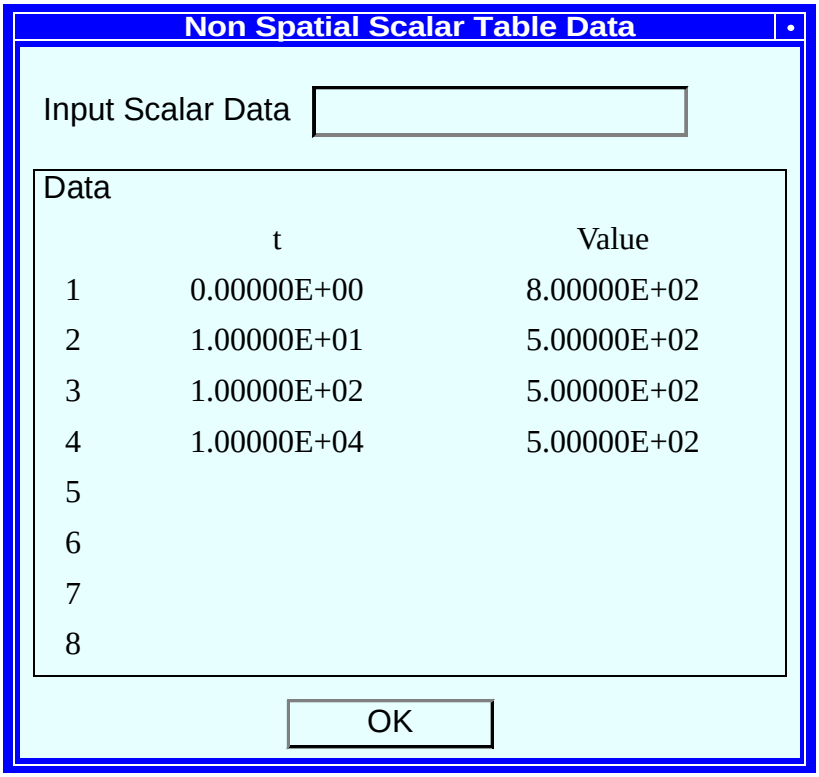
The *Time/Frequency Scalar Table Data* form needs to be filled out as shown in Table 1.

Table 1: Temperature vs. Time data for Inner Temperatures



Time	Temp
0	**800**
10	**500**
100	**500**
10000	**500**

To fill in the table, click on the cell you wish to edit, enter the value in the *Input Scalar Data* databox and press <Return>. The table will automatically tab down.



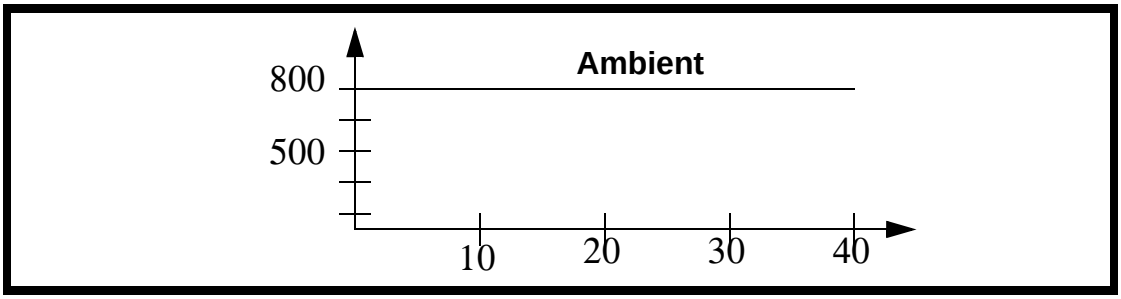
OK

Apply

Repeat the process above entering the name **outer_temp** in the *Field Name* databox and using the table data shown below.

The *Time/Frequency Scalar Table Data* form needs to be filled out as shown in Table 2.

Table 2: Temperature vs. Time data for outer Temperatures



Time	Temp
0	**800**
10	**800**
100	**800**
10000	**800**

10. Create a time dependent load case.

For a transient analysis, structural or thermal, it is required that you define a transient load case prior to creating the LBC's.

