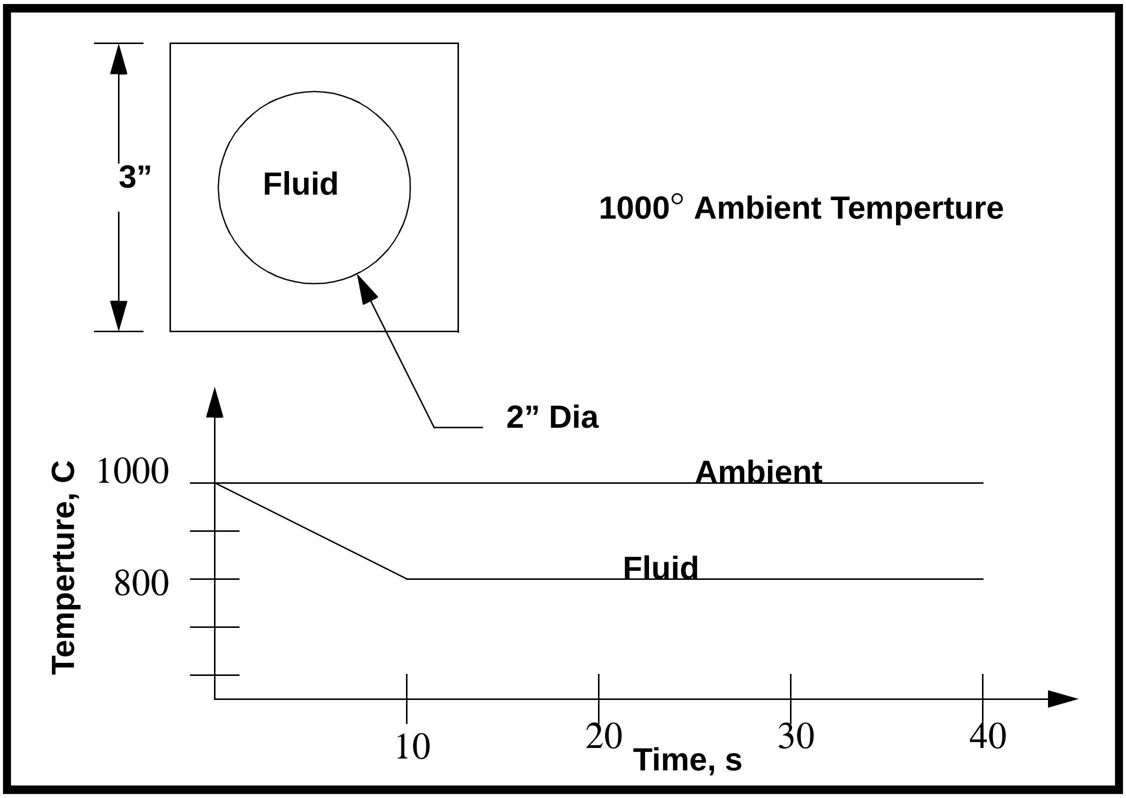
LESSON 16

Transient Heat Transfer Analysis



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 1000°. The temperature of the inner fluid in the model drops from 1000° to 800° 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 element 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/ ADVANCED_FEA**.

