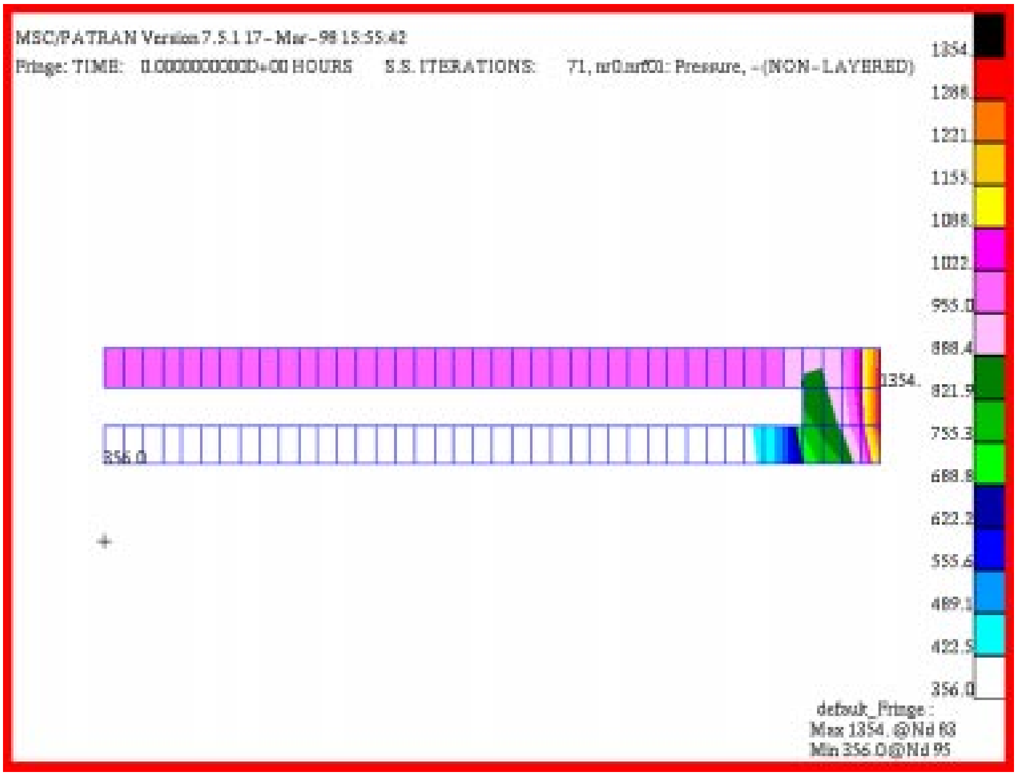
Exercise 12

Analysis of a Fuel Nozzle Tip



Objective:

- Model an axisymmetric slice of a fuel nozzle tip.
- Apply advective, radiative, and convective boundary conditions.
- Run a steady state analysis and display results.

Analysis of a Fuel Nozzle Tip

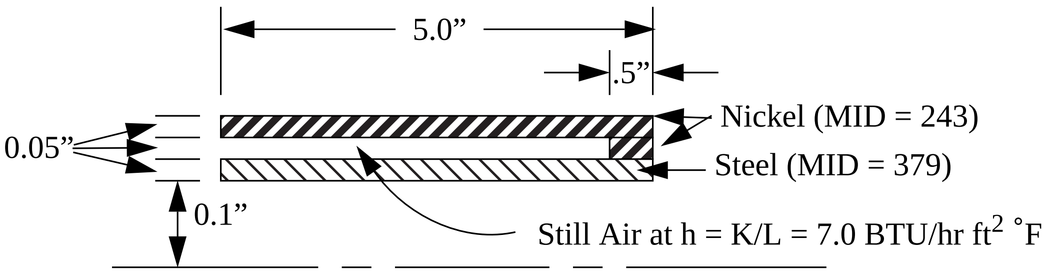
Model Description:

In this exercise you will create an axisymmetric model of a fuel nozzle tip. You will model the heat transfer contribution of the fuel flow by an advective boundary condition. The geometry and boundary conditions for the problem are shown below

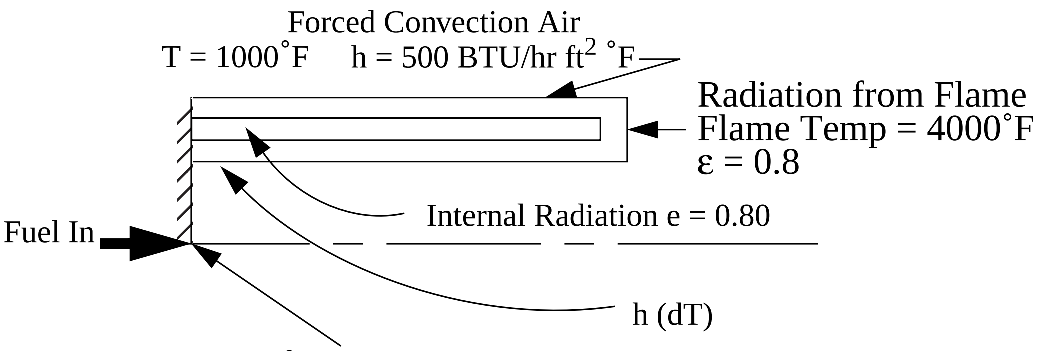
The interior surface of the nozzle across which the fuel flows must be coupled to the fuel flow with a heat transfer coefficient. Since the corresponding fluid sink will not be a single node but a series of nodes the usual Load./BCs *Create/Convection/Use Correlations* form does not apply. Until the *Between Regions* Option is implemented for 2D dimensionality (It is currently applicable only to 3D models.) you must use Element Type *Convective Quads* to couple the inner diameter of the nozzle to the fluid flow.