■ **Load Cases**

Action: **Create**

Load Case Name: **transient_load_case**

Load Case Type: **Time Dependent**

Apply

11. Create the loads and boundary conditions for the model.

■ **Load/BCs**

Action: **Create**

Object: **Initial Temperature**

Type: **Nodal**

New Set Name: **initial_t**

Input Data...

Temperature:	**800**

OK

Select Application Region...

Geometry Filter:	● **FEM**
Select Nodes:	select all posted nodes

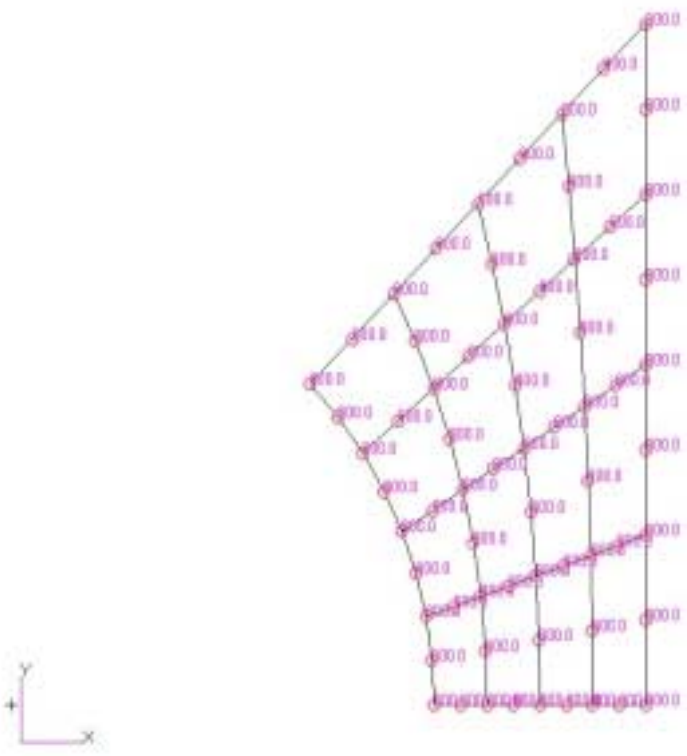
Add

OK

Apply

Your model should now look like the one shown in Figure 25.4:

Figure 25.4 - Initial temperature profile of model



12. Create the convection boundary conditions for the inner and outer surfaces.

In the *Load/Boundary Conditions* form change the *Object* option menu to **Convection**.

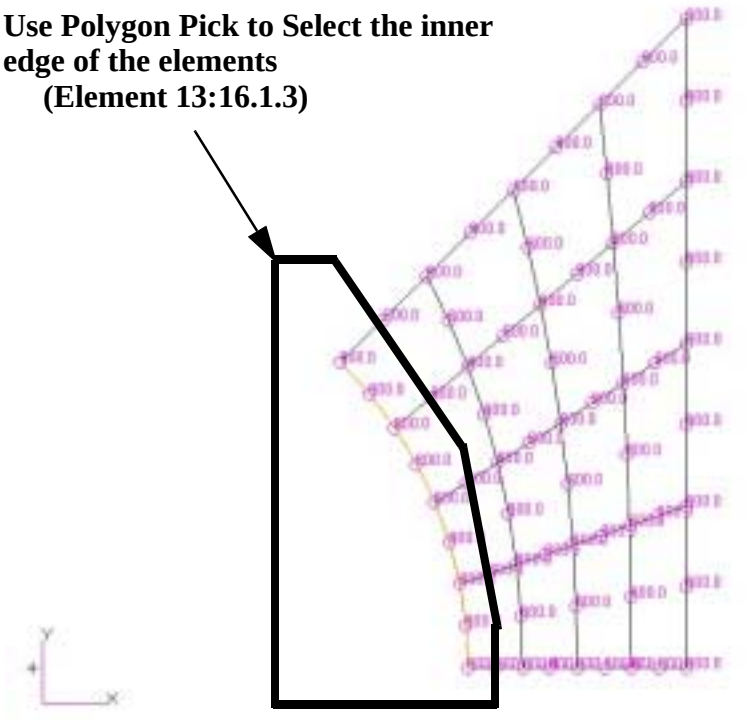
Action:	**Create**
Object:	**Convection**
Type:	**Element Uniform**
New Set Name:	**inner_convection**
Target Element Type:	**2D**
Input Data...	
Edge Convection:	**0.0005**
Ambient Temp:	**1**
Time Dependence: (next to Ambient Temp)	**inner_temp**
OK	
Select Application Region...	
Geometry Filter:	● **FEM**
Element Edges:	see Figure 25.5