Analysis Code: **MSC/ADVANCED_FEA**

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**

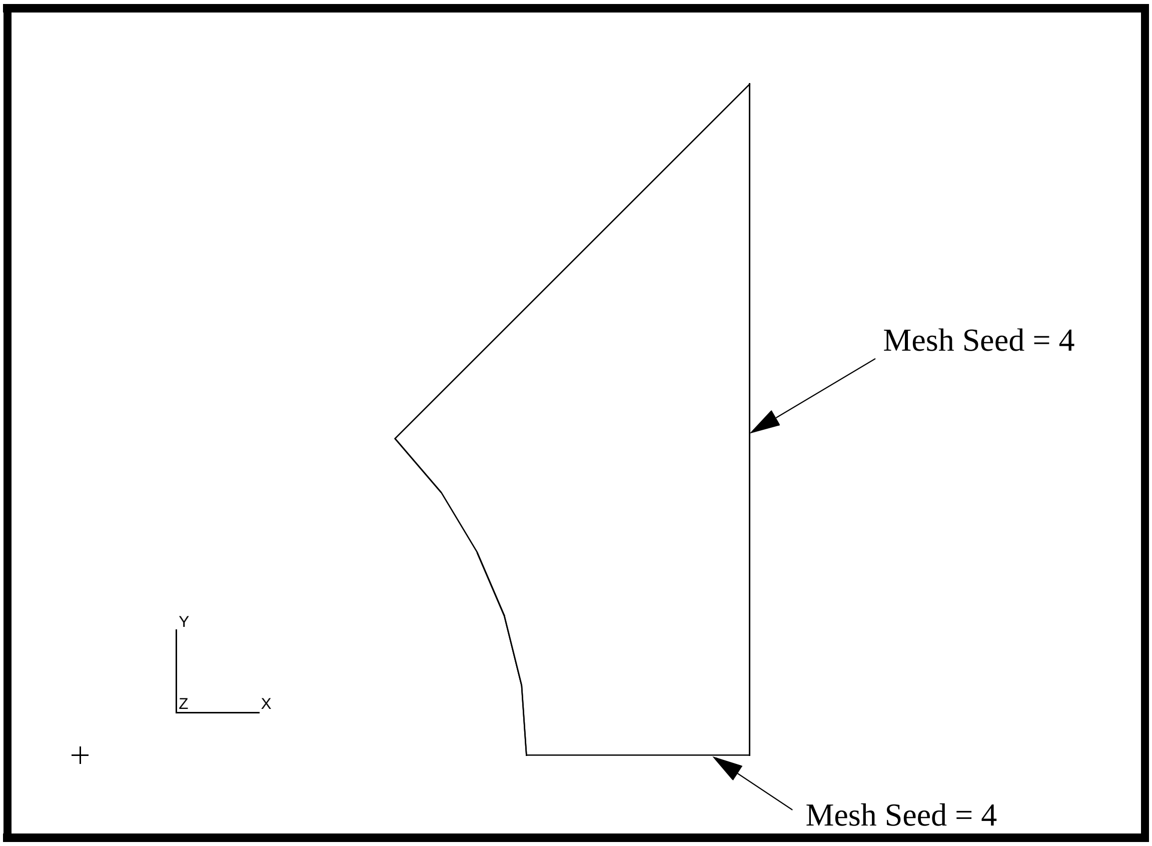
Starting Curve List: **Curve 2**

Ending Curve List: **Curve 1**

Apply

Your model should now look like the one shown in Figure 16.1:

Figure 16.1 - 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.

◆ **Finite Elements**

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

◆ **Number of Elements**

Number:	**4**
Curve List:	see Figure 16.1

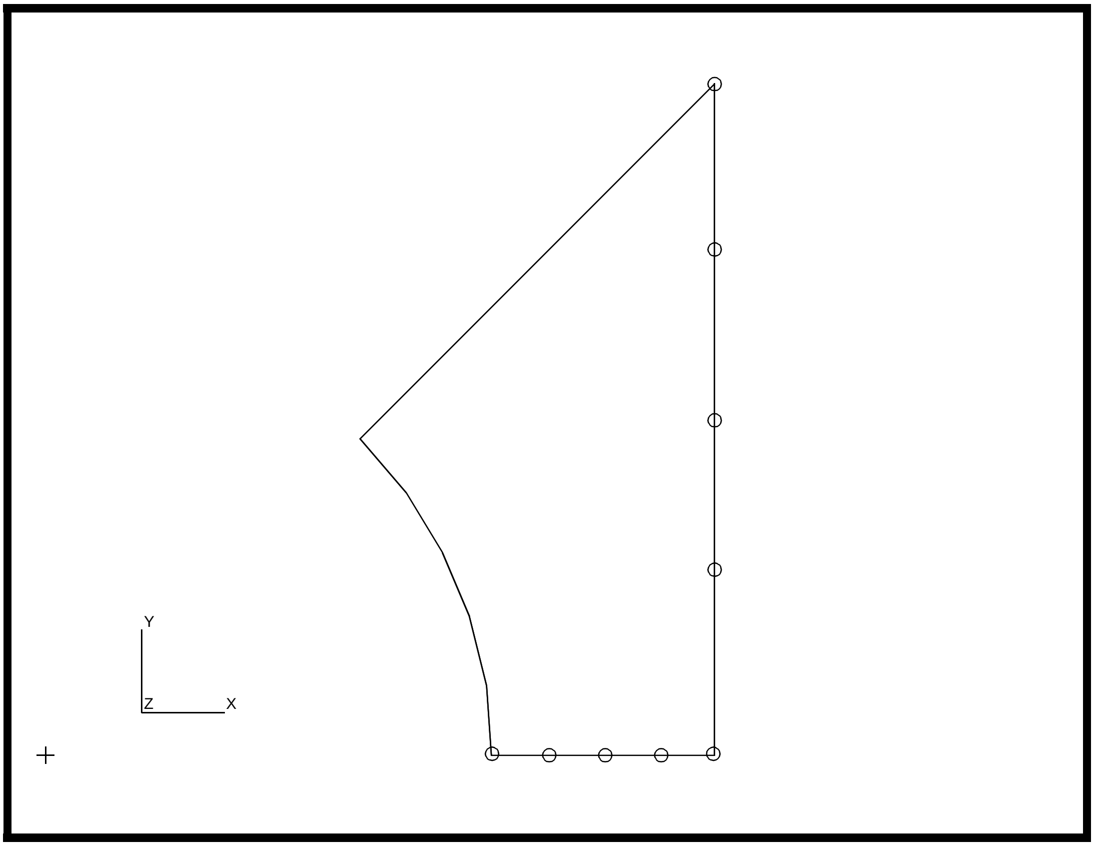
The bottom edge is **Surface 1.1**.

