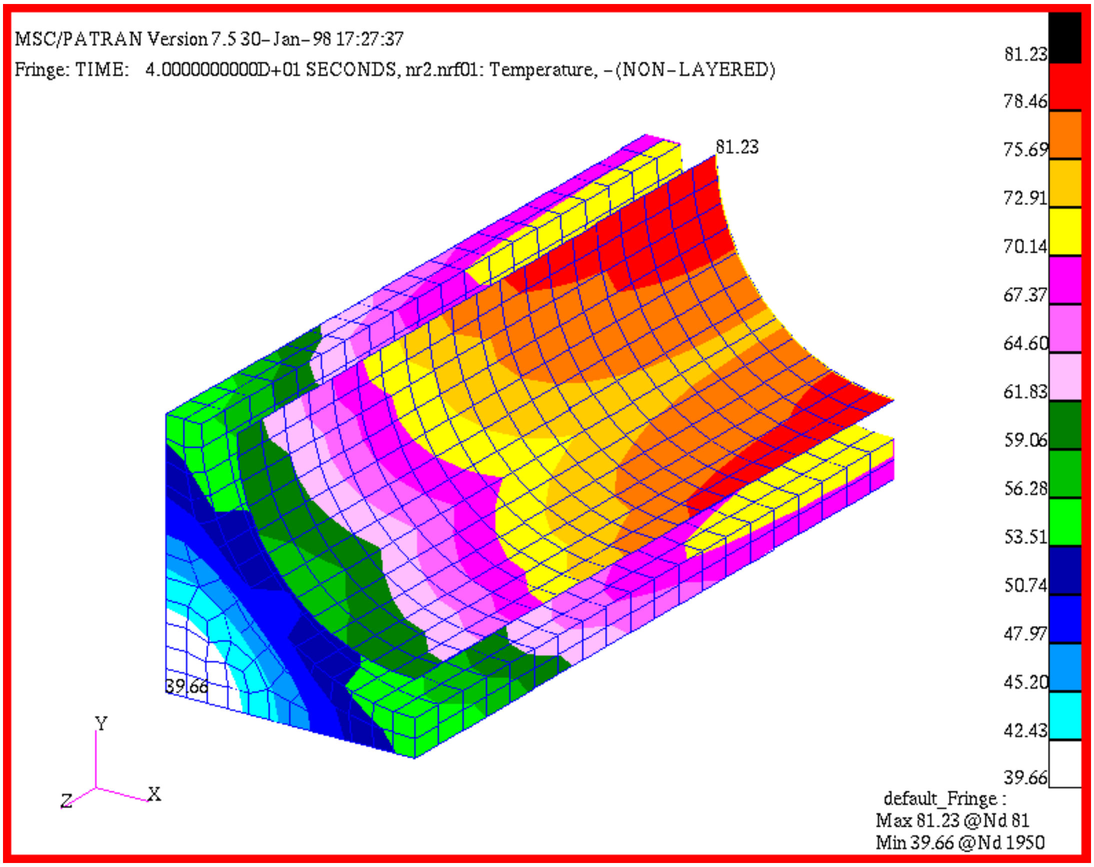
Exercise 16

A Concentric Tube, Counterflow Heat Exchanger



Objective:

- Demonstrate MSC/THERMAL capabilities for gap convection problems.
- Practice basic modeling skills using MSC/PATRAN.