Click on the Element Edge icon, as shown below, in the select menu.

Edge of Element

Select 2D Elements or Edges:	as shown in Figure 25.5

Figure 25.5 - Element edges to select for inner_temp



Hint: to make the selection easier, you may want to use a polygon pick (hold down the <ctrl> pick while selecting the corners or the polygon).

Add

OK

Apply

New Set Name: **outer_convection**

Input Data...

Edge Convection: **0.00001**

Ambient Temp: **1**

Time Dependence: **outer_temp**
(next to Ambient Temp)

OK

Select Application Region ...

Geometry Filter: ● **FEM**

Element Edges: see Figure 25.6

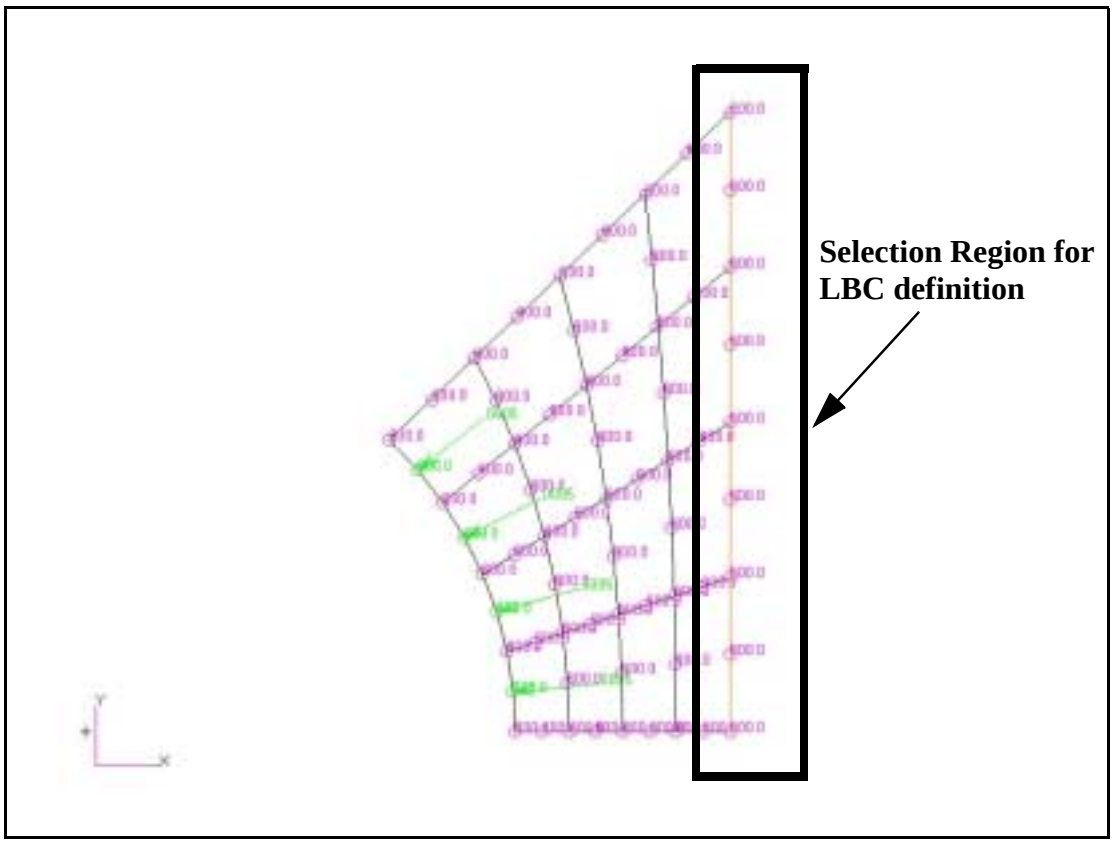
Add

OK

Apply

Click in the *Select 2D Elements or Edges* databox. In the Select Menu that appears, click on the Element Edge icon. Select all the element edges on the right edge of the model.