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 16.1

Your model should now appear as shown in Figure 16.2:

Figure 16.2 - 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**.

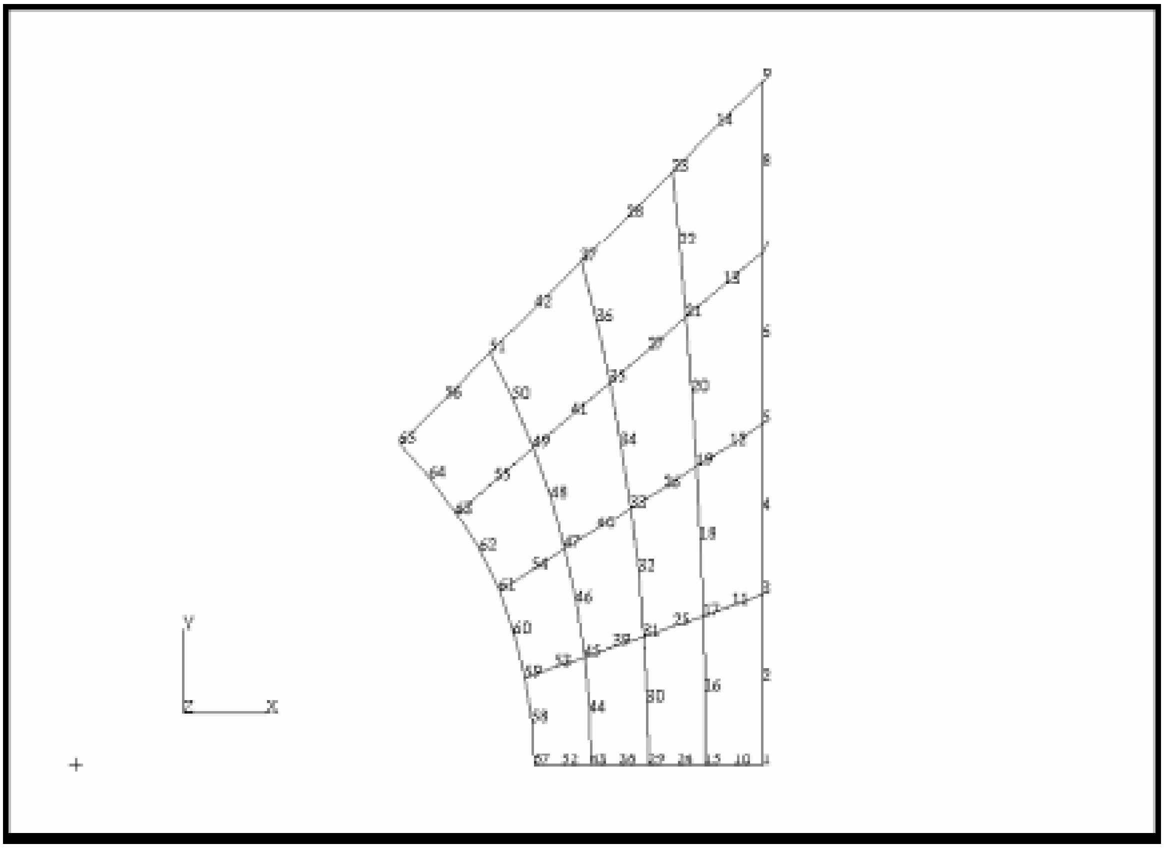
◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad8**
Surface List:	**Surface 1**

Apply

Your model should now appear as shown in Figure 16.3:

Figure 16.3 - 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
Apply	
Cancel	

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	

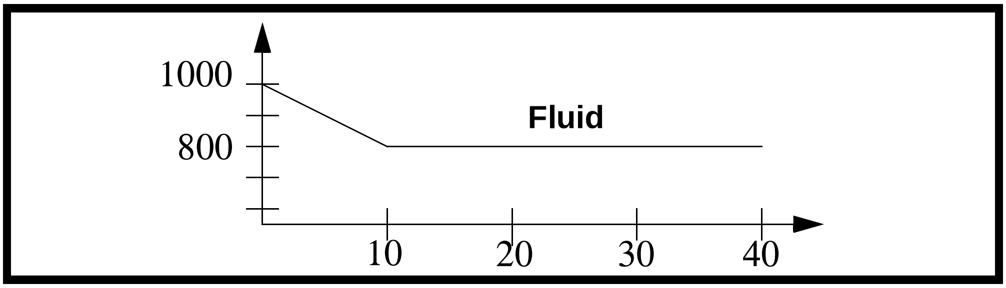
9. Create a time dependent field, which will be applied to the boundary conditions.

◆ **Fields**

Action:	**Create**
Object:	**Non Spacial**
Method:	**Tabular Input**
Field Name:	**inner_temp**
Active Independent Variable:	■ **Time (t)**
Input Data...	

