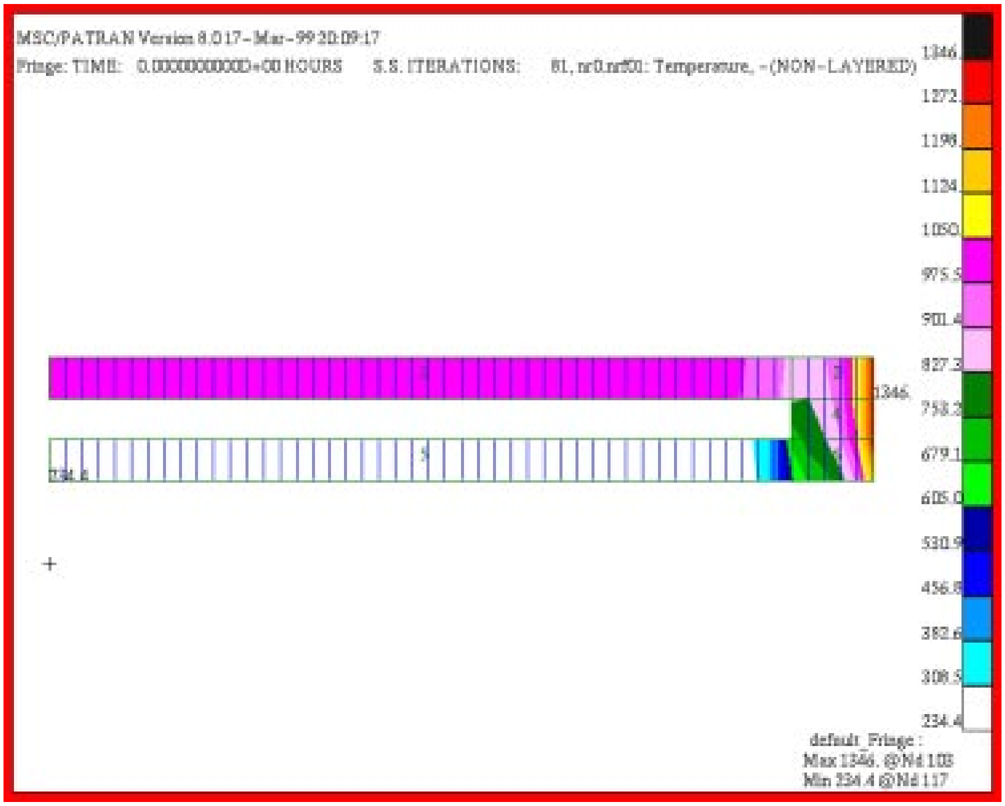
WORKSHOP 17

Analysis of a Fuel Nozzle Tip Using Convection Between Regions



Objective:

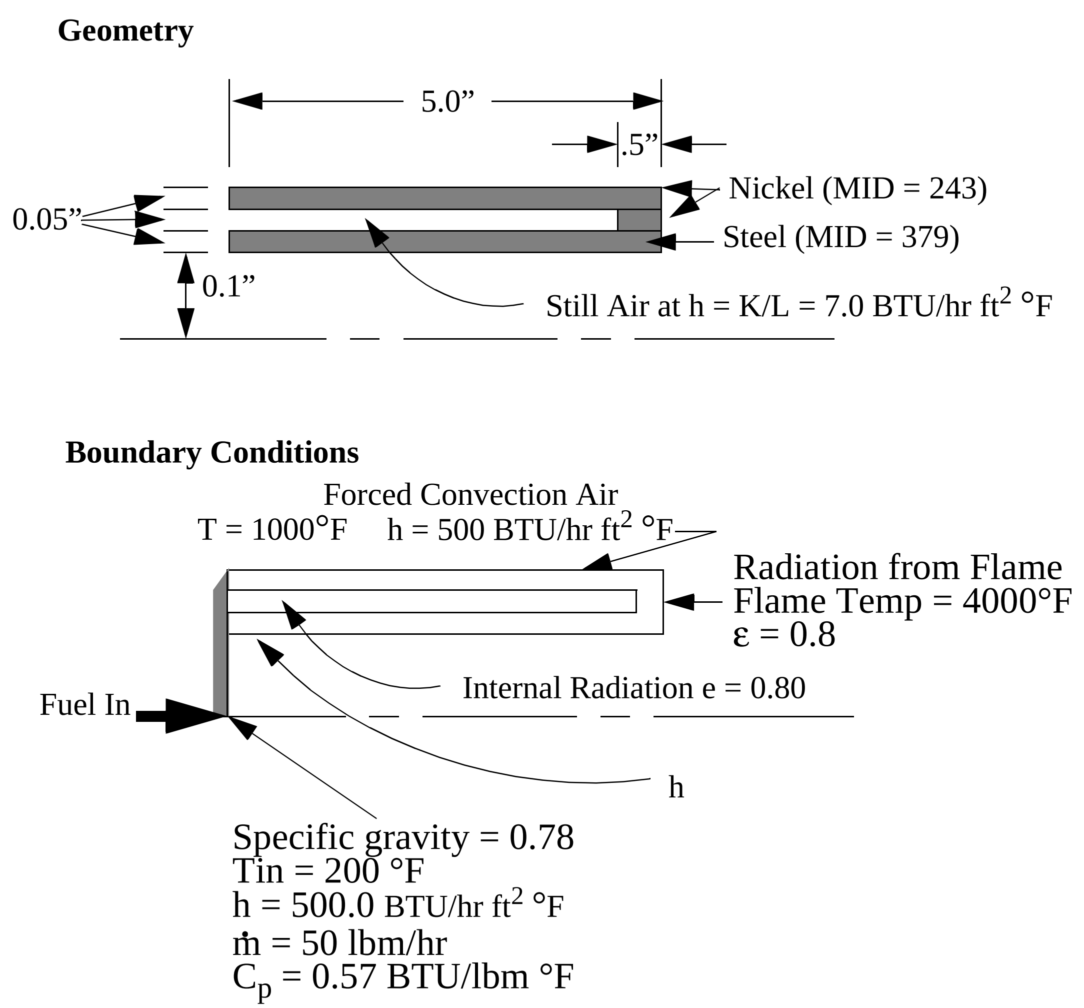
- Model an axisymmetric slice of a fuel nozzle tip.
- Apply advective, radiative, and convective boundary conditions.
- Run a steady state analysis and display results.

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 Loads/BCs *Create/Convection/Template*, *Convection* form does not apply.

Figure 1



Exercise Overview:

- Create a new database named **exercise_17.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 0.100.
- 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.
- Define three fixed temperature, three convective, and two radiative boundary condition in Loads/BC's.
- Create and post a group name Nozzle which only includes the nozzle elements.
- 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:

1. Open a new database named **exercise_17.db**.

Open a new database

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_17.db to the new database by clicking in the *New Database Name* box and entering **exercise_17**

Select **OK** to create the new database

File

New...

New Database Name **exercise_17**

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

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

Create the nozzle and fluid stream 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

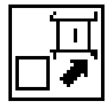
☐ **Auto Execute** **<deselect>**

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

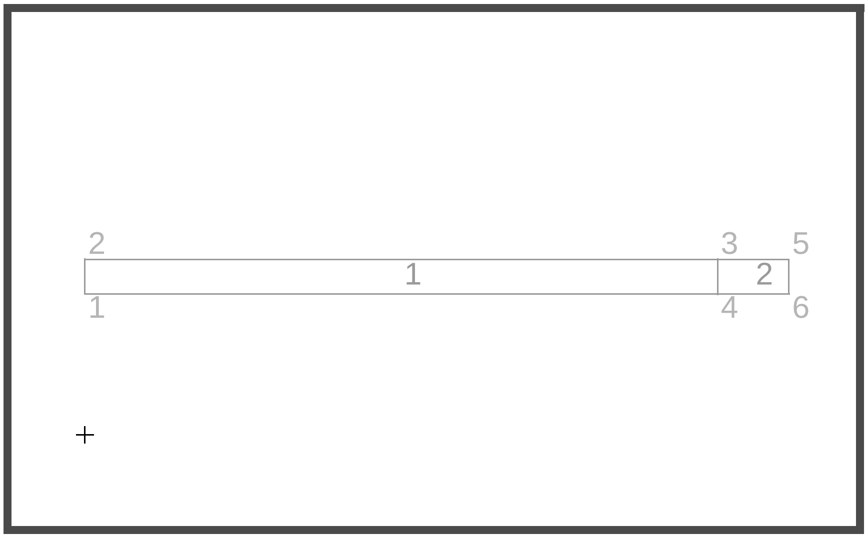
Scale Factors...