Figure 1

Geometry



Boundary Conditions



Specific gravity = 0.78
Tin = 200 ˚F
h (dT) = 1.0 at 0.0 ˚F and 500.0 at 800 ˚F
$\dot{m}$ = 50 lbm/hr
C_p = 0.57 BTU/lbm ˚F

Exercise Overview:

- Create a new database named **exercise_12.db.** Set *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**.
- Create the nozzle, fluid stream, and Convective Quad geometry.
- Verify that surface normals are consistent with RxZ reversing any surface normals which are not consistent with RxZ.
- Mesh the model surfaces with an IsoMesh of Quad4 elements and the curve representing the fluid stream with Bar2 elements, global edge length of 0.25.
- Use **Finite Elements/Create/Node/Edit** to create two ambient nodes 998 and 999 for the ambient and flame temperatures, respectively.
- Equivalence the nodes at the mating surface edges.
- Apply Thermal Axisymmetric element properties to the nozzle and Advection Bar element properties to the flow stream.
- Convert fluid stream nodes to fluid nodes using **Utilities** and apply element properties for Convective Quad's.
- Create fuel convection coefficient as a factor of temperature difference.
- Define three fixed temperature, two convective, and two radiative boundary condition in Loads/BC's.
- Create and post a group which does not contain the Convective Quad elements.
- Open a new window (shell) and in the directory which contains the database vi edit a file named **template.dat.apnd** creating the CONV and VFAC definitions.
- Create a **mat.dat.apnd** file containing the fuel mass flow Cp MPID data provided in Figure 1.
- Prepare and submit the model for analysis specifying that it is steady state analysis including viewfactor and radiation resistor computations, for an axisymmetric model with unit conversions from inches to feet that all calculations and output should be in ^{o}F.
- Read and plot the results.
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_12.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 *Menu Bar* and select **New …** from the drop-down menu. Assign the name exercise_12.db to the new database by clicking in the *New Database Name* box and entering **exercise_12.**

Select **OK** to create the new database

File

New Database...

New Database Name **exercise_12**

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 *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences form.

Tolerance ◆ **Default**

Analysis Code **MSC/THERMAL**

OK

Create the nozzle and fluid stream geometry

2. Create the nozzle, fluid stream, and Convective Quad geometry.

Select the **Geometry** *Applications radio button*. Create the first of two surfaces that represent the geometry of the outer nozzle shell using the following *Action*, *Object*, and *Method*.

◆ **Geometry**

Create/Surface/XYZ

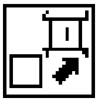
☐ **Auto Execute**

Vector Coordinates List **<4.5 0.05 0>**

Origin Coordinates List **[0 0.2 0]**

Apply

Use Tool Bar *Show Labels* icon to turn on labels.



To create the second surface change the *Vector Coordinates List* to <**0.5, 0.05, 0**>. Click in the *Origin Coordinates List* and select **Point 4** (the lower right corner of Surface 1).

◆ **Geometry**

Create/Surface/XYZ

Vector Coordinates List	**<0.5 0.05 0>**
Origin Coordinates List	**<select Point 4, the lower right corner point of Surface 1,from the viewport>**

Apply

Select **Viewing/Scale Factors...** to increase the scale of the model in the Y-direction. This will expand the model display to facilitate viewing, picking, and displaying results. Only the model display is scaled not the actual model dimensions. Scaling may throw the coordinate system symbol out of the display viewport.

Viewing

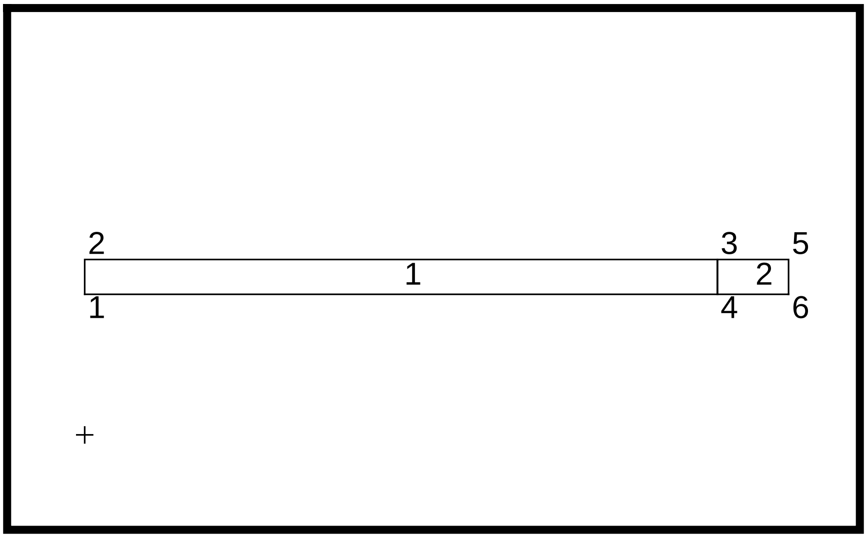
Scale Factors...

Model Y	**5.0**

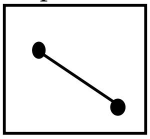
Apply

Cancel

The resulting model is shown below.



To create the surfaces that will represent the geometry where the Steel and Still Air will reside set the <u>Geometry</u> form *Action*, *Object*, and *Method* to **Transform/Surface/Translate**. Click in the *Translator Vector* databox and then choose the following *Select 2 point* icon.



Click on **Point 5** and **Point 6** to define the translation vector. Next, set the *Repeat Count* to **2**, click in the *Surface List* databox and drag a rectangle around **Surface 1** and **Surface 2** in the viewport.