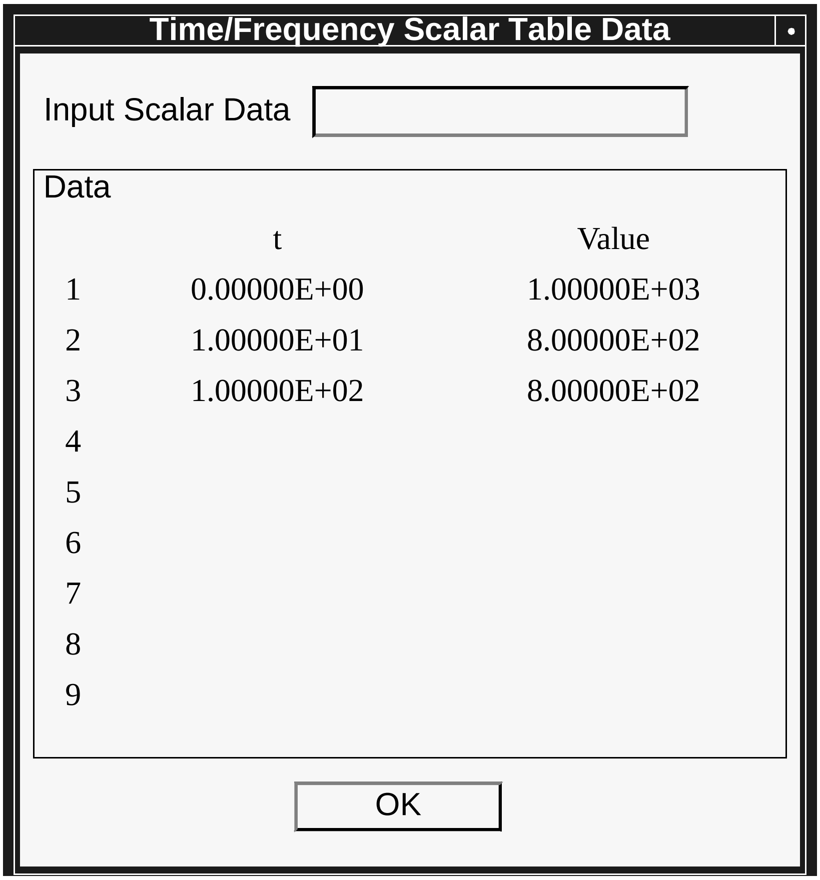
The *Time/Frequency Scalar Table Data* form needs to be filled out as shown in Table 1.

Table 1: Temperature vs. Time data for Inner Temperatures



Time	Temp
0	**1000**
10	**800**
100	**800**

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.



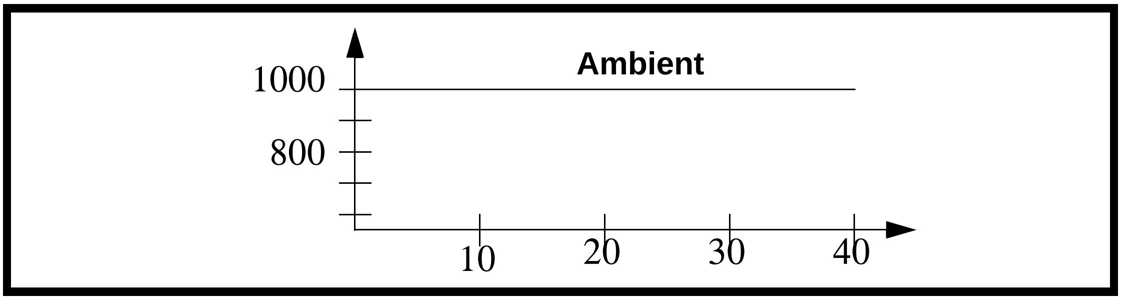
OK

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	**1000**
10	**1000**
100	**1000**

