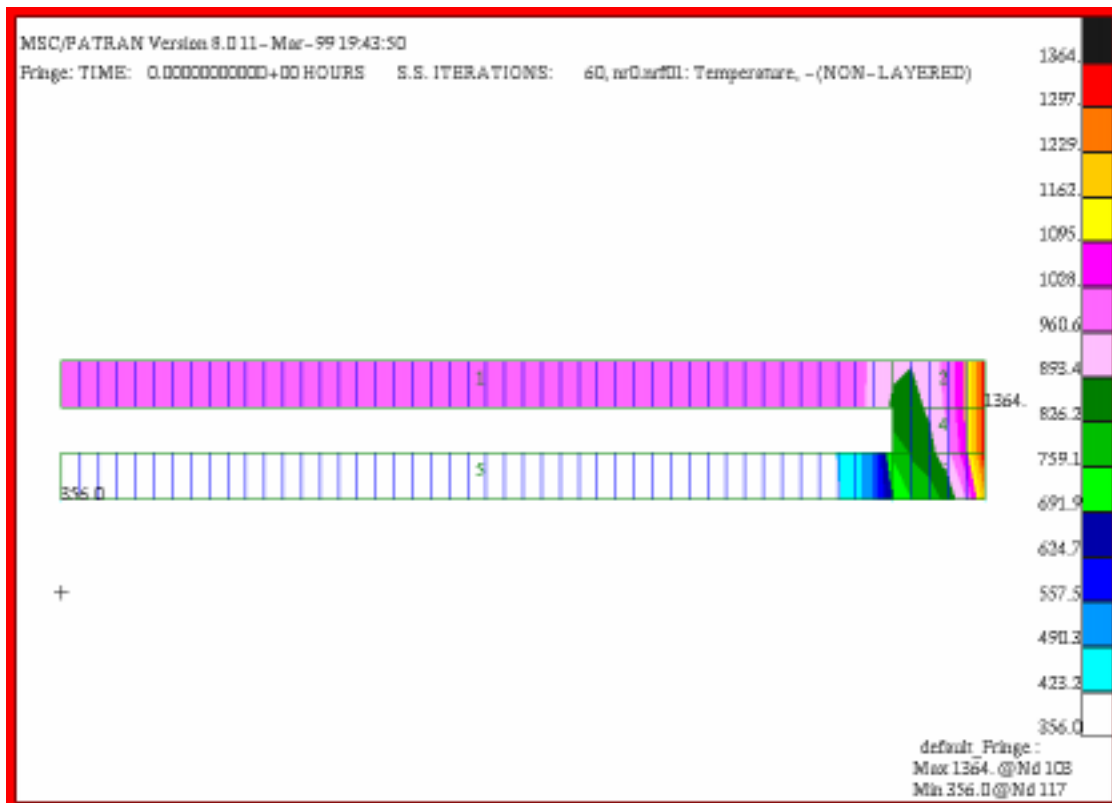


WORKSHOP 12

Analysis of a Fuel Nozzle Tip



Objective:

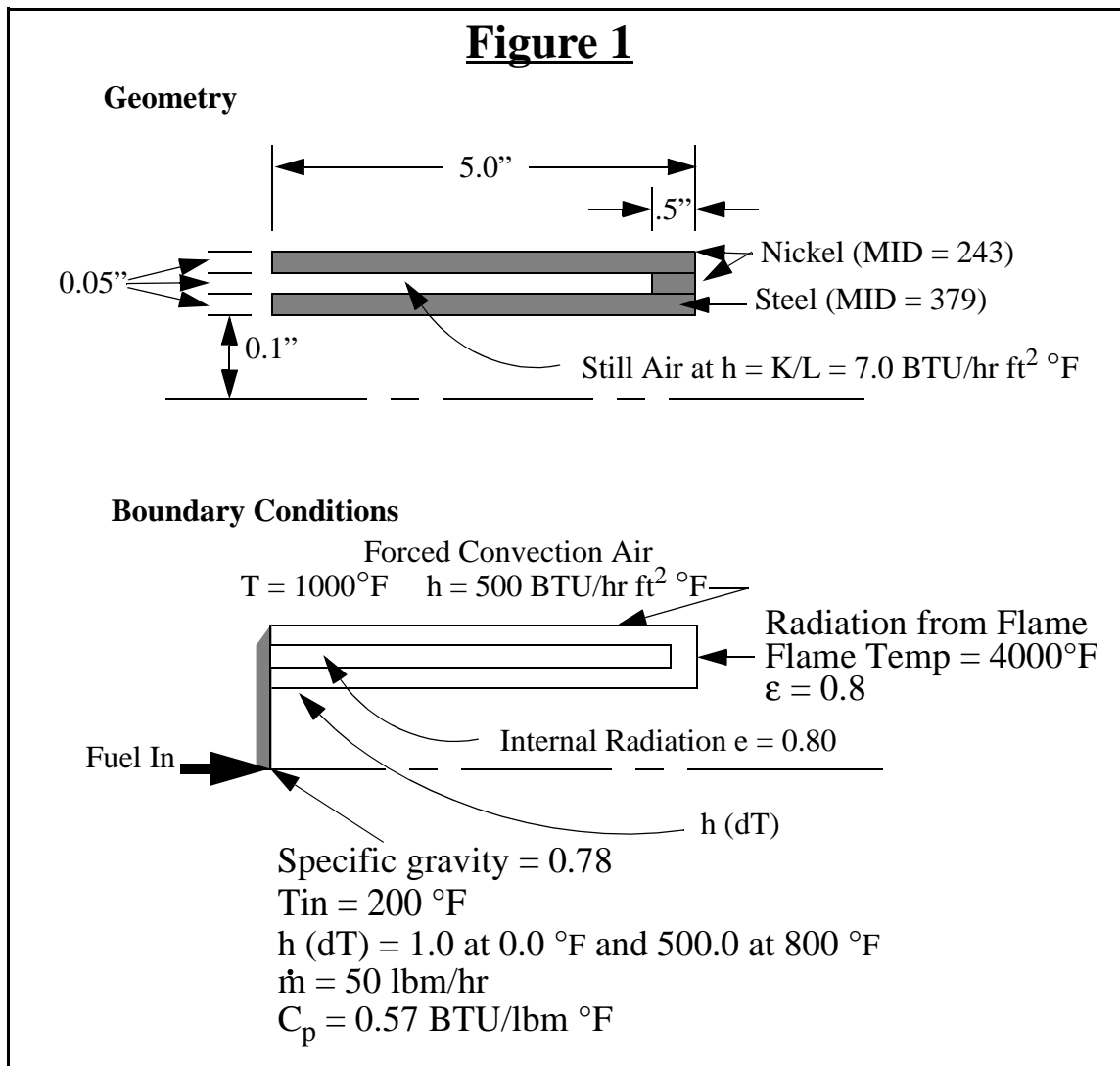
- Model an axisymmetric slice of a fuel nozzle tip.
- Apply advective, radiative, and convective boundary conditions.
- Run a steady state analysis and display results.



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.



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 0.100.
- 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.
- 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.
- Use the new **Analysis/Build Template** function to create 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 °F.
- Read and plot the results.
- **Quit** MSC.Patran.

Exercise Procedure:

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

<input type="button" value="File"/>	
<input type="button" value="New..."/>	
New Database Name	<input type="text" value="exercise_12"/>
<input type="button" value="OK"/>	

Open a new database

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	<input type="button" value="◆ Default"/>
Analysis Code	<input type="text" value="MSC/THERMAL"/>
<input type="button" value="OK"/>	

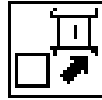
2. Create the nozzle, fluid stream, and Convective Quad geometry.

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*.

<input checked="" type="radio"/> Geometry	
<input type="button" value="Create/Surface/XYZ"/>	
<input type="checkbox"/> Auto Execute	<input type="text" value="<deselect>"/>
Vector Coordinates List	<input type="text" value="<4.5 0.05 0>"/>
Origin Coordinates List	<input type="text" value="[0 0.2 0]"/>
<input type="button" value="Apply"/>	

Create the nozzle and fluid stream geometry

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...