Model Y **5.0**

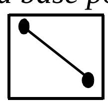
Apply

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 Geometry form *Action*, *Object*, and *Method* to **Transform/Surface/Translate**. Click in the *Translator Vector* databox and then choose the following *Tip and base points* 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.

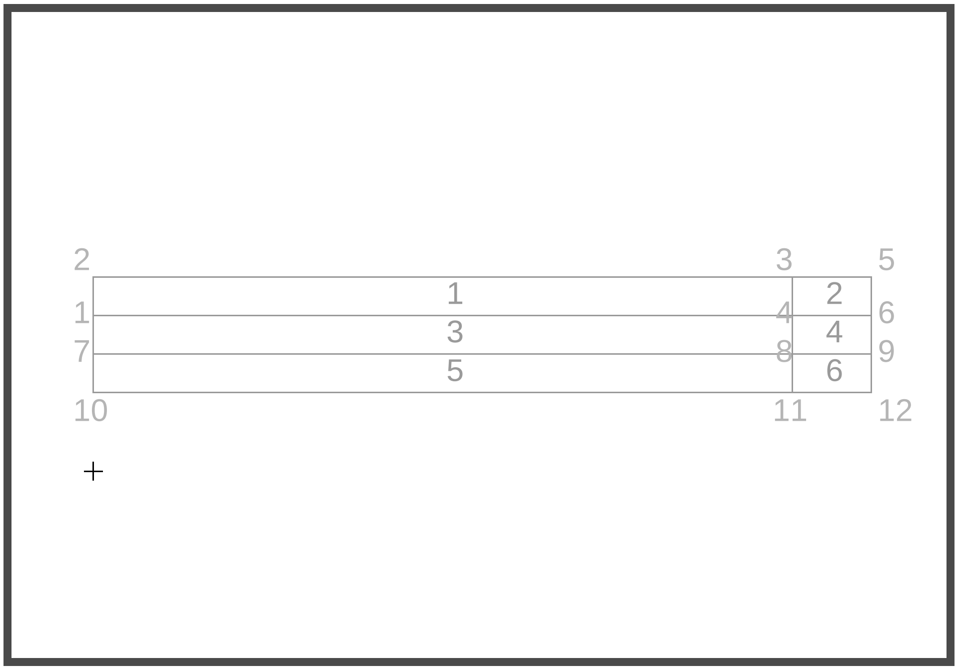
◆ **Geometry**

Transform/Surface/Translate

Translation Vector	**<choose the *Tips and base points* icon (shown above) in the Select Menu and select Point 5 and then Point 6 in the viewport>**
Repeat Count	**2**
☐ **Auto Execute**	**<deselect>**
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** **<deselect>**

