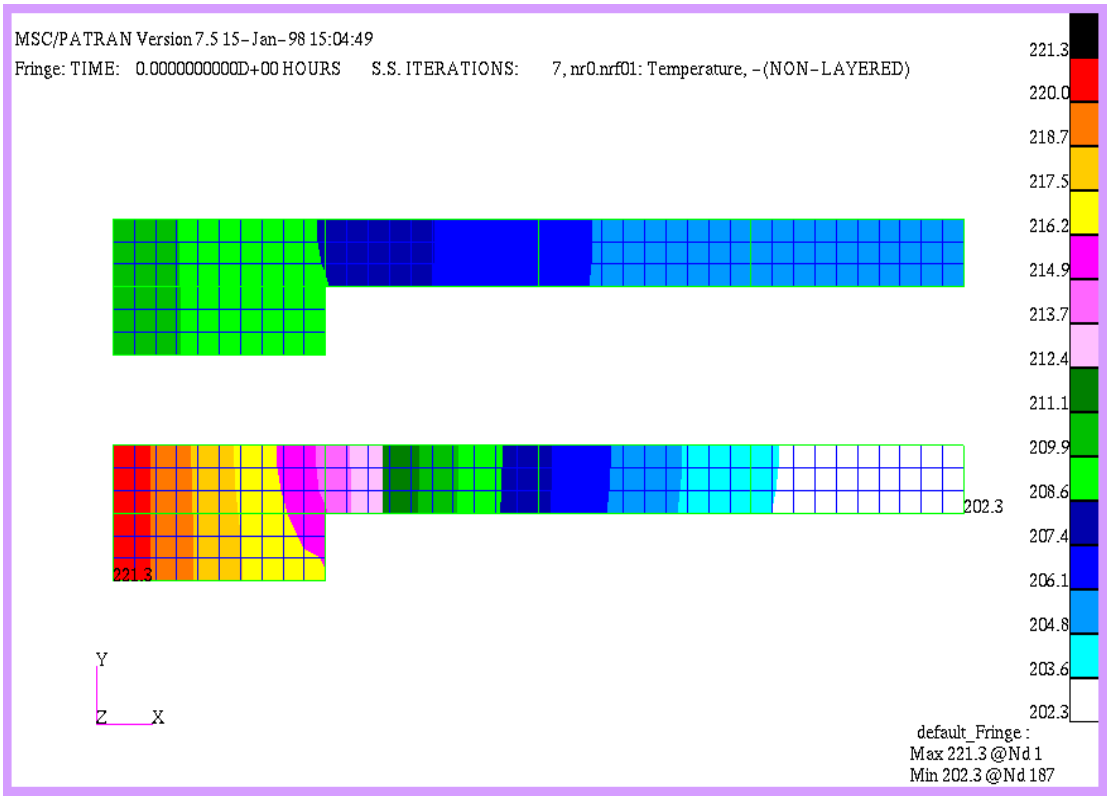
Exercise 6

Comparison of Two Heat Sink Designs



Objective:

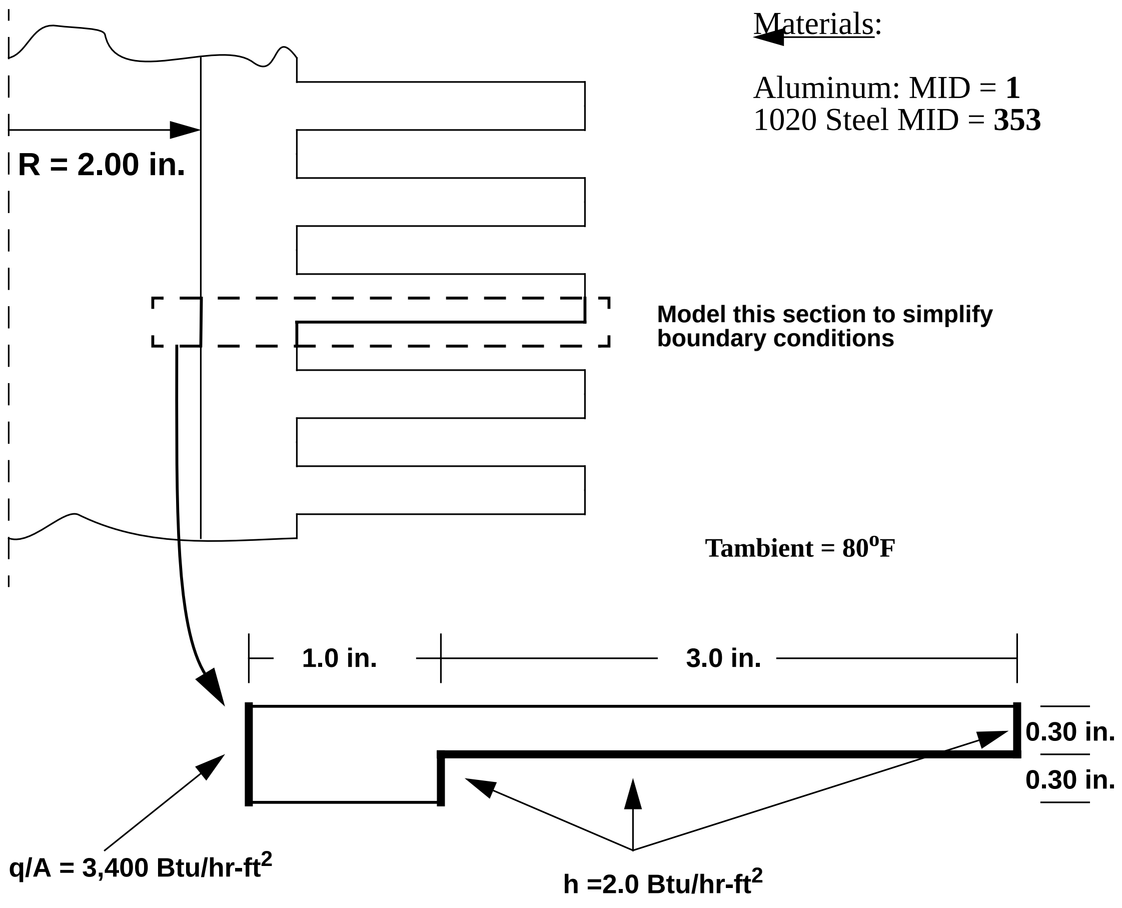
- Model two competing finned heat sinks.
- These will be 2D axisymmetric slices.

Model Description:

In this exercise you will model a section of a finned heatsink in order to compare the effect of using different materials.

The model is a representative fin section of a compressor cylinder casing. The maximum total heating on the casing interior has been determined. A choice of materials is open; the casing can be either aluminum or 1020 steel. Other than cost the only remaining discriminator is temperature; the interior casing surface must not exceed 212^{o}F. If both materials keep this surface at or below 212^{o}F steel will be used otherwise aluminum would be the material of choice. This analysis will determine the material choice.

Figure 1: Slice Through Compressor Cylinder Casing



Exercise Overview:

- Create a new database named **exercise_06.db.** Set *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**.
- Create the five surfaces which define the heat sink geometry.
- Transform the geometry to create the second heat sink.
- Mesh the surfaces with an **IsoMesh**.
- Create an ambient node **999**.
- Equivalence the nodes at the mating surface edges.
- Apply element properties to the elements using the MID's provided. These are **2D Thermal Axisymmetric** elements.
- Apply temperature, flux and convection boundary conditions.
- Prepare and submit the model for analysis specifying that it is an **Axisymmetric Geometry** model, that a units conversion is required, and that the direct solver will be used for analysis.
- Read the results file and plot results.
- Check the results against the requirement of 212^{o}F.
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_06.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_06.db to the new database by clicking in the *New Database Name* box and entering **exercise_06**.

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

File

New ...