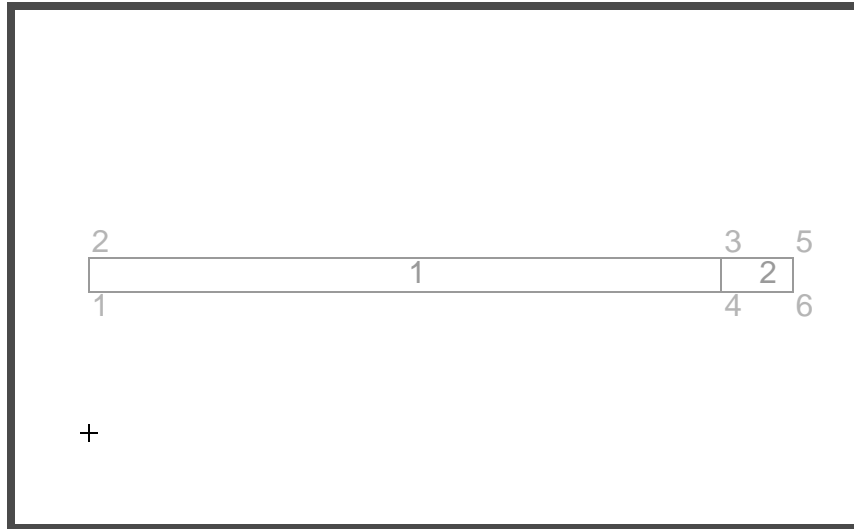
Model Y

5.0

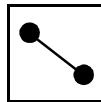
Apply

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 *Tip and base points* icon (shown above) in the **Select Menu** and select **Point 5** and then **Point 6** in the viewport>

Repeat Count

Auto Execute

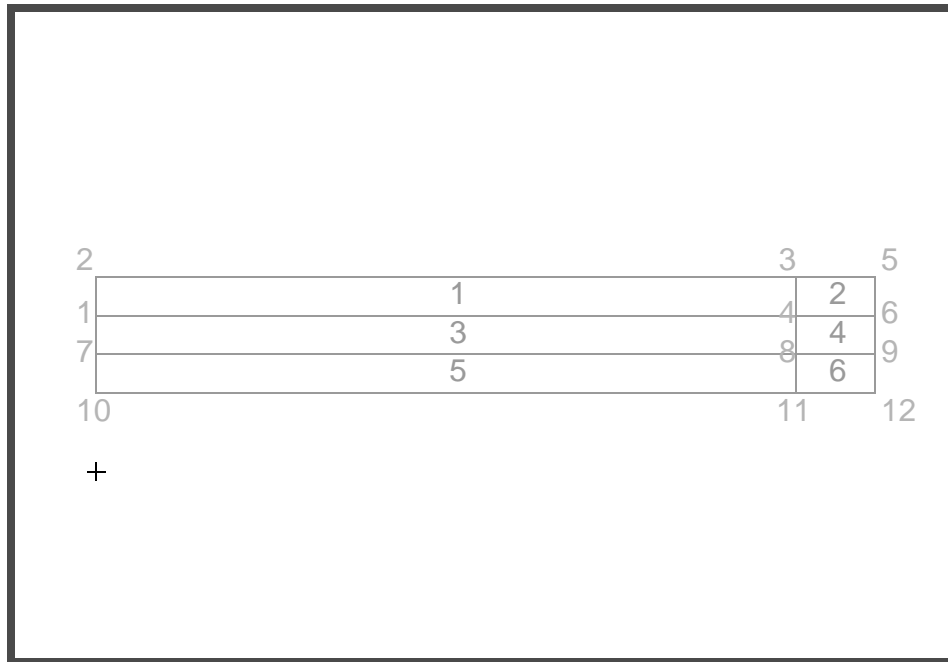
<deselect>

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, $\langle 4.5 \ 0 \ 0 \rangle$ and $[0 \ 0 \ 0]$ respectively.

◆ Geometry

Create/Curve/XYZ

Vector Coordinates List

$\langle 4.5 \ 0 \ 0 \rangle$

Auto Execute

$\langle \text{deselect} \rangle$

Origin Coordinates List

$[0 \ 0 \ 0]$

Apply

To create the second curve set the *Vector* and *Origin Coordinates List* to $\langle 0.5, 0, 0 \rangle$ and **Point 14** respectively.