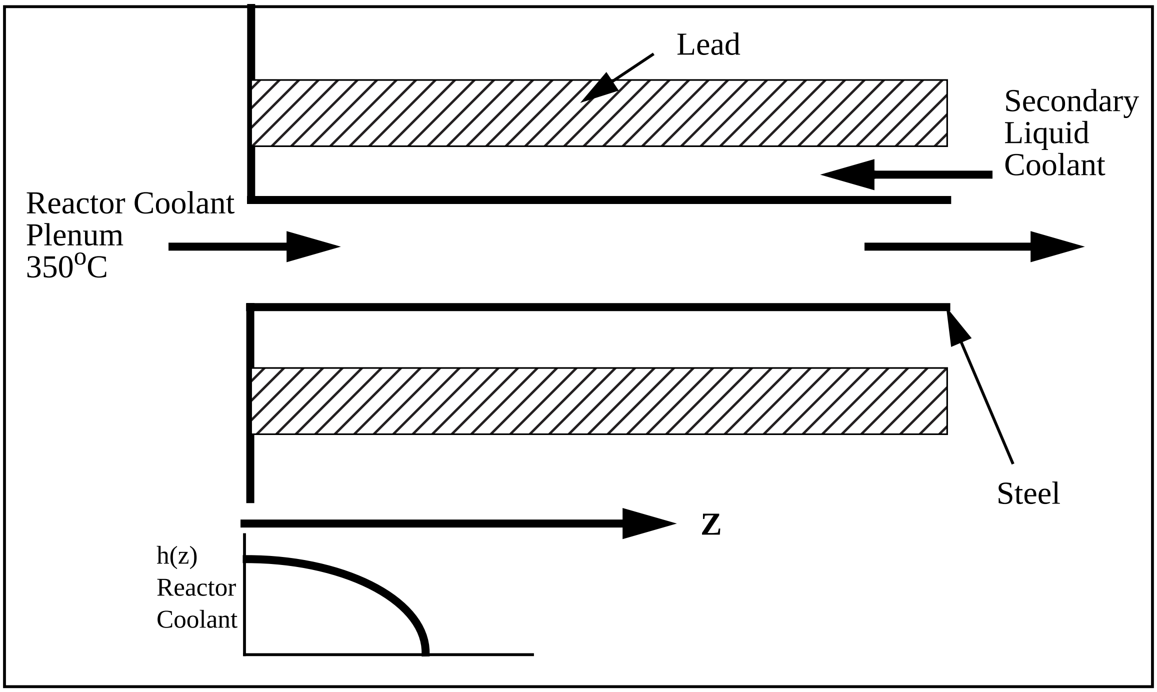
Model Description:

In this exercise you will create a simple 3D model representing a concentric tube, counterflow heat exchanger. Owing to symmetry considerations only one-fourth of the heat exchanger configuration needs to be modeled.

A team of university students is considering a makeshift heat exchanger, to cool and discard gaseous coolant from a small reactor. The heat exchanger is designed to “begin” at the reactor coolant plenum. In the event of an emergency, a safety valve would open to draw the coolant from the plenum into the exchanger (a process which will require approximately 60 seconds to complete). A secondary liquid coolant would then be used to decrease the temperature of the reactor coolant, before the reactor coolant enters an complex filtration process.

The existing reactor coolant system is comprised of steel; and, the material proposed to contain the secondary coolant flow is simple lead. At the junction between the plenum and the heat exchanger, the gaseous fluid would exhibit a high mass flow rate at 350^{o}C; the entry length variation of the convection coefficient between the steel and the gas is expected to follow: h = 200-13000*z^3 w/m^2K (where z is the distance from the plenum). The liquid coolant will flow between the steel coolant tube and its own lead housing, will be fully developed and is expected to exhibit a high convection coefficient (3000 w/m^2K).

The students prime concern with the design is the determination of the maximum temperature that the lead tube will exhibit after 60s of use.



Exercise Overview:

- Create a new database named **exercise_16.db.**
- Use the **Create** and **Edit** actions on the Geometry form to constuct a 2D representation of the heat exchanger.
- Mesh the 2D geometry created in the previous step and use **Sweep/Element/Extrude** to develop he 3D FEM model.
- Create 4 nodes to represent the spatial variation of the convection coefficient of the reactor coolant over the entry length.
- Apply the appropriate Element Properties to the FEM model: Quad4's - Steel MID 353; Hex8's - Lead MID 21.
- **Crate/Spatial/PCL Function** to define the variation of the convection coefficient of the reactor coolant flow in the streamwise direction.
- Apply a fixed temperature of 350^{o}C to the nodes representing gasous coolant.
- Create 2 **Between Regions** Convection Boundary Condtions.
- Perform a Transient Analysis for **60s** assuming a global initial temp of **25oC**.
- Prepare and submit the model for analysis.
- Read results file and plot results.
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_16.db**.

Within your window environment change directories to a convenient working directory. Run MSC/PATRAN by typing **p3** in your xterm window.

Next, select **File** from the *Top Menu Bar* and select **New…** from the drop-down menu. Assign the name exercise_16.db to the new database by clicking in the *New Database Name* box and entering **exercise_16**.

Select **OK** to create the new database.