◆ **Geometry**

Transform/Surface/Translate

Translation Vector — **<choose the *Select 2 points* icon (shown above) in the Select Menu and select Point 5 and then Point 6 in the viewport>**

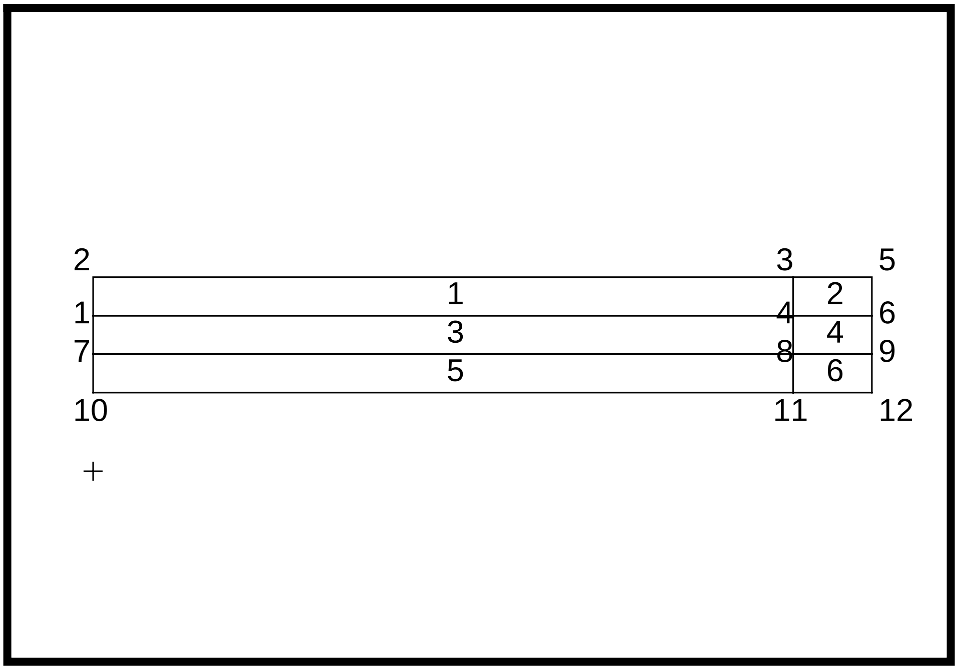
Repeat Count — **2**

☐ **Auto Execute**

Surface List — **<drag a rectangle around both surfaces in the viewport>**

Apply

The resulting model is shown below.



The flow of fuel within the nozzle will be modelled with advection bars. Create the two curves where the bars will be placed. Change the *Action*, *Object*, and *Method* to **Create/Curve/XYZ**. For the first curve set the *Vector* and *Origin Coordinates List* to, **<4.5 0 0>** and **[0 0 0]** respectively.

◆ **Geometry**

Create/Curve/XYZ

Vector Coordinates List **<4.5 0 0>**

☐ **Auto Execute**

Origin Coordinates List **[0 0 0]**

Apply

To create the second curve set the *Vector* and *Origin Coordinates List* to **<0.5, 0, 0>** and **Point 14** respectively.

Vector Coordinates List **<0.5 0 0>**

Origin Coordinates List **<select Point 14>**

Apply

Create surfaces between **Curve 1** and the lower edge of **Surface 5** and between **Curve 2** and the lower edge of **Surface 6**. These surfaces will support the Convection Quad elements.

Set the *Action*, *Object*, and *Method* to Create/Surace/Curve. Select the **2 Curve** *Curve Option* and click in the *Starting Curve List* box. Make sure that the *Curve* icon is highlighted in the Select Menu, then drag a rectangle around **Curve 1 and 2**. Select the *Surface Edge* icon then drag a rectangle around the lower edges of **Surfaces 5 and 6**.

◆ **Geometry**

Create/Surface/Curve

☐ **Auto Execute**

Starting Curve List	**<drag a rectangle around Curves 1 and 2>**
Ending Curve List	**<change the Select Menu icon to *Select an Edge of a Surface* and use shift-left mouse button to select the lower edges of Surfaces 5 and 6>**

Apply

Now, delete **Surface 3** in the air gap.