Vector Coordinates List

$\langle 0.5 \ 0 \ 0 \rangle$

Origin Coordinates List

$\langle \text{select Point 14} \rangle$

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/Surface/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

Ending Curve List

<deselect>

<drag a rectangle around Curves 1 and 2>

<click once in *Ending Curve List* box, then change the Select Menu icon to *Select an Edge of a Surface* and select the lower edges of Surfaces 5 and 6>

Apply

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

◆ **Geometry**

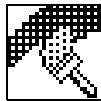
Delete/Any

Geometric Entity List

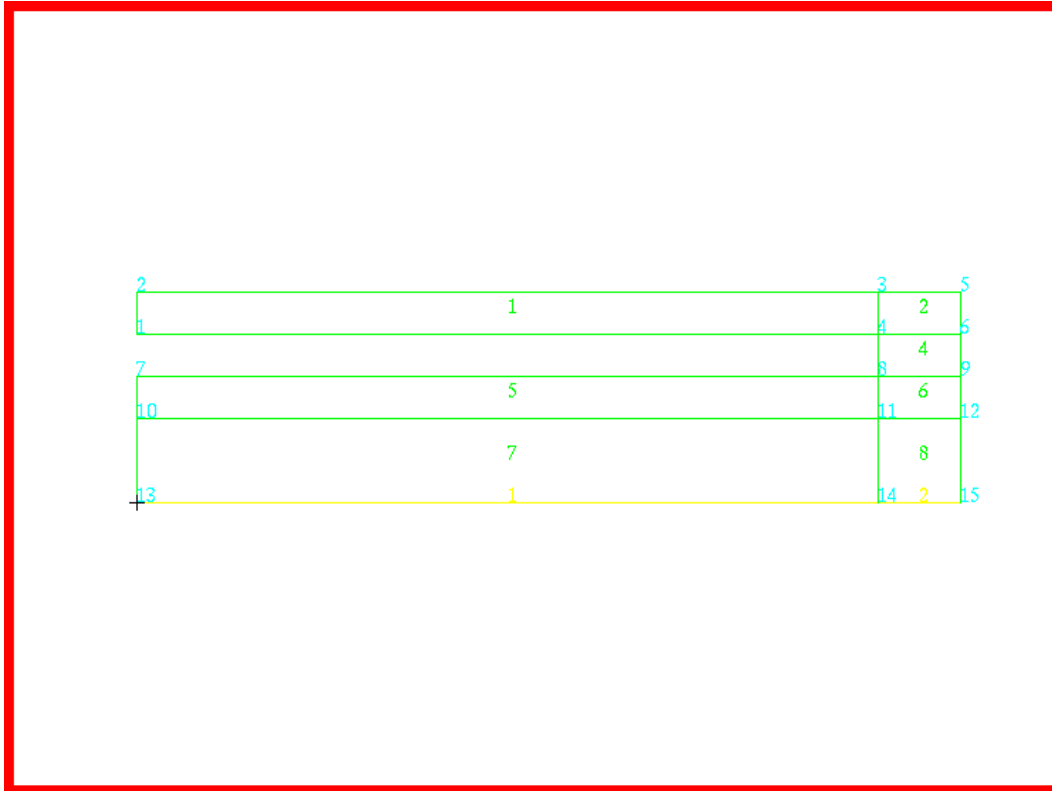
<Surface 3>

Apply

Refresh the graphics.



The resulting model is shown below.



3. Verify that surface normals are consistent with $R \times Z$. Reverse any surface normals which are not consistent with $R \times Z$.

Radiative boundary conditions modeled in an axisymmetric coordinate frame must have all element normals pointing in the $R \times Z$ (read R cross Z) direction. In this model, $R \times Z$ 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.