File	
New	
New Database Name	**exercise_16**
OK	

MSC/PATRAN will open a Viewport and change various *Control Panel* selections from a ghosted appearance to a bold format. When the New Model Preferences form appears on your screen, set the *Approximate Maximum Model Dimension* to **0.07,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences form.

Approximate Maximum Model Dimension	**0.07**
Analysis Code	**MSC/THERMAL**
OK	

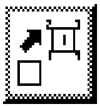
2. Use the **Create** and **Edit** actions on the Geometry form to constuct a 2D representation of the heat exchanger.

Create 2D heat exchanger

Select the **Geometry** *Applications Radio Button*. Create a surface using the following *Action*, *Object*, and *Method*. Click in the appropriate list boxes to edit the default values and change them to values listed below.

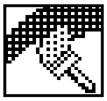
◆ Geometry	
Create/Curve/2D ArcAngles	
Radius	**0.05**
Starting Angle	**180**
End Angle	**270**
Apply	
Radius	**0.06**
Apply	
Create/Surface/XYZ	
Vector Coordinates List	**<-0.07 -0.07 0>**
Apply	

Turn on the label by using the Tool Bar *Show Label* icon.

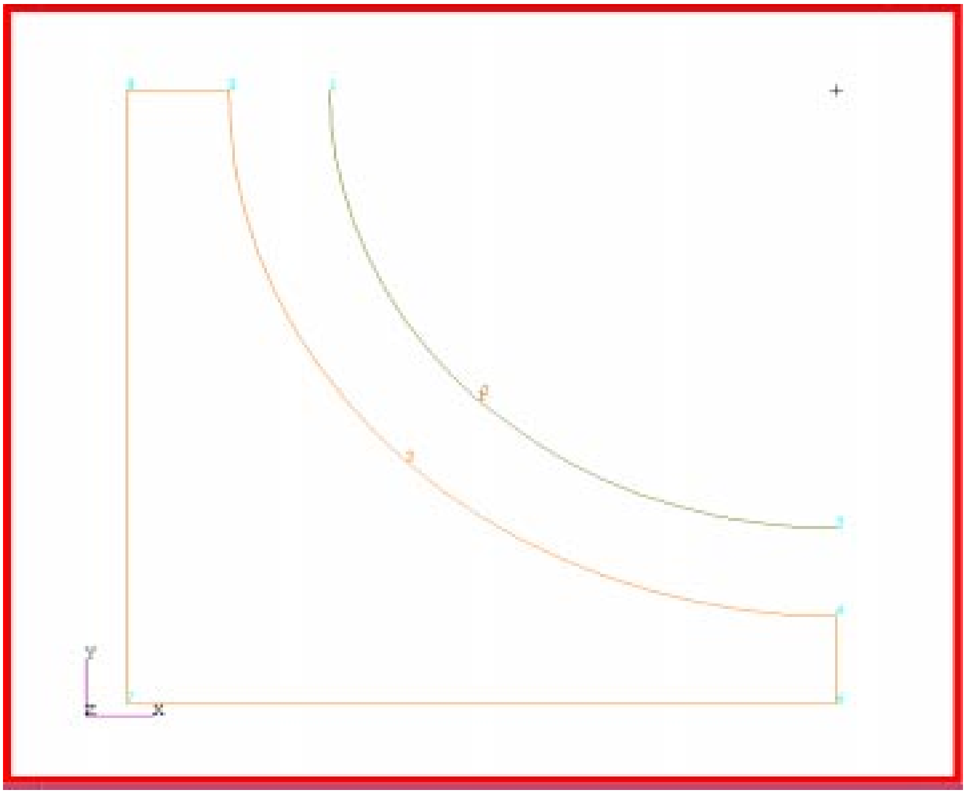


Edit/Surface/Break	
Option:	**Curve**
Surface List:	**Surface 1**
Break Curve List	**Curve 2**
Message! (delete original surface)	**Yes**
Delete/Any	
Geometric Entity List	**Surface 3**
Apply	

At any time during this exercise, use the Tool Bar *Refresh graphics* icon to refresh the graphics when necessary.



The resulting model is shown below.