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

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

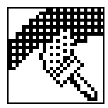
◆ **Geometry**

Delete/Any

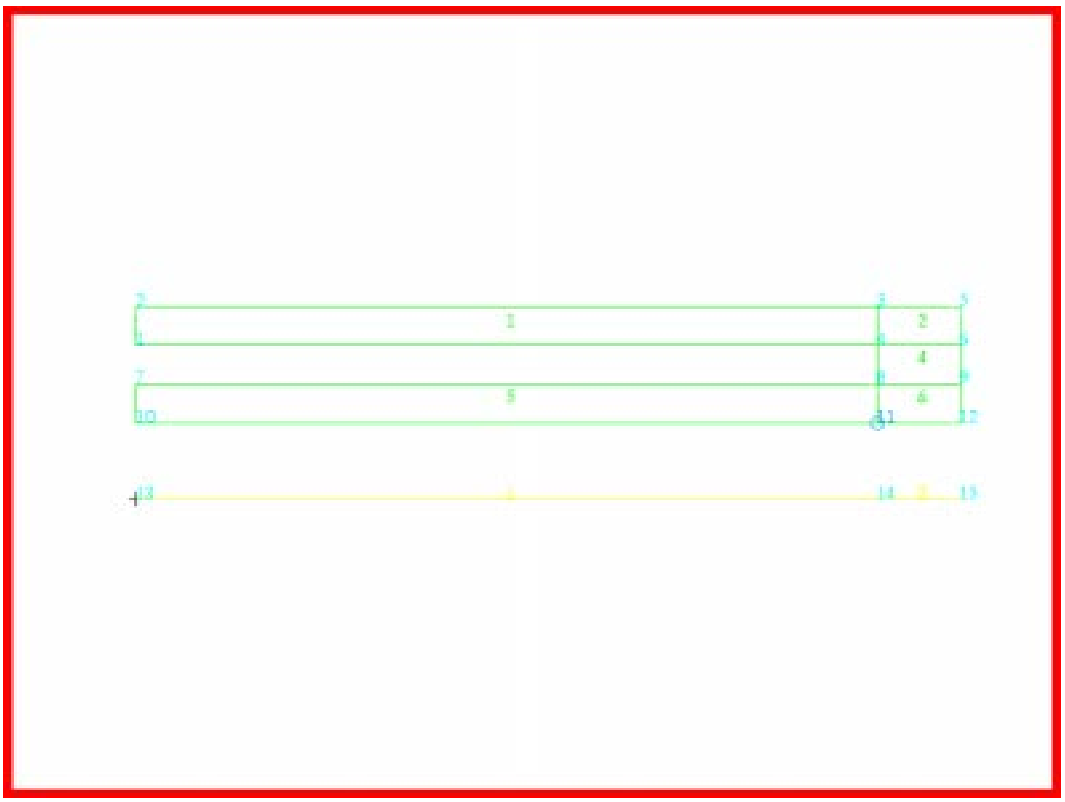
Geometric Entity List **<Surface 3>**

Apply

Refrest the graphics.



The resulting model is shown below.



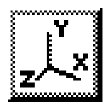
Verify surface normals and flow direction

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**	**<deselect>**
Surface List	**<drag a rectangle around all surfaces in the viewport>**
Apply	
Draw Normal Vectors	

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

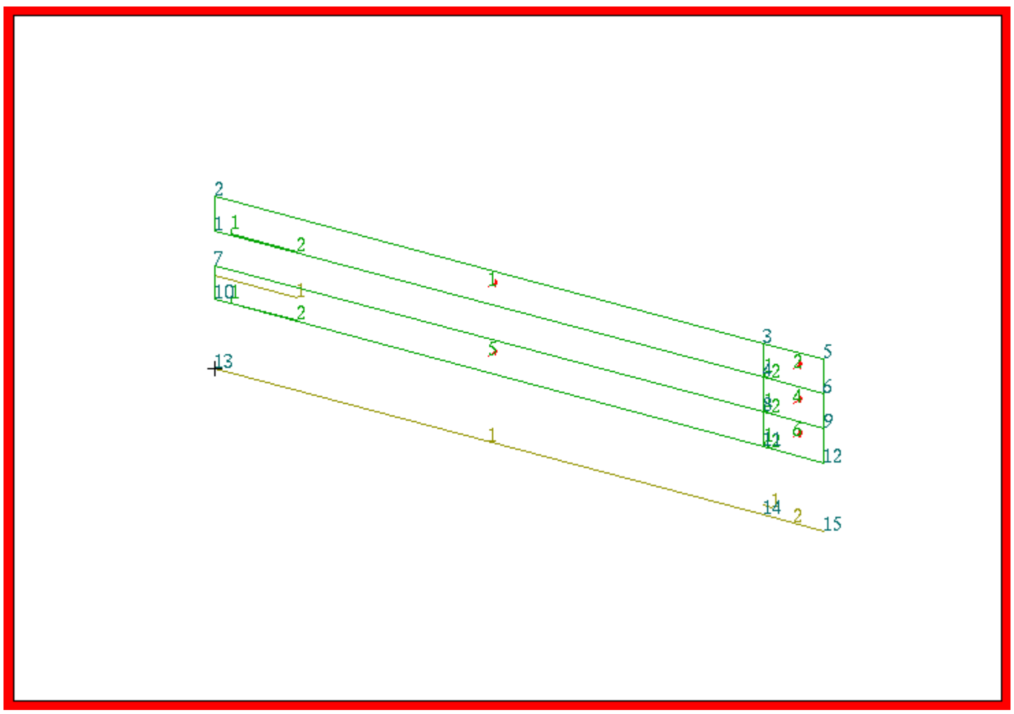
Geometry...