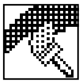
◆ **Geometry**

Delete/Any

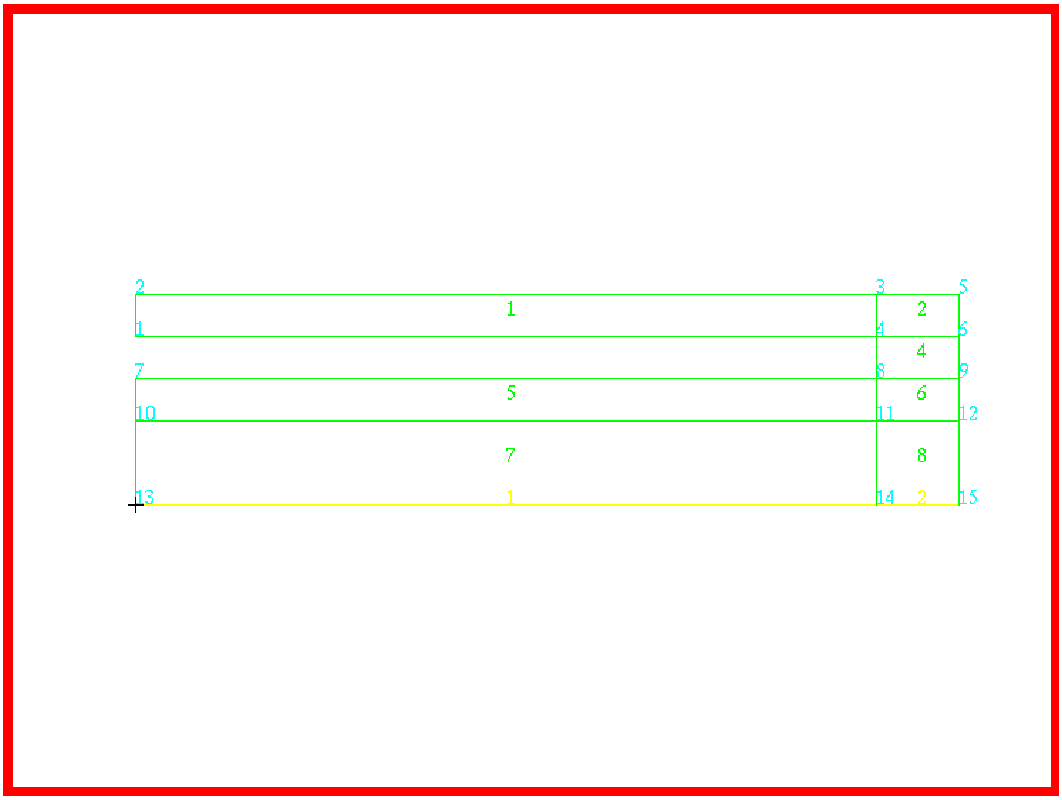
Geometric Entity List	**<Surface 3>**

Apply

Refrest the graphics.



The resulting model is shown below.



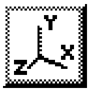
Verify surface normals and flow direction

3. Verify that surface normals are consistent with RxZ. Reverse any surface normals which are not consistent with RxZ.

Radiative boundary conditions modeled in an axisymmetric coordinate frame must have all element normals pointing in the RxZ (read R cross Z) direction. In this model, RxZ is in the global -Z direction. It is wise to verify the normal direction now since there are fewer surfaces than elements. This will facilitate viewing and reversing normals. Element normal will follow geometry normals in a 2D model.

Alternatively, element normals can be reversed, if necessary, later in the modeling process. However, if LBC’s are applied to elements before the normals are reversed then when the element normals are reversed the LBC’s may be dropped from those elements and require review and reapplication.

To verify normals change to an *isometric view* using the Tool Bar icon.



Use **Show/Surface/Normal**. Drag a rectangle around all surfaces. In this model all surfaces normals must be reversed. Use **Edit/Surface/Reverse**, select all the surfaces, **Draw Normal Vectors** to verify reversal.

◆ **Geometry**

Show/Surface/Normal

Surface List **<drag a rectangle around all surfaces in the viewport>**

Apply

Edit/Surface/Reverse

☐ **Auto Execute**

Surface List **<drag a rectangle around all surfaces in the viewport>**

Apply

Draw Normal Vectors

Reset Graphics

It is also prudent to verify the direction of the flow stream. Advection in an element flows in the local node 1 to node 2 direction. Unless reversed, the element local node 1/node 2 direction will follow the parent curve C1, or parametric, direction. Hence, it is sufficient to verify the C1 directions of Curve 1 and Curve 2. There is a toggle for displaying geometric parametric directions in **Display/Geometry**. Curves have only one parametric direction which is shown in the same color as the curve. Scaling may have offset the parametric marker from the curve but it’s color and relative length should facilitate identification.

Display

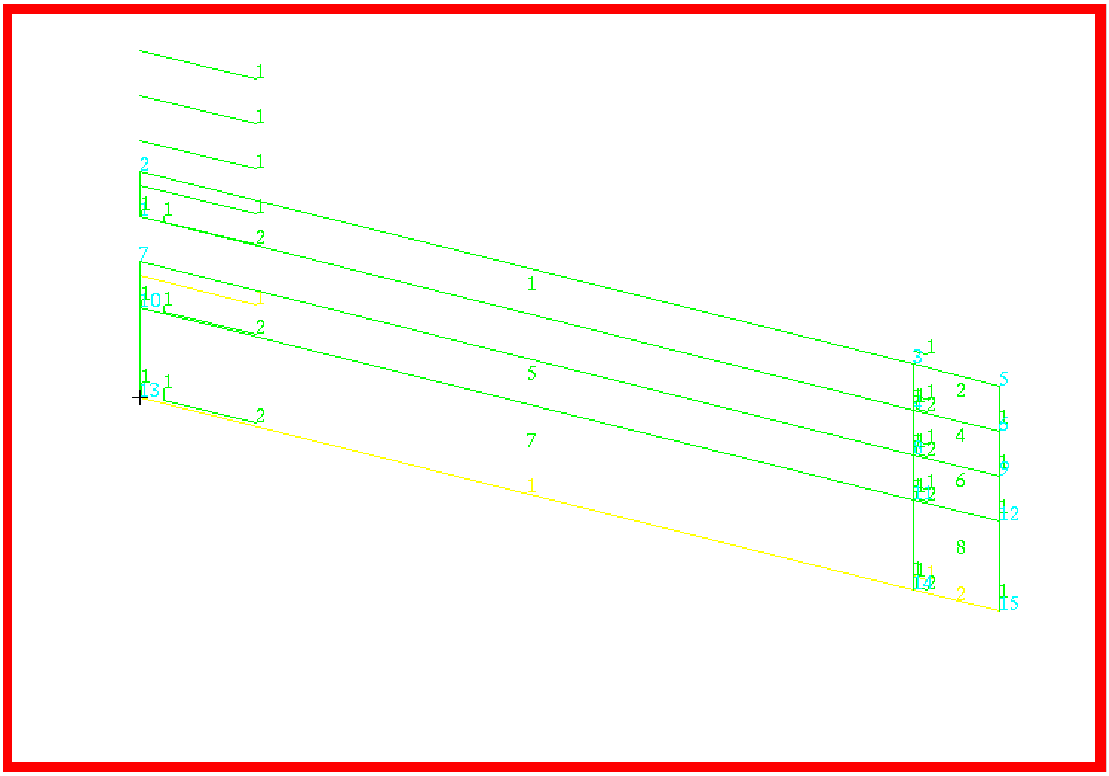
Geometry...