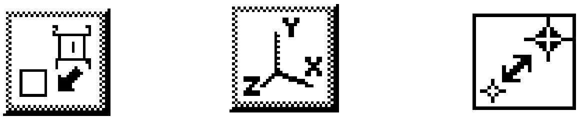
Mesh the surfaces

3. Mesh the surface with quad4 elements. Use the Paver and a global edge length of 0.006.

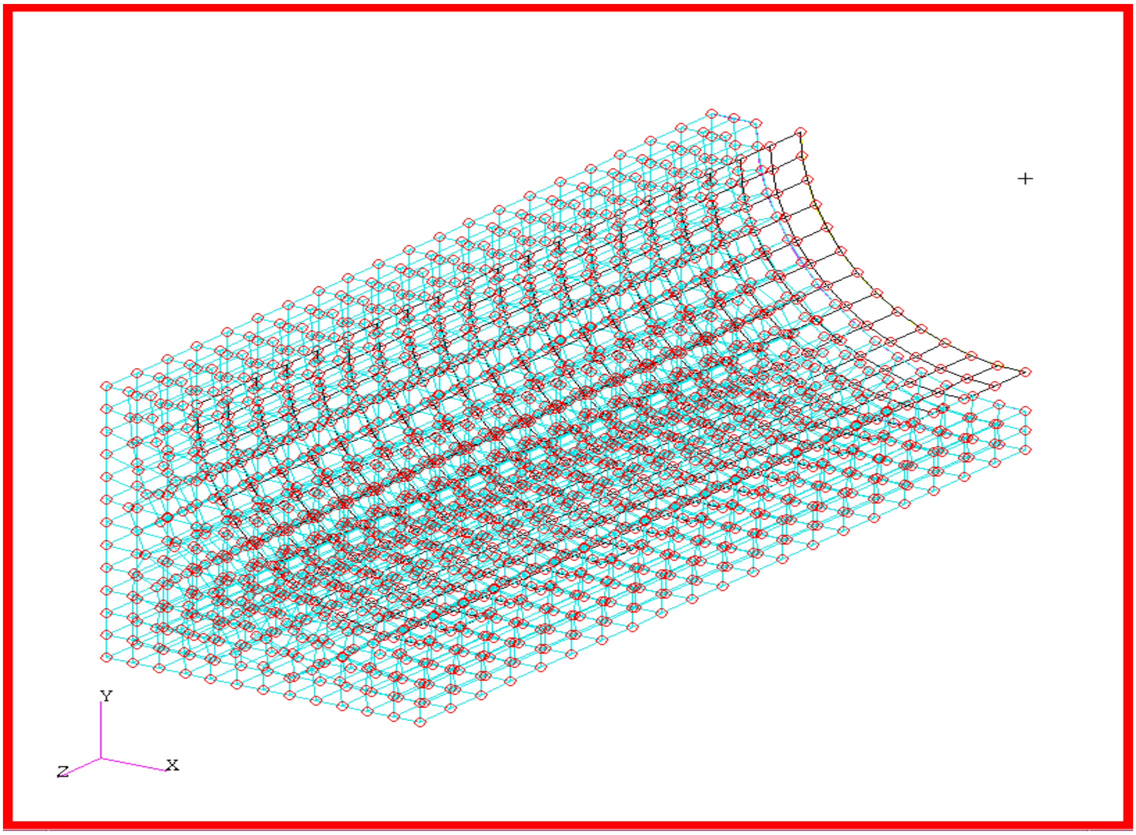
Select the **Finite Elements** *Applications Radio Button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to 0.006 and select Surface 2for inclusion in the *Surface List*.

◆ Finite Elements	
Create/Mesh/Surface	
Global Edge Length	**0.006**
Mesher	**◆ Paver**
Surface List	**Surface 2**
Apply	
Create/Mesh/Curve	
Global Edge Length	**0.006**
Curve List	**Curve 1**
Apply	
Sweep/Element/Extrude	
Mesh Control ...	
Number =	**20**
OK	
Extrude Distance	**0.2**
◆ Delete Original Elements	
Base Entity List	**<type in “Elm 1:#”>**
Apply	

Use the Tool Bar *Hide Label* icon and*Iso 1 View* to get a clearer view of the graphics. Also, increase the size of the nodes by using the Tool Bar *Node Size* icon so the four boundary nodes will be more visible.



The display should now appear as shown below.