Figure 25.6 - Elements to select for outer_temp



13. Create the analysis step

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**thermal_flow**

Load Step Creation...

Job Step Name:	**transient case**
Solution Type:	**Transient Heat Transfer**

Solution Parameters...

Maximum Temperature Change Allowed:	**20**
Initial Time Step Size:	**2**
Total Time:	**10000**
Max # of Steps:	**100**

OK

Select Load Cases...

transient_load_case

OK

Apply

Cancel

Load Step Selection...

Selected Job Steps:	**transient case**

OK

Apply

The analysis job will take (on average) about 2 minutes to run. When the job is done there will be a results file titled **thermal_flow.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at the ***thermal_flow.log***, ***thermal_flow.sts***, and ***thermal_flow.out*** files during and after the analysis is complete. Also, a more convenient method is to use the **Monitor** action in the **Analysis** application.

Action: **Monitor**

Object: **Job**

View Status File...

When the end of the line has the following line, tyou know that the job has completed successfully: Job ends with exit number: 3004

14. After the job has completed execution, import the results.

■ **Analysis**

Action: **Read Results**

Select Results File...

Available Files: **thermal_flow.t16**

OK

Apply

15. Create a fringe plot of the last step.

First, you will clean up the graphics window. Use the Clean Up broom icon to remove all Loads/Boundary conditions markers:

Reset Graphics

Post the group fem before displaying the results.

Group/Post...

Select Group to Post: **fem**

Apply

Cancel

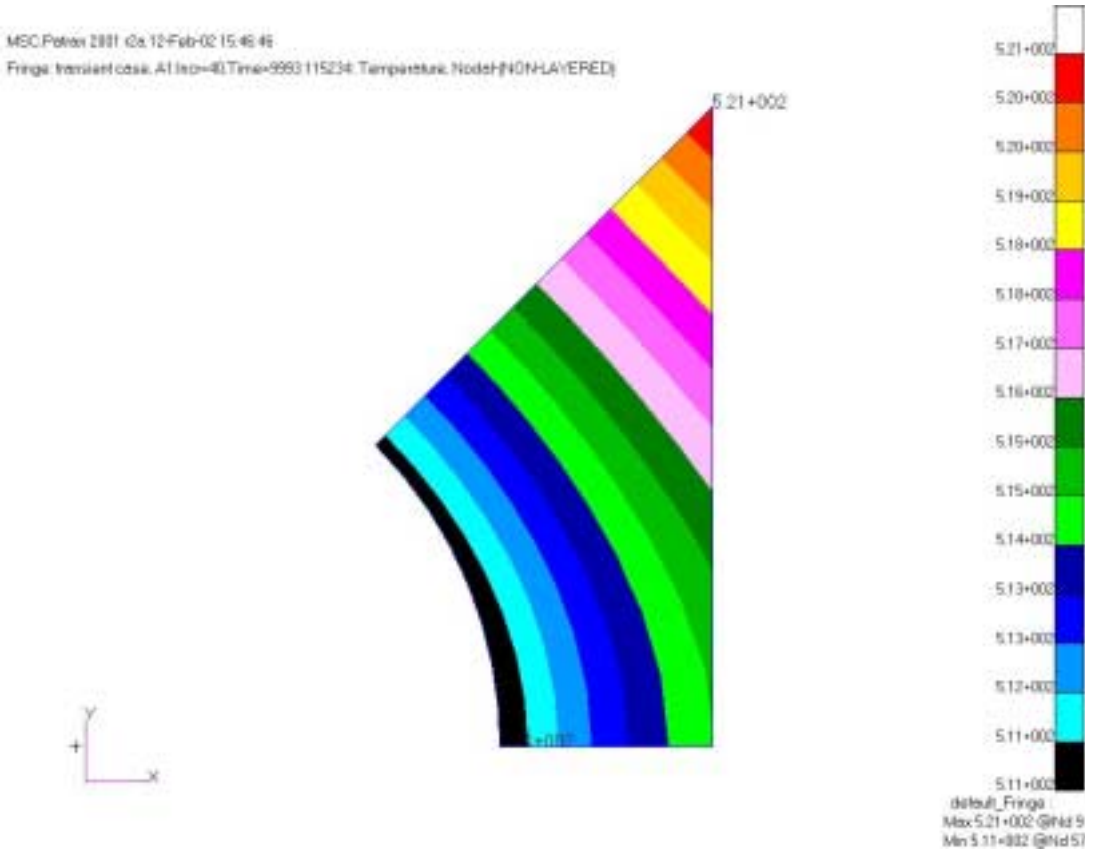
■ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	select last step
Select Fringe Result:	**Temperature, Nodal**