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

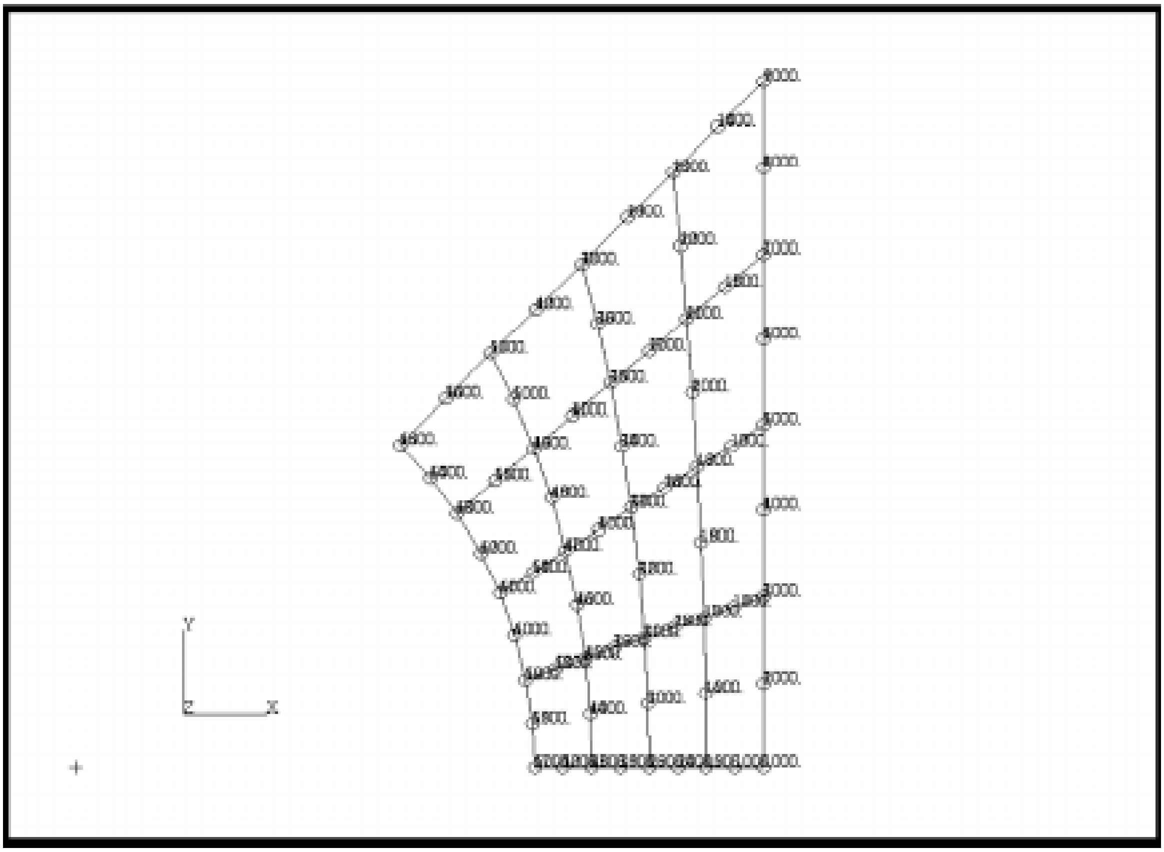
11. Create the loads and boundary conditions for the model.

◆ **Load/BCs**

Action:	**Create**
Object:	**Initial Temperature**
Type:	**Nodal**
New Set Name:	**initial_t**
Input Data...	
Temperature:	**1000**
Time Dependance:	(leave blank)
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 16.4:

Figure 16.4 - 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: **inner_temp**
(next to Ambient Temp)

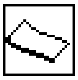
OK

Select Application Region...