Create ambient nodes

4. Create 4 nodes to effect a spatial variation of the convection coefficient magnitude to represent a developing flow.

Using the Finite Elements form, create 4 boundary nodes which are not associated with geometry. The nodes are numbered **9996 to 9999**. Click in the appropriate list boxes to edit the default values and change them to values listed below.

Create/Node/Edit

Node ID List	**9996**
☐ **Associate with Geometry**	
Node Location List	**[0 0 0]**
Apply	
Node ID List	**9997**
Node Location List	**[0 0 0.05]**
Apply	

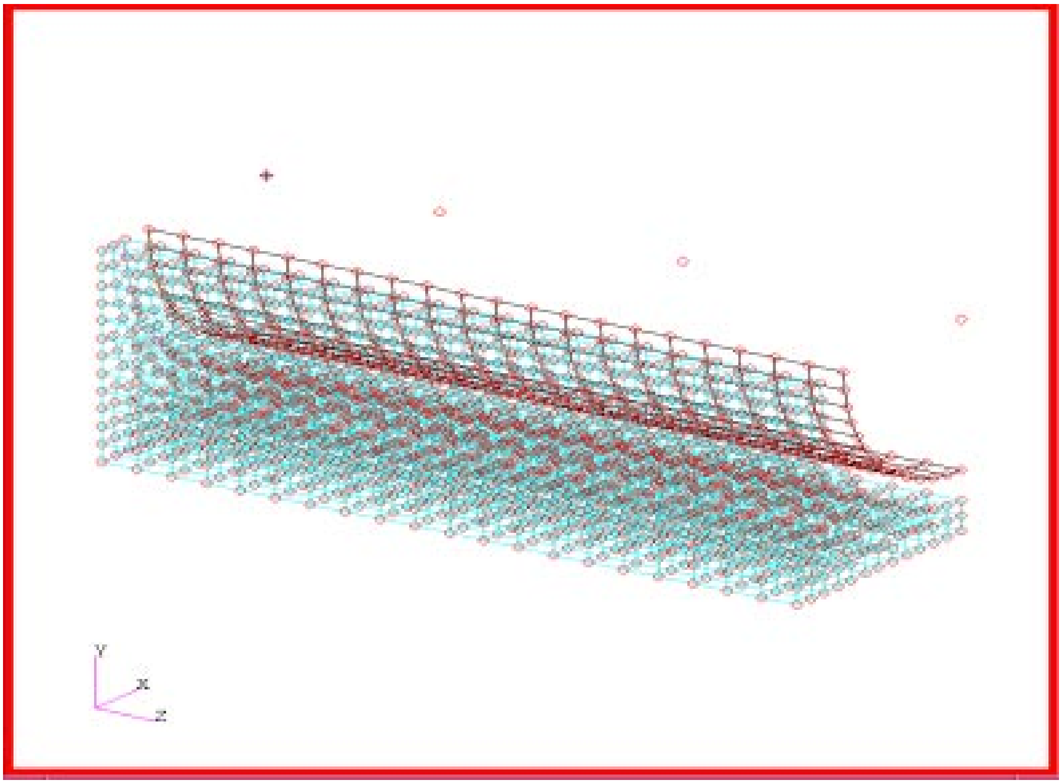
Node ID List	**9998**
Node Location List	**[0 0 0.12]**

Apply

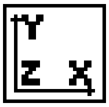
Node ID List	**9999**
Node Location List	**[0 0 0.20]**

Apply

Rotate the display to verify the locations of the new nodes. Using the *Iso 2 View*, the model should appear as shown below.



Revert the display back to the *Front View* for the next section.



Apply element properties

5. Apply two element properties to the elements using the material property MID's **353 and 21**.

In a typical modelling sequence the Materials *Application radio button* would be the next stop to define a material for application in Element Properties. However, MSC/THERMAL includes a Material Properties Database which contains 970 materials with thermal properties already defined. We will use this database to facilitate the analysis.

Select the **Properties** *Applications radio button*. Set the *Action*, *Dimension*, and *Method* to **Create/2D/Shell**. Enter *Property Set Name* **Steel**. Select the *Input Properties...* box. In the Input Properties form, click in the *Material Name* box and enter **353**, and thickness of 0.005m. Select **OK** to close the form. First, select 2D element from the Select Menu Form. Click in the *Select Members* box and drag a rectangle around the model in the viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties

Create/2D/Shell

Property Set Name **Steel**

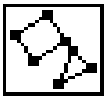
Input Properties...