Verify surface normals and flow direction

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

<deselect>

<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

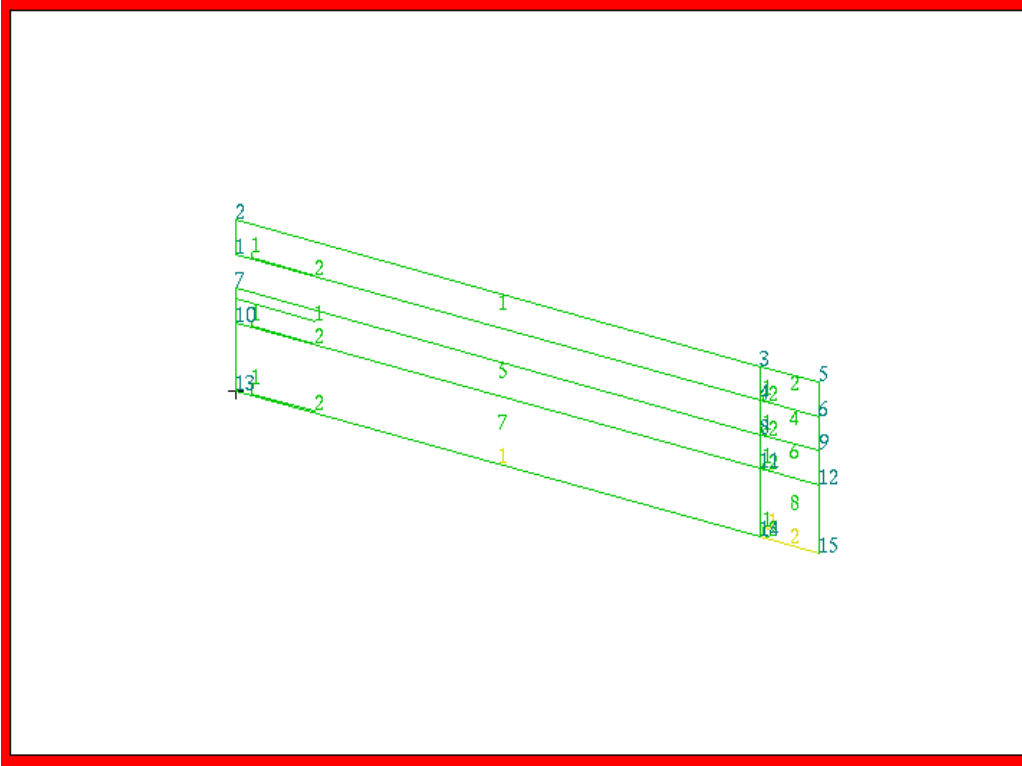
Geometry...

Show Parametric Direction

Apply

Cancel

The resulting display is shown below.



Return to default **Front** view.



Remove parametric directions display.

Display

Geometry...

Show Parametric Direction

Apply

Cancel

- 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.

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 0.100 and click in the *Surface List* box. Drag a rectangle around all surfaces in the viewport.

◆ Finite Elements	
Create/Mesh/Surface	
Global Edge Length	<input type="text" value="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	<input type="text" value="0.100"/>
Curve List	<select Curves 1 and 2 using the shift-left mouse button>
Apply	

- 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	<input type="text" value="998"/>
<input type="checkbox"/> Associate with Geometry	<deselect>
<input type="checkbox"/> Auto Execute	<deselect>
Node Location List	<input type="text" value="[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

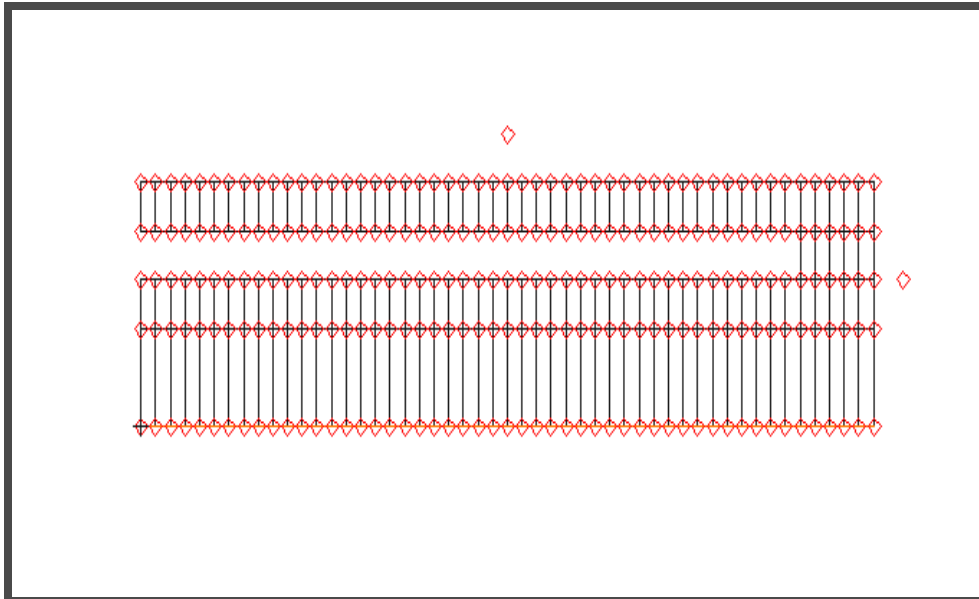
Apply

Cancel

or,



The display should now appear as shown below.