Geometry Filter: ◆ **FEM**

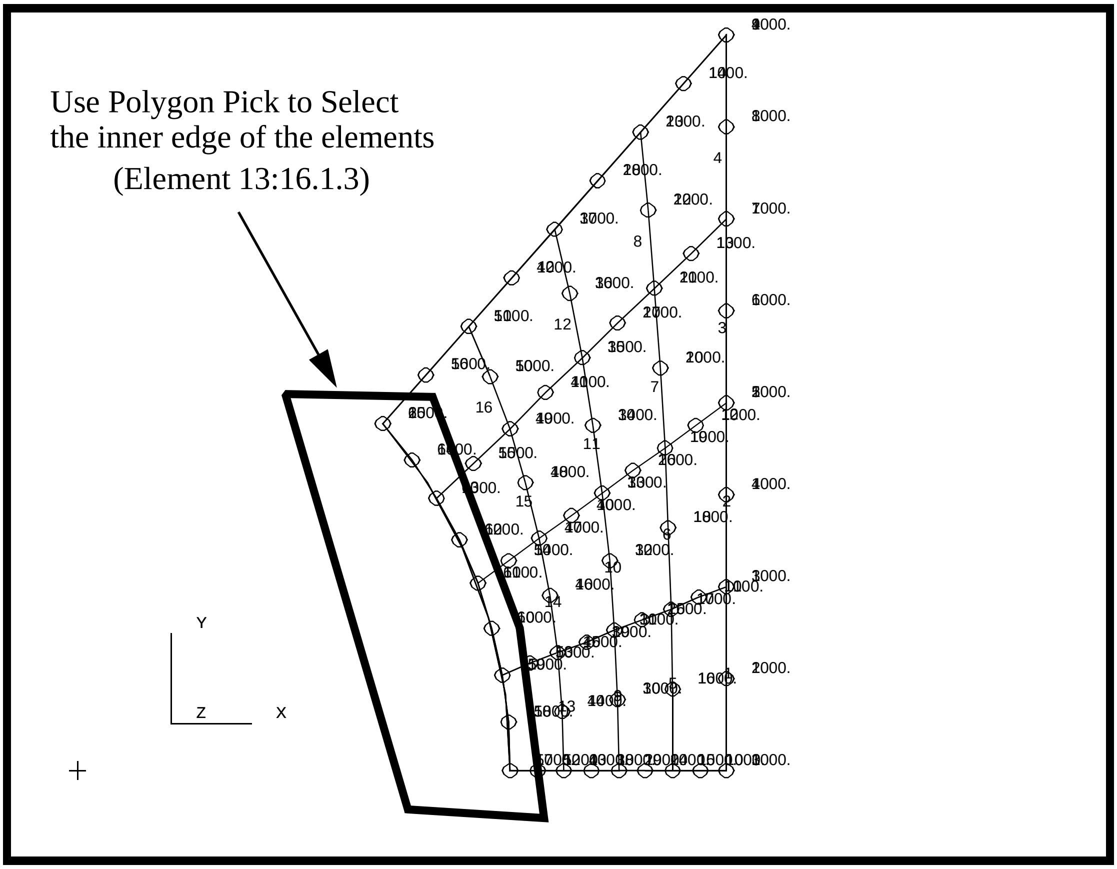
Element Edges: see Figure 16.5

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 16.5

Figure 16.5 - 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

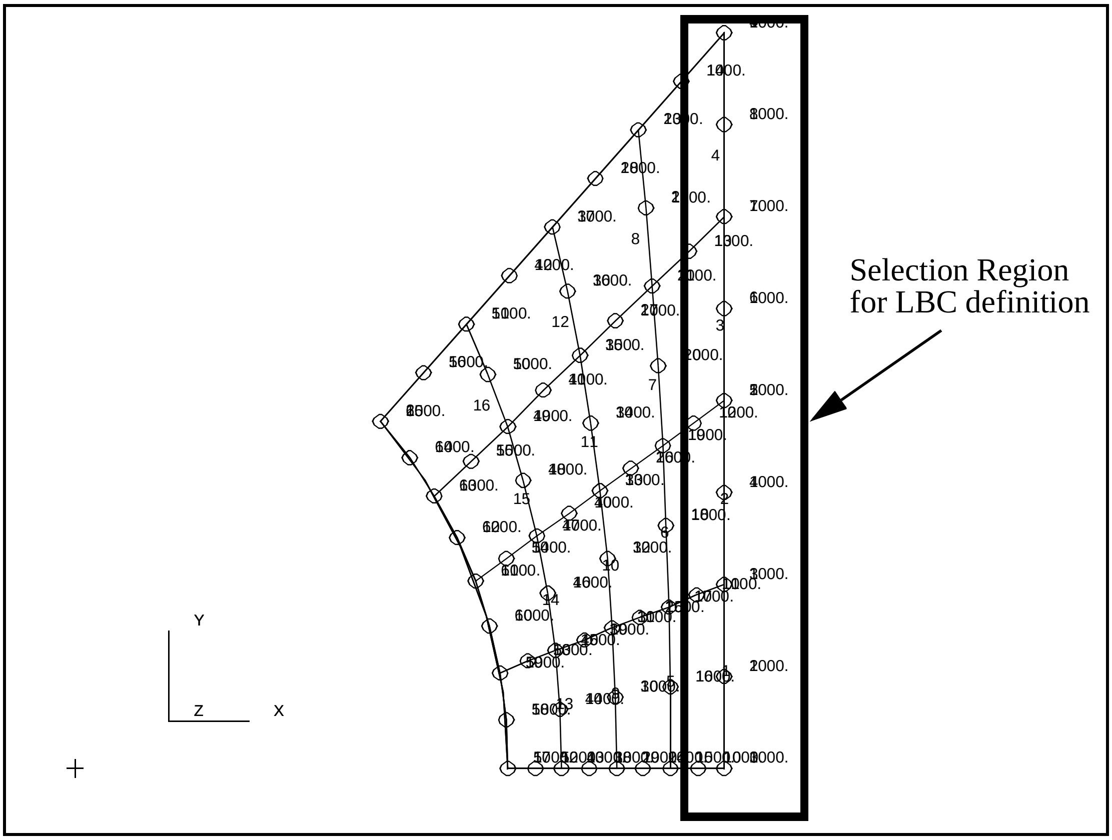
OK

Apply

New Set Name:	**outer_convection**
Input Data...	
Edge Convection:	**0.00001**
Ambient Temp:	**1**
Time Dependence: (next to Ambient Temp)	**outer_temp**
OK	
Select Application Region ...	
Geometry Filter:	◆ **FEM**
Element Edges:	see Figure 16.6
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 16.6 - Elements to select for outer_temp



13. Create the analysis step

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**thermal_flow**
Step Creation...	
Job Step Name:	**transient case**
Solution Type:	**Transient Heat Transfer**
Solution Parameters...	
Max No. of Increments:	**100**
Max Allowable Temp Change:	**20**
Delta-T:	**2**

Minimum Delta-T: **1**

Maximum Delta-T: **1000**

Time Duration of Step: **10000**

Stopping Condition: **Steady State**

Max Temp Change Rate: **0.01**

OK

Select Load Cases...

transient_load_case

OK

Apply

Cancel

Step Selection...