■ **Show Parametric Direction**

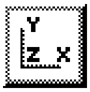
Apply

Cancel

The resulting display is shown below.



Return to default **Front view**.



Remove parametric directions display.

Display

Geometry...

☐ **Parametric Direction**

Apply

Cancel

4. Mesh the model surfaces with an IsoMesh of Quad4 elements and the curve representing the fluid stream with Bar2 elements, global edge length of 0.25.

IsoMesh the surfaces and fluid stream curve

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to 0.25 and click in the *Surface List* box. Drag a rectangle around all surfaces in the viewport.

◆ **Finite Elements**

Create/Mesh/Surface

Global Edge Length **0.125**

Surface List **<drag a rectangle around all surfaces in the viewport>**

Apply

Create Bar2 elements along Curves 1 and 2.

◆ **Finite Elements**

Create/Mesh/Curve

Global Edge Length **0.125**

Curve List **<select Curves 1 and 2 using the shift-left mouse button>**

Apply

5. Use **Finite Elements/Create/Node/Edit** to create two ambient nodes 998 and 999 for the ambient and flame temperatures.

Create boundary nodes

In the Finite Elements form create a boundary node which is not associated with geometry. The node is numbered **998**. Locate the node at **[2.5 0.3 0]**.

◆ **Finite Elements**

Create/Node/Edit

Node ID List **998**

☐ **Associate with Geometry**

☐ **Auto Execute**

Node Location List **[2.5 0.3 0]**

Apply

Repeat for Node 999 located at [5.2 0.15 0].

Increase the display size of nodes. Use either **Display/Finite Elements ...** or the associated Tool Bar icon to change the node size.

Display

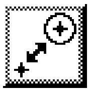
Finite Elements...

Node Size | **9** <use slider bar>

Apply

Cancel

or,



Select **Display/Entity Color/Label/Render .../Hide All Entity Labels** or use the Tool Bar *Labels Hide* icon to remove all labels and unclutter the display.

Display

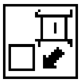
Entity Color/Label/Render...

Hide All Entity Labels

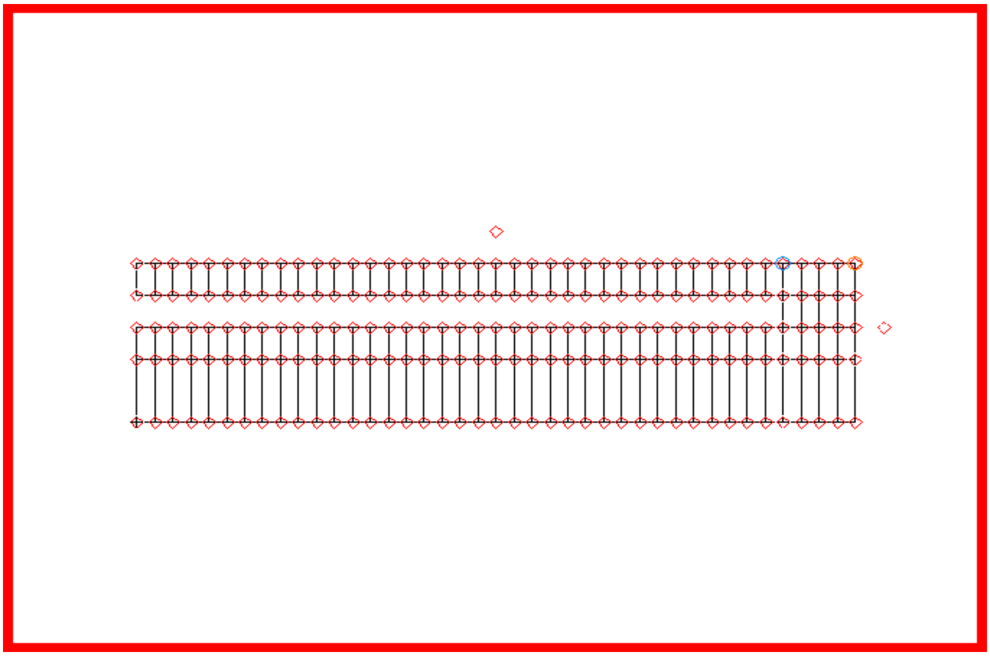
Apply

Cancel

or,



The display should now appear as shown below.



6. Equivalence the nodes at the mating surface edges.

Equivalence nodes

Using the Finite Elements form set the *Action/Object/Method* to **Equivalence/All/Tolerance Cube** and select **Apply** to eliminate duplicate nodes created at geometric entity edges.

◆ **Finite Elements**

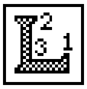
Equivalence/All/Tolerance Cube

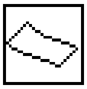
Apply

7. Apply Thermal Axisymmetric element properties to the nozzle and Advection Bar element properties to the flow stream.

Apply element properties to nozzle

Use Tool Bar *Label Control* icon to turn on *Surface* labels.





Close

Select the **Properties** *Applications radio button*. Set the *Action*, *Dimension*, and *Type* to **Create/2D/Thermal Axisymmetric**. Enter *Property Set Name* **Nickel**. Select the *Input Properties...* box. Click in the *Material Name* box and enter **243**. Select **OK** to close the form.Click in the *Select Members* box and select **Surfaces 1, 2, and 4** in the viewport using the shift-left mouse button. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties

Create/2D/Thermal Axisymmetric

Property Set Name **Nickel**

Input Properties...

Material Name **243**

OK

Select Members **<select Surfaces 1, 2, and 4 in the viewport using shift-left mouse button>**

Add

Apply

Repeat these steps for Steel, MID 379, on Surfaces 5 and 6.