Show Parametric Direction

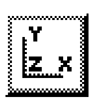
Apply

Cancel

The resulting display is shown below.



Return to default **Front view**.



Remove parametric directions display.

Display

Geometry...

☐ **Show Parametric Direction**

Apply

Cancel

Remove the Normal Vectors.

◆ **Geometry**

Reset Graphics

IsoMesh the surfaces and fluid stream curve

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 0.100.

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to 0.100 and click in the *Surface List* box. Drag a rectangle around all surfaces in the viewport.

◆ **Finite Elements**

Create/Mesh/Surface

Global Edge Length	**0.100**
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	**0.100**
Curve List	**<select Curves 1 and 2 using the shift-left mouse button>**

Apply

Create boundary nodes

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

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**	**<deselect>**
☐ **Auto Execute**	**<deselect>**
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

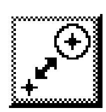
Finite Elements...

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

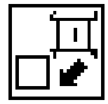
Entity Color/Label/Render...

Hide All Entity Labels

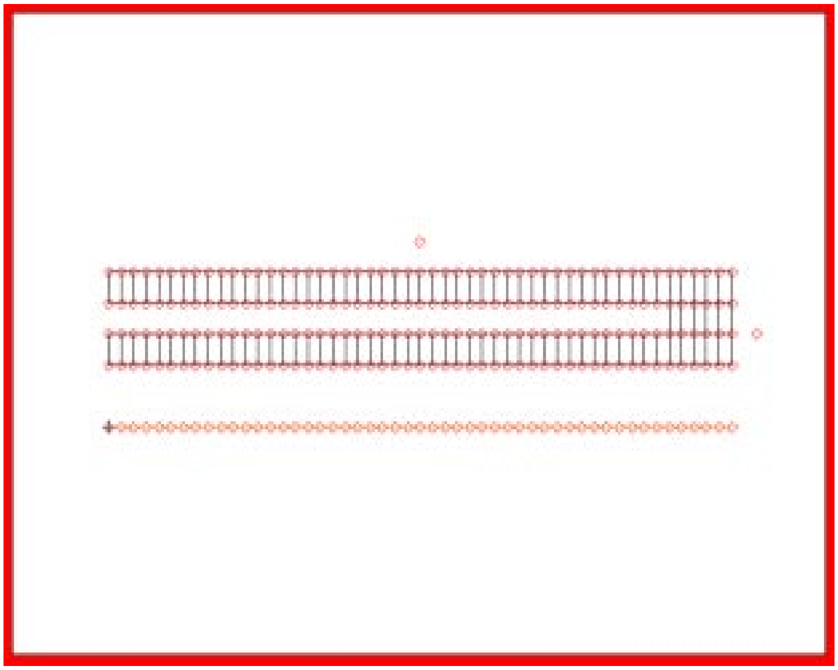
Apply

Cancel

or,



The display should now appear as shown below.



Equivalence nodes

6. Equivalence the nodes at the mating surface edges.

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**

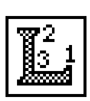
Equivalence/All/Tolerance Cube

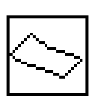
Apply

Apply element properties to nozzle

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

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 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.

Apply boundary conditions

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

Select the **Loads/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.

◆ Loads/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 221, 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**.

◆ **Loads/BCs**

Create/Convection/Element Uniform

Options:	**Template, Convection**
New Set Name	**Amb_conv**
Target Element Type	**2D**

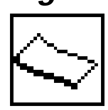
Input Data...

Convection Coefficient	**500**
Fluid Node ID	**<select node 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**. .