Material Name **353**

Shell Corner Thickness **0.005**

Ok

Select Members/*Select Menu* **<2D *Element* icon, second from top>**



Select Members **<select all entities, (Elm 1316:1575)>**

Add

Apply

Perform the same steps for outer lead portion, using *Action*, *Dimension*, and*Method* to **Create/3D/Thermal 3D Solid**, **Lead** for the *Property Set Name*, **21** for the *Material Name*.

◆ Properties

Create/3D/Thermal 3D Solid

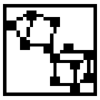
Property Set Name **Lead**

Input Properties...

Material Name — **21**

Ok

Select Members/*Select Menu* — **<*Solid Element* icon, second from top>**



Select Members — **<select all entities, (Elm 76:1315)>**

Add

Apply

Create Function

6. **Create/Spatial/PCL Function** to define the variation of the convection coefficient of the reactor coolant flow in the stream direction.

◆ Fields

Create/Spatial/PCL Function

Field Name — **convection_f_of_z**

Scalar Function (‘X, ‘Y, ‘Z) — **200-(13000*’Z*’Z*’Z)**

Apply

Show

Select Field To Show — **convection_f_of_z**

Specify Range...

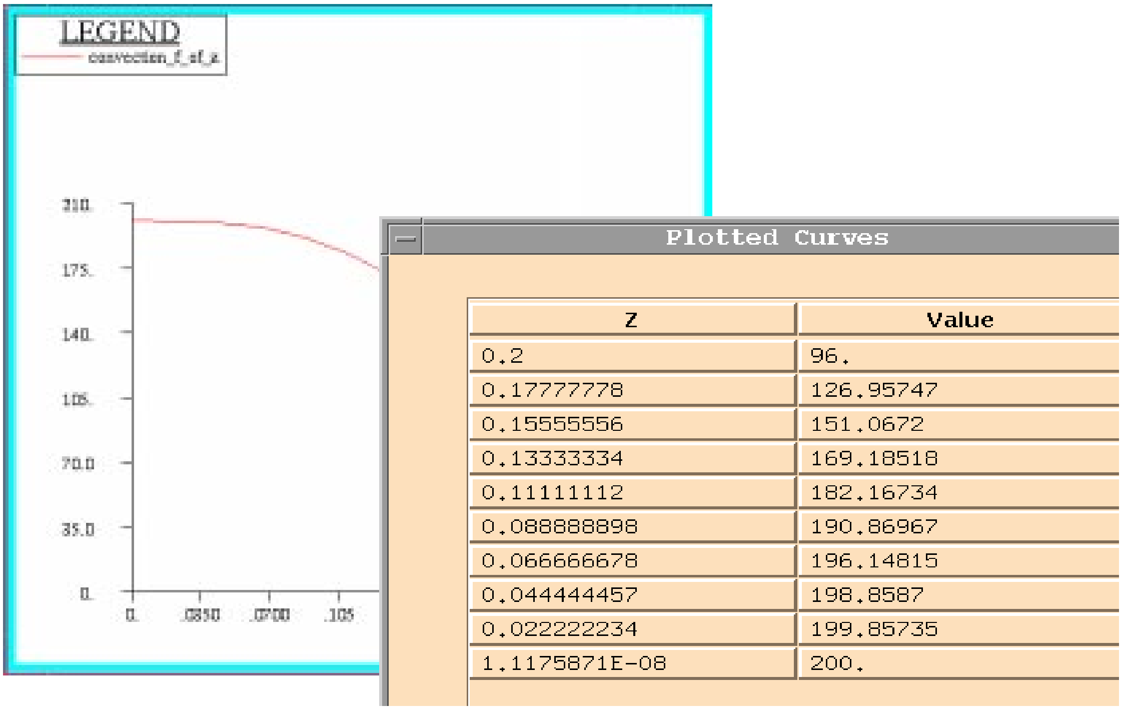
Maximum — **0.2**

No. of Points — **10**

OK

Apply

The XY Result Window and Table should appear as shown below.



Z	Value
0.2	96.
0.17777778	126.95747
0.15555556	151.0672
0.13333334	169.18518
0.11111112	182.16734
0.088888898	190.86967
0.066666678	196.14815
0.044444457	198.8587
0.022222234	199.85735
1.1175871E-08	200.