Selected Job Steps: **transient case**

Apply

Apply

The analysis job will take (on average) about 5 minutes to run. When the job is done there will be a results file titled **thermal_flow.fil** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at **thermal_flow.msg** and **thermal_flow.sta** with the ***more*** command. Also, you may use ***ps -ef | grep afea*** and ***tail -lf thermal_flow.sta*** to monitor the status.

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

◆ **Analysis**

Action: **Read Results**

Select Results File...

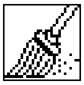
Available Files: **thermal_flow.fil**

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

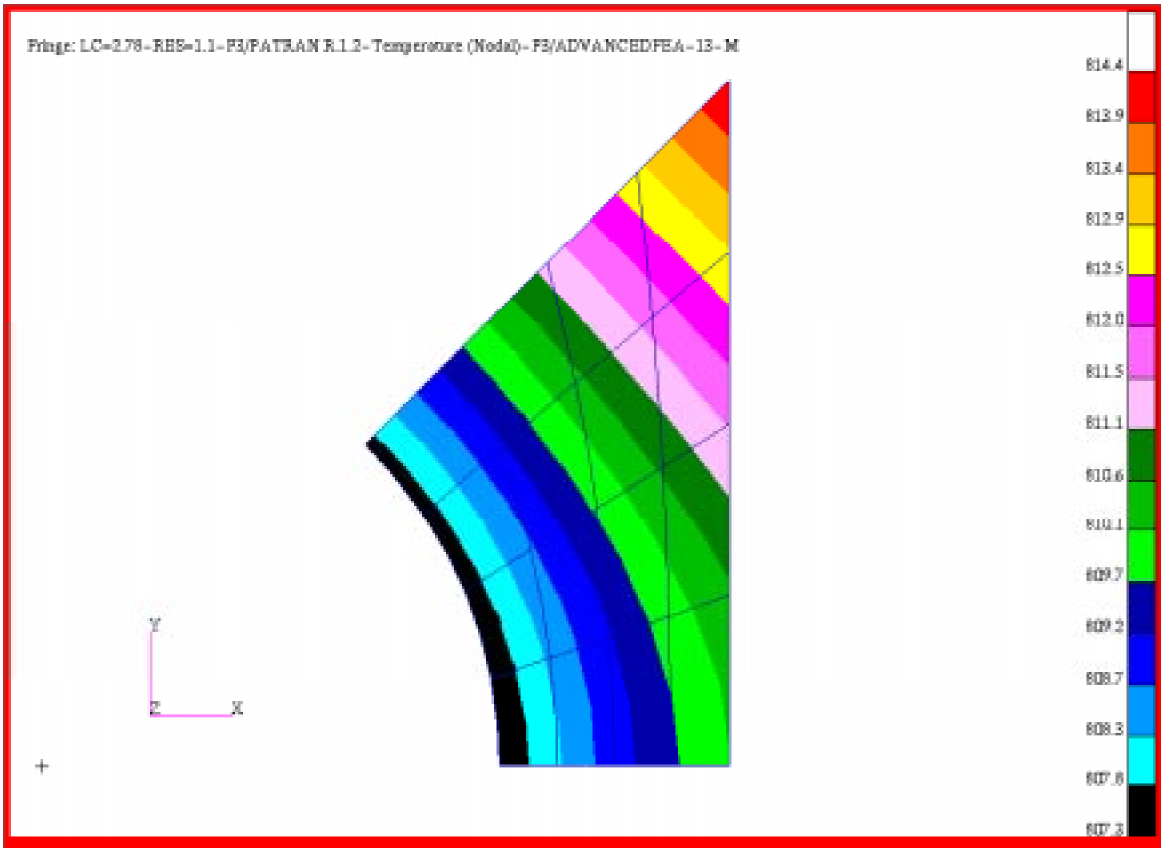
◆ **Results**

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 16.7:

Figure 16.7 - Resulting steady state temperature distribution



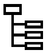
16. Plot the temperature as a function of time.

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**

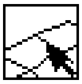
Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