◆ **Loads/BCs**

Create/Convection/Element Uniform

Option	**Fixed Coefficient**
New Set Name	**Still_air**
Target Element Type	**2D**

Region 2	**2D**
Input Data...	
Convection Coefficient	**7.0**
OK	
Select Application Region...	
Click in Application Region	
Select Surface or Edges	**<Select the bottom edge of Surface 1. (Surface 1.1)>**
Add	
Click in Coupling Region	
Select Surface or Edges	**<Select the top edge of Surface 5. (Surface 5.3)>**
Add	
OK	
Apply	

Since the convection coefficient from fuel-to-nozzle is now constant, h=500BTU/hr ft^2 °F, the **Convection/Fixed Coefficient** Regions option can also be used for the fuel-to-nozzle connection.

◆ **Loads/BCs**

Create/Convection/Element Uniform

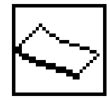
Option	**Fixed Coefficient**
New Set Name	**Fuel_convection**
Target Element Type	**2D**
Region 2	**1D**
Input Data...	
Convection Coefficient	**500.0**
OK	
Select Application Region...	
Click in Application Region	

Select Surface or Edges	**<Select the lower edge of nozzle. (Surface 5.1 and 6.1) using shift-left mouse button.>**
Add	
Click in Coupling Region	
Select Surface or Edges	**<Select Curve 1 and 2 using shift-left mouse button.>**
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**.

◆ **Loads/BCs**

Create/Radiation/Element Uniform	
Option	**Template, View Factor**
New Set Name	**Flame_rad**
Target Element Type	**2D**
Input Data...	
Enclosure ID	**1**
VFAC Template ID	**10**
Convex Surface ID	**999**
Ambient Node ID	**<select node 999>**
☐ Can be obstructing surface	**<deselect>**
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.>**

Add

OK

Apply

Create the radiation effect in the still air gap.

◆ **Loads/BCs**

Create/Radiation/Element Uniform

Option	**Template, View Factor**
New Set Name	**Still_air_rad**
Target Element Type	**2D**

Input Data...

Enclosure ID	**2**
VFAC Template ID	**10**
Convex Surface ID	**<no entry>**
Ambient Node ID	**<no entry>**
☐ Can be obstructing surface	**<deselect>**

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

OK

Select Application Region...

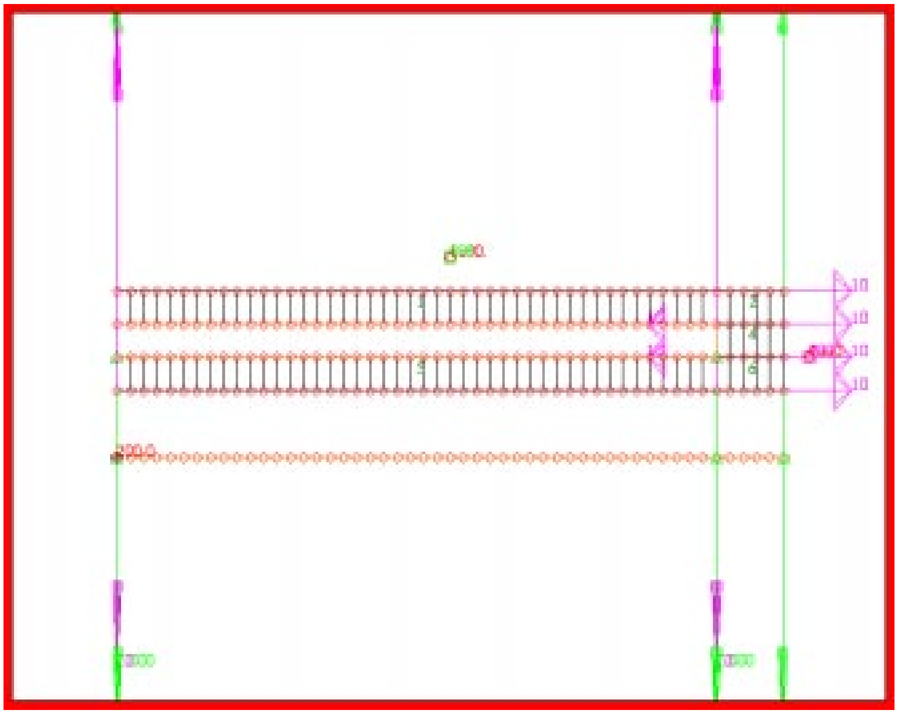
Geometry Filter	◆**Geometry**
Select Surface or Edges	**<Select the perimeter of the still air gap, Surface 1.1, 4.4, and 5.3 using the shift left mouse button.>**