Close the window and table by clicking on the Unpost Current XYWindow.

Cancel

Unpost Current XYWindow

7. Apply a fixed temperature of 350°C to the nodes representing gasous coolant.

Apply boundary conditions

Begin applying boundary conditions. Select the **Load/BCs** *Applications Radio Button*. Create a fixed **350°C** nodal boundary temperature named **interior_flow**. In the Input Data form define the fixed temperature. In the Select Applications Region form pick **Node 9996 to 9999** located in the upper right corner of the display screen.

◆ Load/BCs

Create/Temperature/Nodal

Option:	**Fixed**
New Set Name	**interior_flow**

Input Data...

Fixed Temperature	**350.0**

OK

Select Application Region...	
Geometry Filter	◆ **FEM**
Select Nodes	**<select Node 9996 to 9999>**
Add	
OK	
Apply	

Create a between regions convection flow named **inner_flow** with the data show as follow.

Create/Convection/Element Uniform	
Option:	**Between Regions**
New Set Name	**inner_flow**
Target Element Type:	**2D**
Region 2:	**Nodal**
Input Data...	
Convection Coefficient	**<select convection_f_of_z in the Spatial FIelds Box>**
OK	
Select Application Region...	
Geometry Filter	◆ **FEM**
Order:	**Closest Approach**
Select 2D Elements or Edges	**<select all entities, (Elm 1211:1470)>**
Add	
Application Region	**<select the bottom one>** ◆ **Active List**
Select Nodes	**<select Node 9996 to 9999 in the upper right corner of the display.>**
Add	
OK	

Apply

A blue line should verify the newly defined association.

Before creating the next convection condition, make sure that the polygon picking preference is set at Enclose entire entity.

Preference

Picking...

Rectangle/Polygon Picking	◆**Enclose entire entity**

Close

Now, construct the next Between Regions Convection condition called **outer_flow** as follow.

Create/Convection/Element
Uniform

Option:	**Between Regions**
New Set Name	**outer_flow**
Target Element Type:	**3D**
Region 2:	**2D**

Input Data...

Convection Coefficient	**3000**

OK

Select Application Region...

Geometry Filter	◆ **FEM**
Order:	**Closest Approach**
Application Region	**<select the top one>** ◆ **Active List**
Select 3D Element Faces	**<Use the <CNTL> key and the left mouse button to create a polygon selecting only those lead element which contact the outer fluid flow>**

Add

Application Region	**<select the bottom one>** ◆ **Active List**

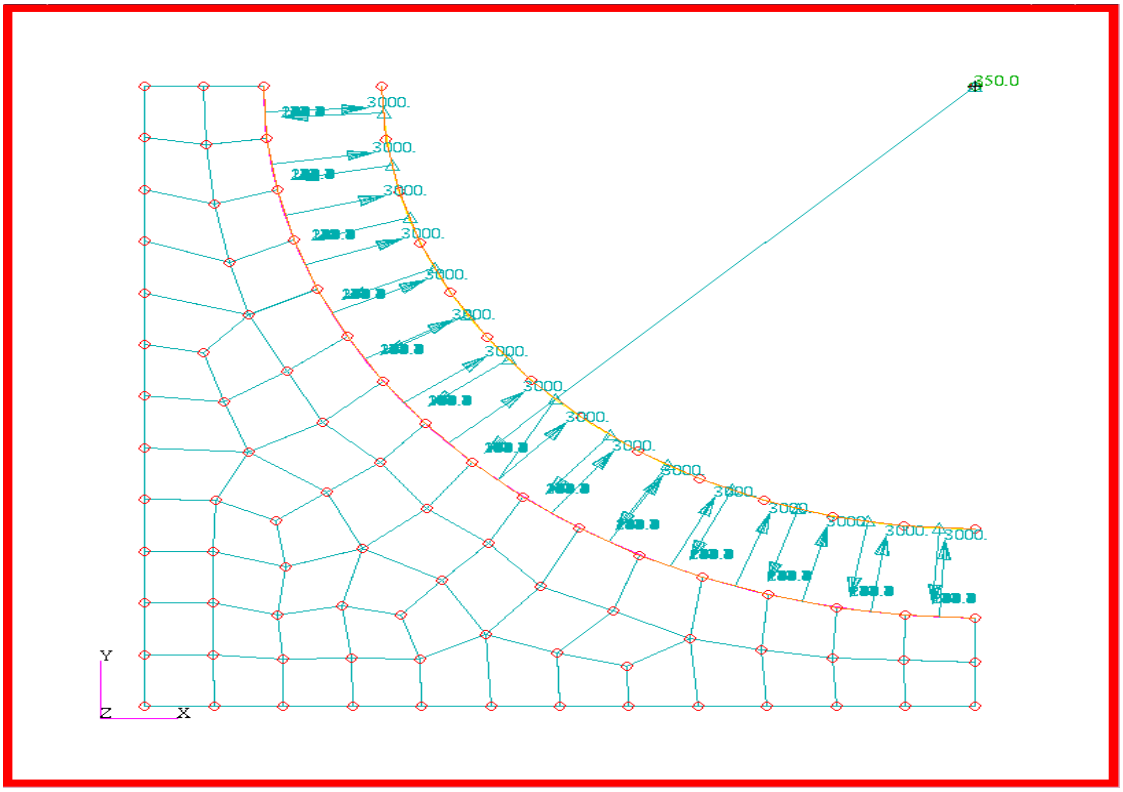
Select 2D Elements or Edges	**<select all entities, (Elm 1211:1470).>**