6. Equivalence the nodes at the mating surface edges.

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.

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



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

Equivalence nodes

Apply element properties to nozzle

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

- 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

Node 1 =

Point

<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

“Fluid Node”

OK

Select Members
(Click the Point Element from the select Menu.)

<drag a rectangle around the string of point elements 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, 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
Description - Property Table	fuel
Material Property ID (MPID)	1001
Temperature Units	Fahrenheit
Input Temperature Value	0.0 <cr>
Enter	
Value, Function	1.0 <cr>
Enter	
Input Temperature Value	800.0 <cr>
Enter	
Value, Function	500.0 <cr>
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 **Loads/BCs Applications radio button**. Create a fixed **1000°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 °F** applied to **Node 999**, located to the right of the nozzle and for a *New Set Name* **T_fuel** of **200°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	
Option	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 the *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 $k = 0.029 \text{ BTU/hr ft}^2 \text{ }^\circ\text{F}$ and $L = 0.05/12 \text{ ft}$. Hence $h = 7.0 \text{ BTU/hr ft}^2 \text{ }^\circ\text{F}$.

◆ **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* input box>

Select Surface or Edges

<select the bottom edge of Surfaces 1. (Surface 1.1)>

Add

<click in *Coupling Region* input box>

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**.

◆ Loads/BCs

Create/Radiation/Element Uniform

Option

Template, View Factors

New Set Name

Flame_rad

Target Element Type

2D

Input Data...

Enclosure ID

1

VFAC Template ID

10

Ambient Node ID

<select Node 999>

Can Be Obstructing Surface

<on by default>

OK

Select Application Region...

Geometry Filter

◆ Geometry

Select Menu

<select the *Edge* icon>



Select Surface or Edges

<select the right edges of Surfaces 2, 4, and 6, by using the shift-left mouse button. Do not include the right edge of Surface 8 (the convective quads)>

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

Ambient Node ID

<blank>

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

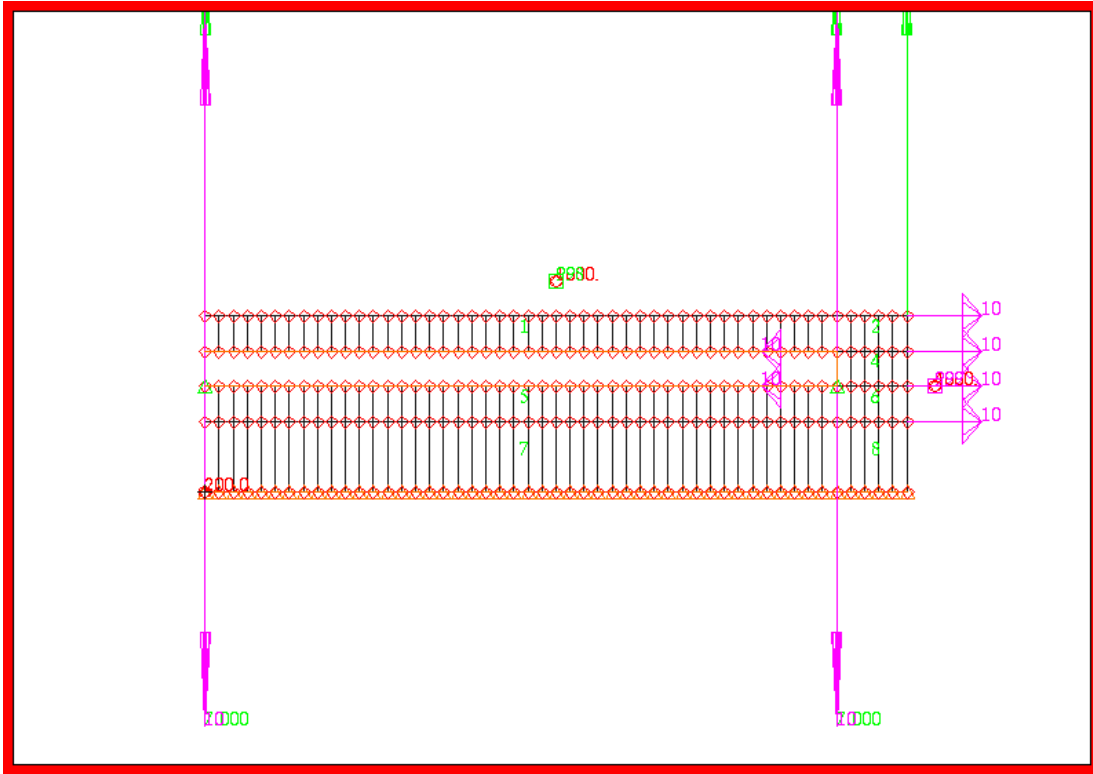
<Select the perimeter of the still air gap, Surface 1.1, 4.4, and 5.3 using the shift left mouse button.>