The two element property set names should now appear in the *Property Set Name* list box.

Create fluid nodes and Convective Quads

8. Convert fluid stream nodes to fluid nodes using **Utilities** and apply element properties for Convective Quad's.

Convective Quad elements must have at least one Fluid Node associated with each element. Fluid nodes are a 0D element type applied to selected nodes. There are two means of creating Fluid Nodes, using **Element Properties** or using **Utilities**. **Choose one of the two following methods**.

Using Utilities:

Utilities

Thermal

Create Node Type Elements...

OK

New Set Name **Fluid_nodes**

Select Nodes — **<drag a rectangle around the string of nodes at the bottom of Surfaces 7 and 8 along the flow stream>**

Apply

Cancel

Or, using the Properties form:

◆ Finite Elements

Create/Element/Edit

Shape — **Point**

Node 1 = — **<drag a rectangle around the string of nodes at the bottom of Surfaces 7 and 8 along the flow stream>**

Apply

◆ Properties

Create/0D/Node Type

Property Set Name — **Fluid_nodes**

Input Properties...

Value Type — **String(Fluid Node)**

OK

Select Members — **<drag a rectangle around the string of nodes at the bottom of Surfaces 7 and 8 along the flow stream>**

Add

Apply

Small triangles will mark each fluid node.

Now **Create/2D/Convective Quad** elements on **Surfaces 7 and 8** with a *Template ID* of **10**. Later you will input the convection heat transfer coefficient in the *template.dat.apnd* file.

Convective Quads have no physical reality in the model; they are a device for passing cross sectional area data, convection configuration data (GP's), and fluid node data to the convection algorithm. When the *Between Region* option is expanded to include 2D dimensionality, the need for Convection Quads will be limited to passing data to user defined configurations.

◆ **Properties**

Create/2D/Convective Quad	
Property Set Name	**Conv_quads**
Input Properties...	
[Template ID]	**10**
OK	
Select Members	**<select Surfaces 7 and 8 in the viewport using shift-left mouse button>**
Add	
Apply	

The last element property you will create will define the Bar2 elements as advective bars. Change the *Dimension* to **1D** and the *Type* to **Advection Bar**. Enter **Adv_bars** for the *Property Set Name* and then click on the **Input Properties…** button. When the *Input Properties* form appears enter **1** for the *Cp-MPID* and **50** for the *Mass Flow Rate*.

Create/1D/Advection bar	
Property Set Name	**Adv_bars**
Input Properties...	
[Specific Heat MPID]	**1**
Mass Flow Rate	**50**
OK	
Select Members	**<select Curves 1 and 2 using shift-left mouse button>**
Add	
Apply	

Though the Specific Heat MPID appears in square brackets it is, in fact, not an optional entry. Even in a steady state analysis advective conductors are derived from the product of specific heat and mass flow rate.

Five *Existing Property Sets* should now be listed in the Element Properties form. Adv_bars, Conv_quads, Fluid_Nodes[_Fluid], Nickel, and Steel. Scroll through the list to verify it.

9. Create fuel convection coefficient as a factor of temperature difference.

Select the **Fields** application radial button. Use **Create/Material Property/General** to create the variable h called **h_fuel**.

◆ Fields	
Create/Material Property/ General	
Field Name	**h_fuel**
Input Data...	
Select Function Term	**mpid_indx_linr_tabl**
Material Property ID (MPID)	**1001**
Temperature Units	**Fahrenheit**
Independent Variable, (X)	**0.0**
Value, Function (X)	**1.0**
Enter	
Independent Variable, (X)	**800.0**
Value, Function (X)	**500.0**
Enter	
OK	
OK	
Apply	

Apply boundary conditions

10. Define three fixed temperature, two convective, and two radiative boundary condition in Loads/BC's.

Select the **Load/BCs** *Applications radio button*. Create a fixed **1000^{o}F** nodal boundary temperature named **T_air**. In the Input Data form define the fixed temperature. In the Select Region form pick **Node 998**, located above the nozzle model.

◆ **Load/BCs**	
Create/Temperature/Nodal	
Option:	**Fixed**
New Set Name	**T_air**
Input Data...	
Fixed Temperature	**1000.0**
OK	
Select Application Region...	
Geometry Filter	◆ **FEM**
Select Nodes	**<select Node 998>**
Add	
OK	
Apply	

Repeat these steps for a *New Set Name* T_flame of 4000 ^{o}F applied to Node 999, located to the right of the nozzle and for a *New Set Name* T_fuel of 200^{o}F applied to Node 179, located at the lower left corner of the model at the fuel stream inlet.

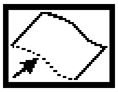
Create the ambient convection boundary condition. Use a *New Set Name* **Amb_conv**, a *Convection Coefficient* of **500.0**, and a *Fluid Node* **998**.

◆ **Load/BCs**	
Create/Convection/ Element Uniform	
New Set Name	**Amb_conv**
Target Element Type	**2D**
Input Data...	
Convection Coefficient	**500**
Fluid Node ID	**998**

OK

Select Application Region...

Geometry Filter	**◆Geometry**
Select Menu	***Select an Edge*** **icon**



Select Surface or Edges	**<Select the top edges of Surfaces 1 and 2 (Surface 1.3 and 2.3) using the Shift-left mouse button>**

Add

OK

Apply

Create gap condition across still air gap with h=k/L where **k = 0.029 BTU/ hr ft^2 °F** and **L = 0.05/12 ft**. Hence **h = 7.0 BTU/hr ft^2 °F**.

◆ Load/BCs

Create/Convection/Element Uniform

Option	**Between Regions**
New Set Name	**Still_air**
Target Element Type	**2D**
Region 2	**2D**

Input Data...

Convection Coefficient	**7.0**

OK

Select Application Region...