Apply

Your plot should look like the one shown in Figure 25.7:

Figure 25.7 - Resulting steady state temperature distribution



16. Plot the temperature as a function of time.

In this step, you will select 3 nodes to plot their temperature as a function of time. The three nodes are located at the upper right tip, upper left corner, and in the middle of the top edge.

■ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

Click on the **View Subcases** icon then the **Select Subcases** to bring up the *Select Result Case* form


Select Result Case:	**transient_case**
Filter Method	**All**

Filter

Apply

Close

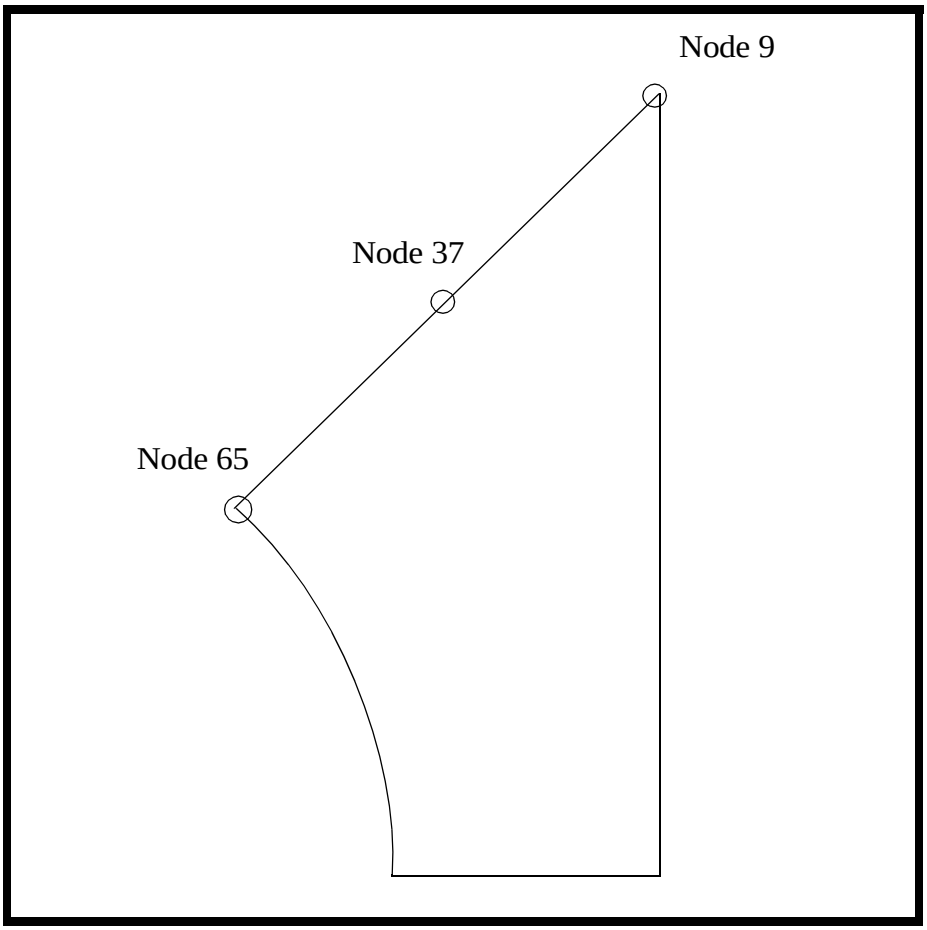
Y:	**Result**
Select Y Result:	**Temperature, Nodal**
X:	**Global Variable**
Variable:	**Time**