Add

OK

Apply

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



Create a group named 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.

Preferences

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>

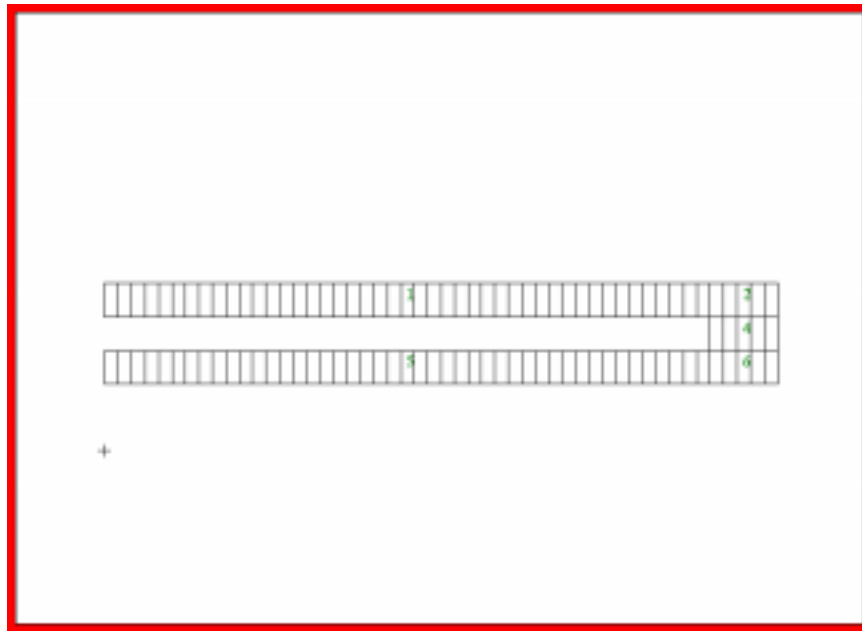
Apply

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

- Use the new **Analysis/Build Template** form to create a file named **template.dat.apnd** creating the CONV and 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 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 not enter any blank lines.

◆ Analysis	
Build Template	
Create Template File...	
Create/CONV/Data Entry	
CONV ID	10
CFG ID	30
MPID's	1001
Apply	
Cancel	<closes Template Entries form>
Create/VFAC/Data Entry	
VFAC ID	10
Emissivity	0.8
Apply	
Write File...	
OK	
Cancel	
Cancel	<closes Template File Data form>

The **template.dat.apnd** file should appear as follow:

```
CONV 10 30 0 1
1001
VFAC 10 0
0.8 1. 0 0 0. 0. 0 1
```

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
```

```
/
```

**In unix edit a
mat.dat.apnd
file**

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 °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 ndefined 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/FT²/R⁴

Initial Temperature =

1000.0

Initial Temperature Scale

◆ **Fahrenheit**

OK

OK

Output Requests...

Units Scale for Output Temperatures

Units Definition for Time Label

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

15. Read and plot the 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.

Read and plot results

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.

Directories

Available Files

Files

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

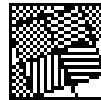
Select Result Cases

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 **F**ile on the *Menu Bar* and select **Q**uit from the drop-down menu.