Add

OK

Apply

The display should now appear as shown below.



8. Prepare and submit the model for analysis.

Prepare and run analysis

Select the **Analysis** *Applications Radio Button* to prepare the analysis. Select the parameter forms reviewing and changing the settings as shown below. The analysis is submitted by selecting **Apply** in the Analysis form.

◆ Analysis

Analyze/Full Model/Full Run

Solution Type...

Select Thermal Solution	**◆ 1, Transient Run**

OK

Solution Parameters...

Calculation Temperature Scale	◆ **Celsius**

Run Control Parameters...

Stop Time =	**60**
Initial Temperature =	**25.0**

OK

OK

Output Requests...

Units Scale for Output Temperatures	◆ **Celsius**

Print Interval Control...

Initial Print Interval =	**20.0**

OK

OK

Apply

When the *Heartbeat* returns to green open a UNIX shell to monitor the progress of your job. Recall that the tools for monitoring your job are as follows:

1) **cd** - to change the current directory to the *Job Name* subdirectory,

2) **tail - f patq.msg.01** - to monitor the generation of the input deck,

3) **qstat l** - to link the status file from each time step together and,

4) **qstat c** - to monitor the solver progress.

9. Read results file and plot results.

Read and plot results

From within MCS/PATRAN the only indication that the analysis has successfully finished is the existence of an **nrX.nrf.01** results file in a subdirectory one level below your working directory.

P3 was initiated from a working directory which contained the **exercise_16.db** database. Applying the analysis created a new subdirectory with the same name as the *Job Name*; **exercise_016**/. By using **Read Result** in the Analysis form and Selecting **Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of a results file.

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories **<path>/exercise_16**

Filter

Available Files **nr2.nrf.01**

OK

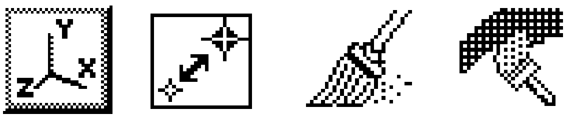
Select Rslt Template File...

Files **pthermal_1_nodal.res_tmpl**

OK

Apply

Change the display to the Iso 1 View, reduce the node size, and remove the BC vectors by using the Tool Bar *Iso 1 View, Node Size, Reset graphics,* and then *Refresh graphic* icons.



To plot the results to posted FEM use the **Results** *Application radio button.*

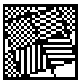
◆ **Results**

Create/Quick Plot

Select Result Cases **TIME: 4.0000000000D+01 S...**

Select Fringe Result **Temperature,**

Select the *Fringe Attributes* icon.



Display:	**Element Edges**
Label Style...	
Label Format:	**Fixed**
Significant figures	**4 <use slider bar>**
OK	
Apply	

The model should now appear as shown on the front panel of this exercise.

Quit MSC/ Patran

10. Quit MSC/PATRAN

To stop MSC/PATRAN select **File** on the *Top Menu Bar* and select **Quit** from the drop-down menu.