Add

OK

Apply

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



9. Create and post a group name Nozzle which only includes the nozzle elements.

Create a group maned nozzle

You will create a group which will contain only entities associated with the nozzle.

Group

Create...

New Group Name	**Nozzle**
◼ Make Current	**<select>**
◼ Unpost All Other Groups	**<select>**

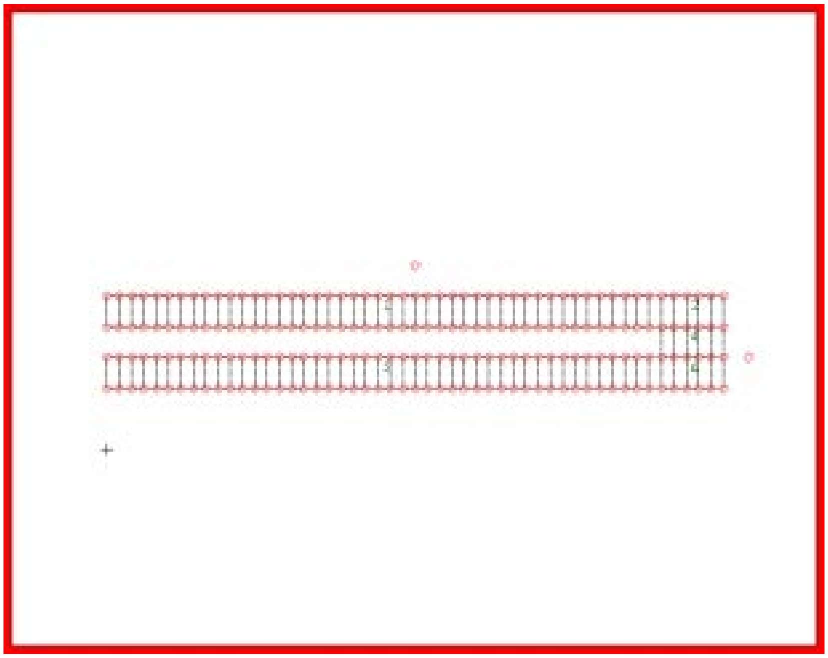
Entity Selections

<drag a rectangle around the nozzle portion of the model including the two boundary nodes>

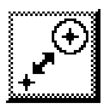
Apply

Cancel

The model should now appear as shown below.



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



In unix create template.dat.apnd file

10. Use the new Analysis/Build Template form to create a file named **template.dat.apnd** containing the VFAC definitions.

Using Analysis/Build Template, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC.Patran is running.

Create a definition 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.

◆ Analysis

Build Template

Create Template File...

Create/VFAC/Data Entry

VFAC ID	**10**
Emissivity	**0.8**

Apply

Write File...

OK

Cancel

Cancel

```
VFAC 10 0
0.8 1
```

A **mat.dat.apnd** file is required to define the specific heat property of the advecture flow. The **mat.dat.apnd** file required for this exercise is identical to that created in Exercise 12. The final form of the **mat.dat.apnd** file is shown below for reference.

```
MPID 1 C F 1.0
MDATA O.57
/
```

Prepare and run analysis

11. 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-lbm-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

12. Read and plot the results.

Read and plot results

From within MSC.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_17.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*, exercise_17/. 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_17**

Filter

Available Files	**nr0.nrf.01**

OK

Select Rslt Template File...

Files	**pthermal_1_nodal.res_tmpl**

OK

Apply

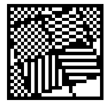
After results are read in plot the results. To plot the results use the **Results** *Application radio button.* Select you results file.

◆ **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.

Quit MSC. Patran

13. Quit MSC.Patran

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