Order	**Closest Approach**
Select Surface or Edges	**<Select the bottom edge of Surfaces 1. (Surface 1.1)>**

Add

Select the bottom most Active List, which is the bottom to select region 2.

Active List

Select Surface or Edges

<Select the top edge of Surfaces 5. (Surface 5.3)>

Add

OK

Apply

Create the flame radiation boundary condition. Use a *New Set Name* **Flame_rad**, a *VFAC Template ID* of **10**, and an *Ambient Node* **999**, a Convex Surface ID of **999**, an *Obstr Flag* of **1**, and an *Enclosure ID* of **1**.7

◆ **Load/BCs**

Create/Radiation/Element Uniform

New Set Name	**Flame_rad**
Target Element Type	**2D**

Input Data...

Vfac Template ID	**10**
Ambient Node ID	**999**
Convex Surface ID	**999**

(Note: Use Scroll bar to access more fields)

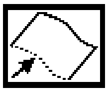
Obstr Flag(0=Obstr, 1=No-Obtrs)	**1**
Enclosure ID	**1**

OK

Select Application Region...

Geometry Filter — ◆ **Geometry**

Select Menu — ***Select an Edge*** **icon**



Select Surface or Edges	**<Select the right edges of Surfaces 2, 4, and 6, by using the shift-left mouse botton. Do not include the right edge of Surface 8 (the convective quads>**

Add

OK

Apply

Create the radiation effect in the still air gap.

◆ **Load/BCs**

Create/Radiation/Element Uniform

New Set Name	**Still_air_rad**
Target Element Type	**2D**

Input Data...

Vfac Template ID	**10**
Ambient Node ID	**<no entry>**
Convex Surface ID	**<no entry>**
Obstr Flag(0=Obstr, 1=No-Obtrs)	**1**
Enclosure ID	**2**

There are only 2 entries in this Input Data form. VFAC Template ID and Enclosure ID.

OK

Select Application Region...

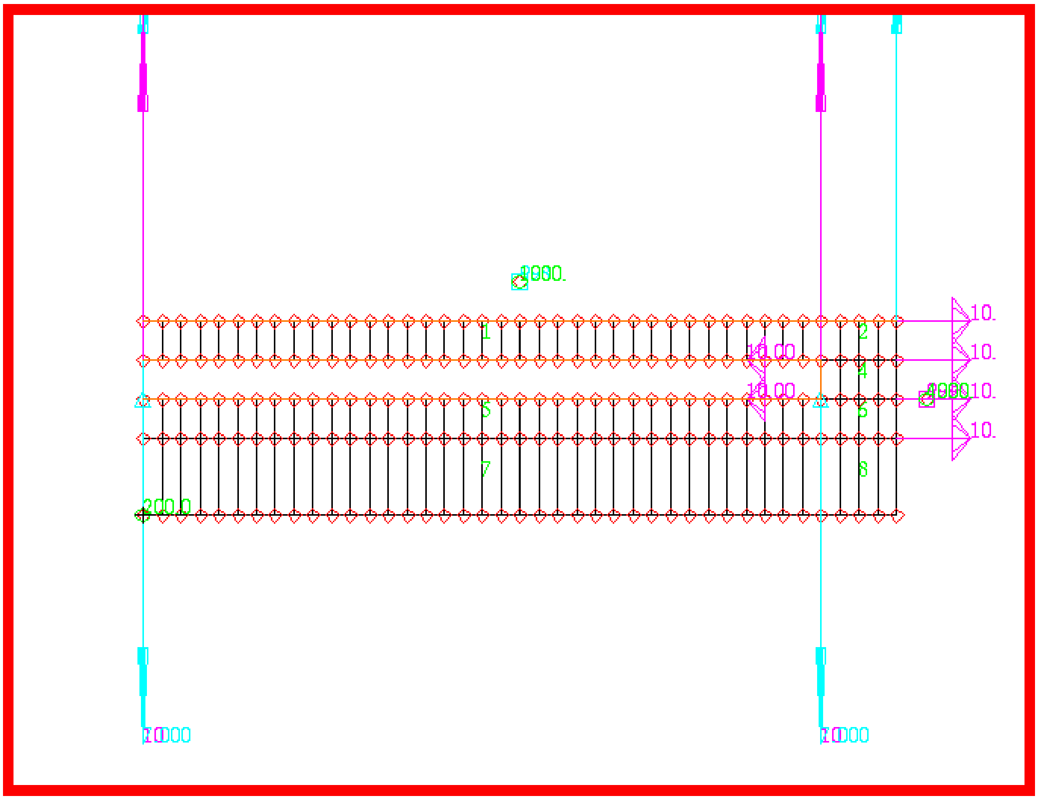
Geometry Filter	◆ **Geometry**
Select Surface or Edges	**<Select the perimeter of the still air gap, Surface 1.1, 4.4, and 5.3 using the shift left mouse button.>**

Add

OK

Apply

With boundary conditions applied the model should now appear as shown below.



Create a group maned nozzle

11. Create and post a group name Nozzle which does not contain the Convective Quad elements.

Since Convective Quad elements have no physical reality in the model we will prepare the display by eliminating them from the viewport You will create a group which will contain only entities associated with the nozzle. To avoid picking the Convective Quads, first change the Rectangle Picking option.

Preference

Picking...

Rectangle/Polygon Picking ◆**Enclose centroid**

Close

Now, create the group.

Group

Create...