Select the **Target Entity** icon

Target Entity:	**Nodes**
Select Nodes:	see Figure 25.8

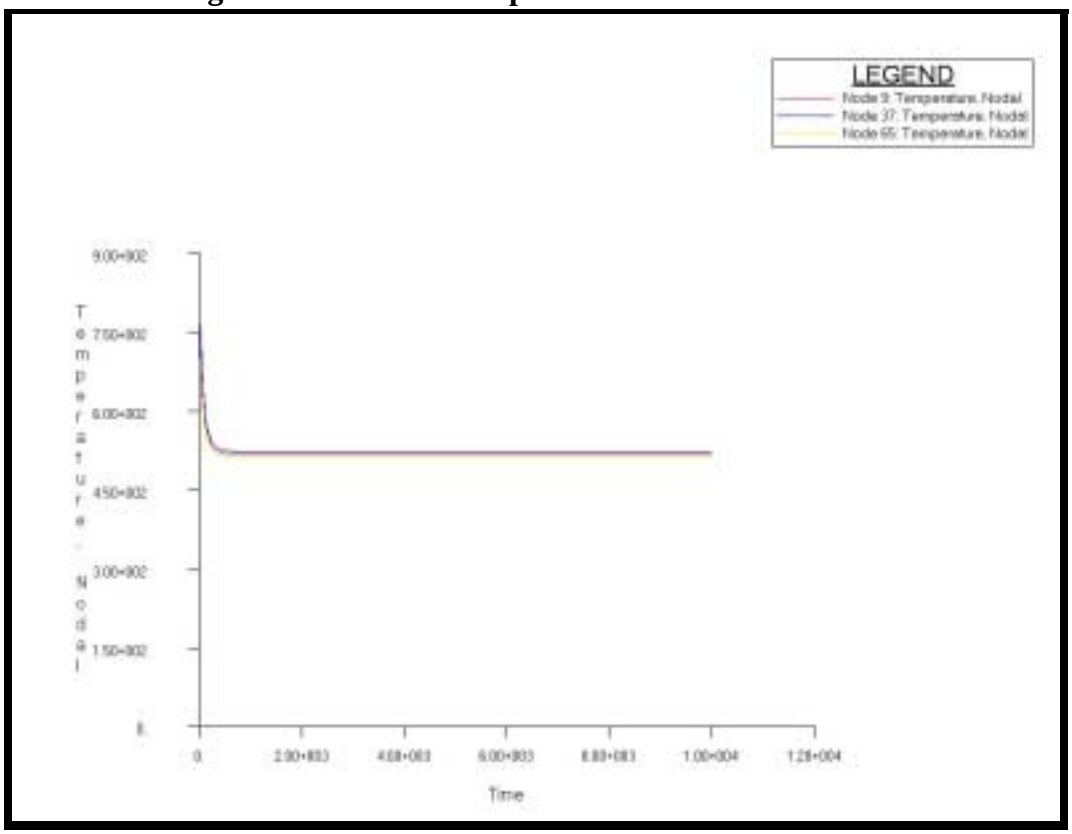
Apply

Figure 25.8 - Nodes to select for XY plot of temp vs. time



Your plot should look like the one in Figure 25.9:

Figure 25.9 - Plot of temp vs. time for three nodes



Close the database and quit PATRAN.

This concludes the exercise