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

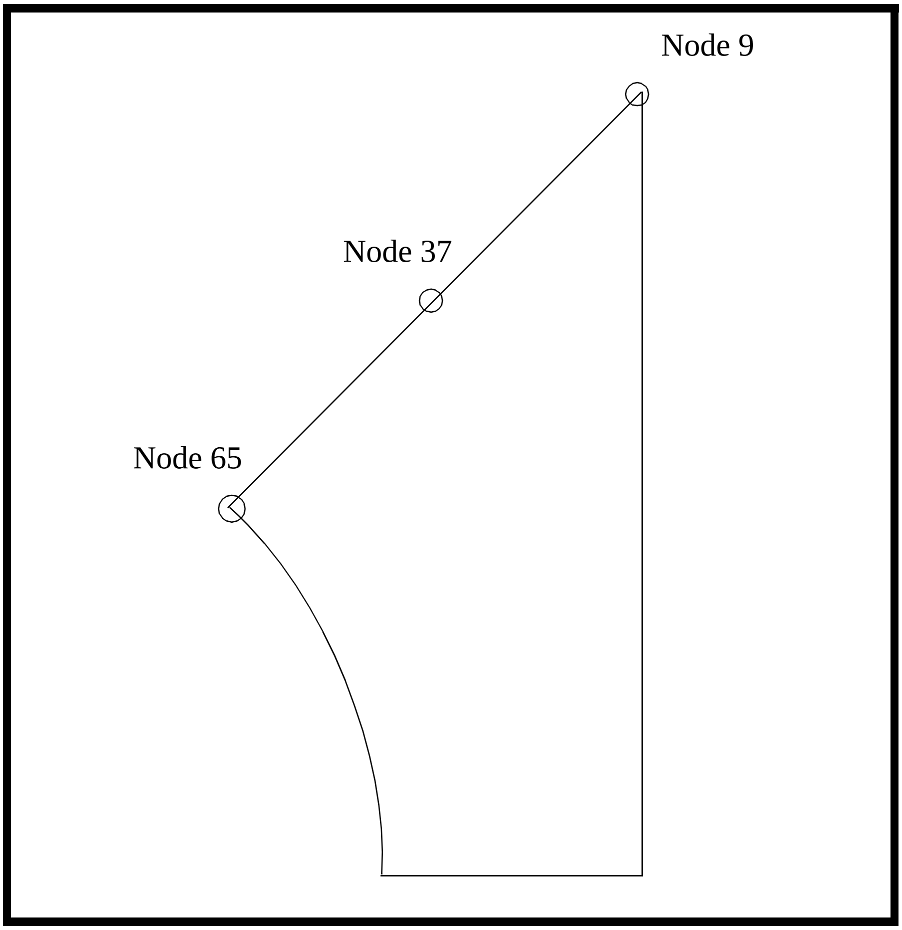
Select Result Case:	**transient_case**
Filter Method	**All**
Filter	
Apply	
Close	
Y:	**Result**
Select Y Result:	**Temperature (Nodal)**
X:	**Global Variable**
Variable:	**Time**

Select the **Target Entity** icon



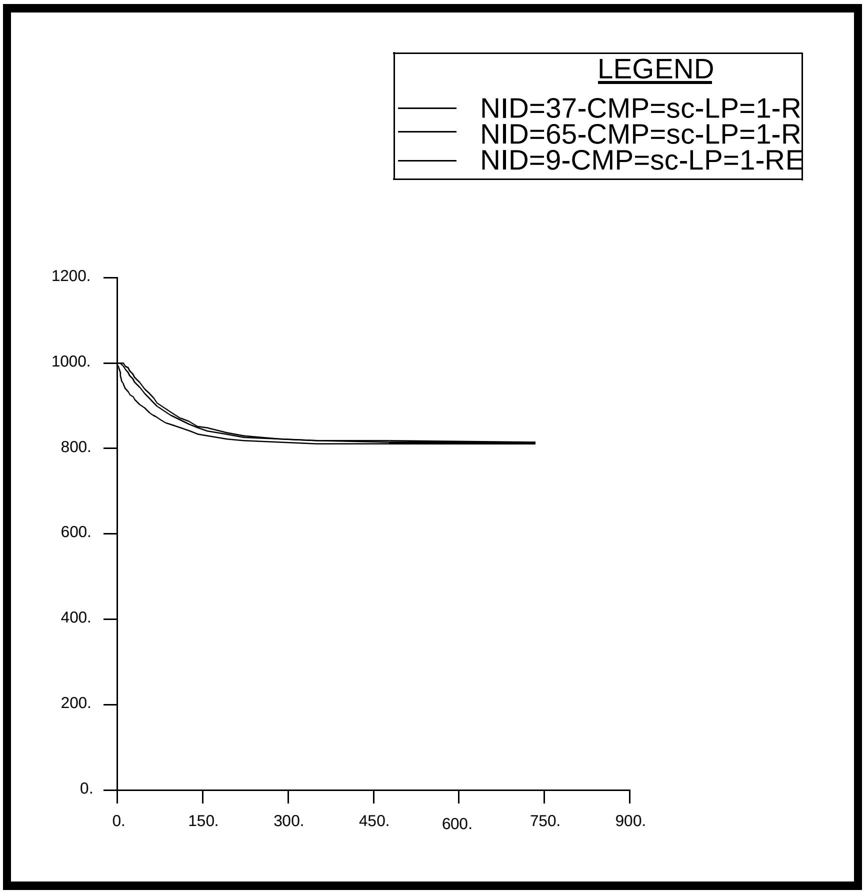
Target Entity:	**Nodes**
Select Nodes:	see Figure 16.8
Apply	

Figure 16.8 - Nodes to select for XY plot of temp vs. time



Your plot should look like the one in Figure 16.9:

Figure 16.9 - Plot of temp vs. time for three nodes



Close the database and quit PATRAN.

This concludes the exercise