New Group Name **Nozzle**

Make Current

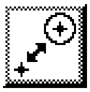
Unpost All Other Groups

Entity Selections **<drag a rectangle around the nozzle portion of the model including the two boundary nodes excluding Convective Quads>**

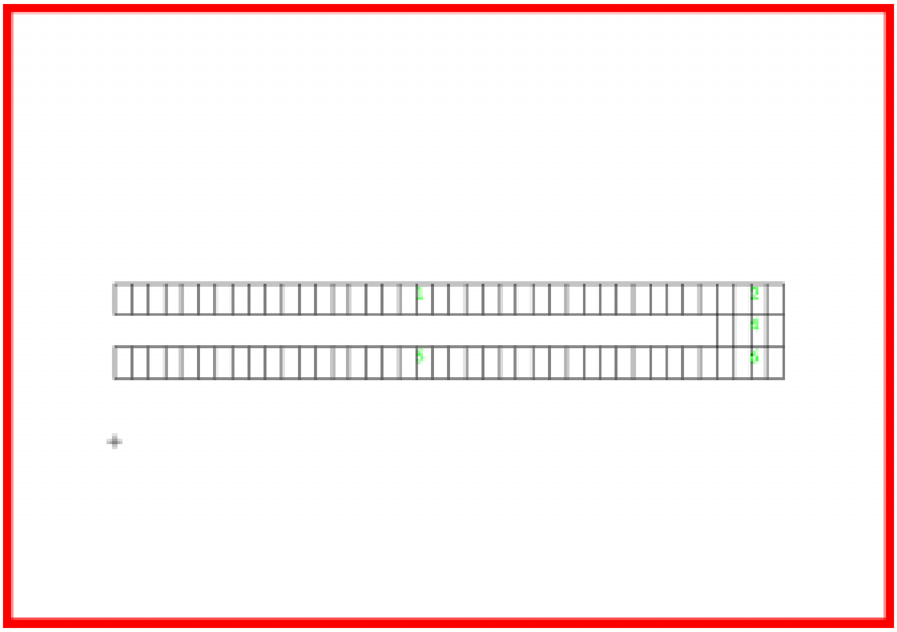
Apply

Cancel

Reduce the node size with the *Node Size* icon.



The model should now appear as shown below.



In unix create template.dat. apnd file

12. Open a new window (shell) and in the directory which contains the database vi edit a file named **template.dat.apnd** creating the CONV and VFAC definitions.

Open a unix xterm window and change directories (cd) to the directory which contains your database.

If a template.dat.apnd already exists in this directory rename it to associate it with that previous analysis. For instance, in Exercise 11 you created a tempate.dat.apnd file. Use the following unix command to move it to a new name associated with that analysis:

> **mv template.dat.apnd 11_template.dat.apnd**

Using the system editor, typically vi, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC/PATRAN is running.

Create two definitions, a CONV for the fuel flow Convective Quads and the other, a VFAC for the flame radiation boundary condition. Shown below is the final form of the **template.dat.apnd** file created for this exercise. Note that any comment lines must be started with an * in column 1 and make sure that there are no blank lines especially at the end of the file. Start typing from the first column and do no enter any blank lines.

```
CONV 10 30 0 1

1001

VFAC 10 0

0.8 1
```

In unix edit a mat.dat.apnd file

13. Create a **mat.dat.apnd** file containing the fuel mass flow Cp MPID data provided in Figure 1.

If a mat.dat.apnd already exists in this directory rename it to associate it with that previous analysis. For instance, in Exercise 11 you copied a mat.dat.apnd file. Use the following unix command to move it to a new name associated with that analysis:

> **mv mat.dat.apnd 11_mat.dat.apnd**

Using the system editor, typically vi, create and edit a new file **mat.dat.apnd** in the directory which contains your database and where MSC/PATRAN is running.

You will define MPID 1 for the specific heat property of the advective flow. There is an alternative method for creating MPID definitions. Recall, you can also use FIelds/Material Property/General to accomplish this. Shown below is the final form of the **mat.dat.apnd** file created for this exercise.

Make sure that there are no blank lines especially at the end of the file. Start typing from the first column and make sure to close the MPID definition with a slash (/).

```
MPID 1 C F 1.0

MDATA 0.57

/
```

Prepare and run analysis

14. Prepare and submit the model for analysis specifying that it is steady state analysis including viewfactor and radiation resistor computations, for an axisymmetric model with unit conversions from inches to feet that all calculations and output should be in ^{o}F.

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

Translation Parameters...

Model Dimensionality	**◆ Axisymmetric Geometry, R Z Co-ordinates**
Radial, R Co-ordinate	**◆ Yaxis**
Centerline, Z Co-ordinate	**◆ Xaxis**

Perform Geometry Units Conversion

From Units	**inches**
To Units	**feet**
File to Extract Undefined Materials:	**3,mpidfph.bin (Btu-feet-hour.**

OK

Solution Type...

Perform Viewfactor Analysis

OK

Solution Parameters...

Calculation Temperature Scale	**◆ Fahrenheit**

Run Control Parameters...

Stefan-Boltzmann Constant	**1.7140E-9 BTU/HR/FT^2/R^4**
Initial Temperature =	**1000.0**
Initial Temperature Scale	**◆ Fahrenheit**

OK

OK

Output Requests...

Units Scale for Output Temperatures	**◆ Fahrenheit**
Units Definition for Time Label	**Hours**

OK

Submit Options...

Make sure both Create ViewFactor Control FIle (vf.ctl) and Execute Viewfactor Analysis are selected.

OK

Apply

15. Read and plot the 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_12.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*, exercise_12/. 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_12**

Filter

Available Files	**nr0.nrf.01**

OK

Select Rslt Template File...

Files **pthermal_1_nodal.res_tmpl**

OK

Apply

After results are read in plot the results. To plot the results use the **Results** *Application radio button.* Select you results file.

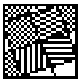
◆ Results

Create/Quick Plot

Select Result Cases **TIME: 0.0000000000D+00 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.

16. Quit MSC/PATRAN

Quit MSC/ Patran

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