New Database Name **exercise_06**

OK

MSC/PATRAN will open a Viewport and change various *Main Form* 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	

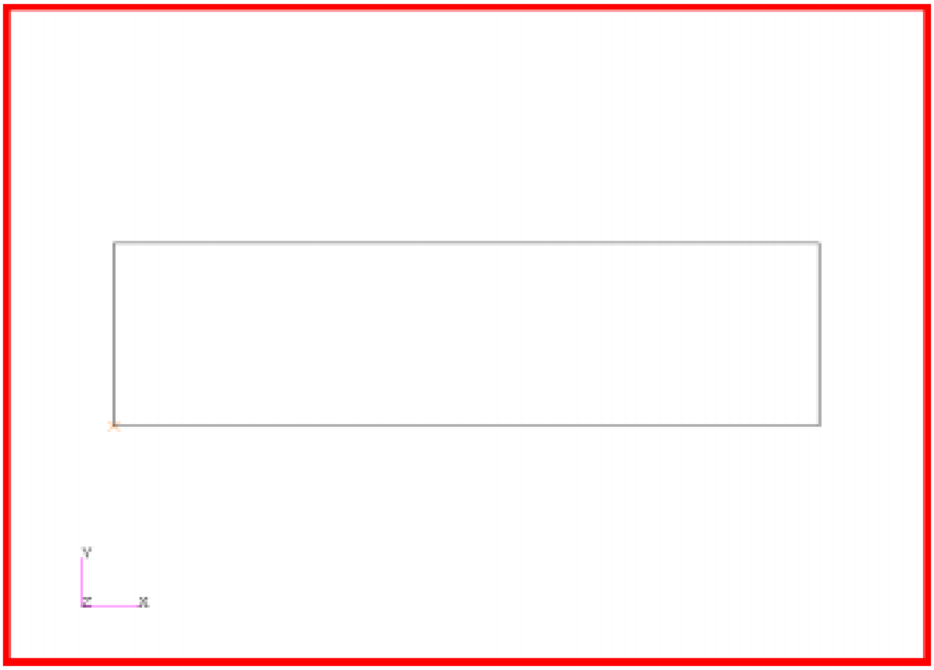
Create the heat sink surfaces

2. Create the five surfaces which define the model geometry.

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/Surface/XYZ	
Vector Coordinate List	**<1 0.3 0>**
Origin Coordinates List	**[2 0 0]**
Apply	

Surface 1 will appear in the viewport. As shown below



Remaining in the Geometry form change the *Action*, *Object*, and *Method* to **Transform** the existing surface into the required geometry. The *Translation Vector entries* are easily determined from Figure 1 and are included in the form entries below.

◆ **Geometry**

Transform/Surface/Translate

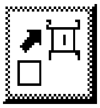
Translation Vector **<0 0.3 0>**

☐ **Auto Execute**

Surface List **<select Surface 1 in the viewport>**

Apply

Or, use *Show Label* icon to display labels.



Complete the fin by transforming the newly created **Surface 2**. Note that the *Repeat Count* is adjusted to **3** to create the full fin length.

◆ **Geometry**

Transform/Surface/Translate

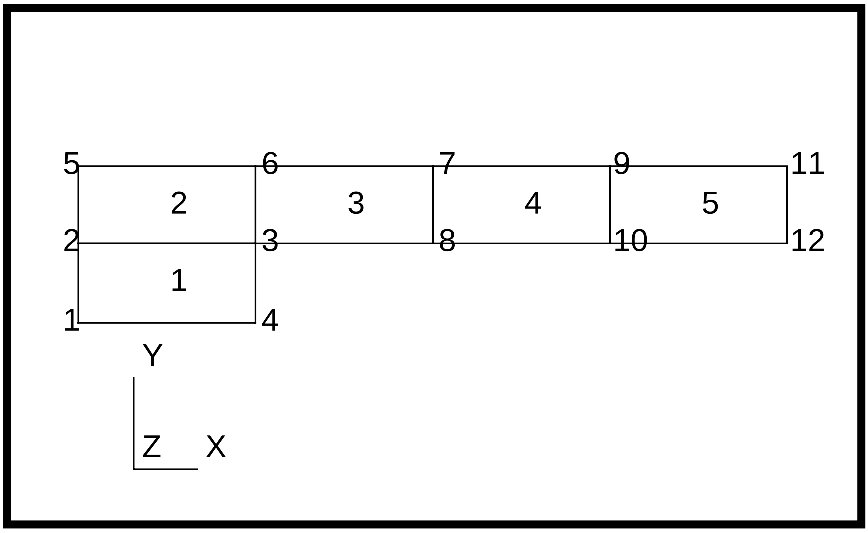
Translation Vector **<1 0 0>**

Repeat Count **3**

Surface List **<select Surface 2 in the viewport>**

Apply

The resulting model is shown below.



Create the second heat sink

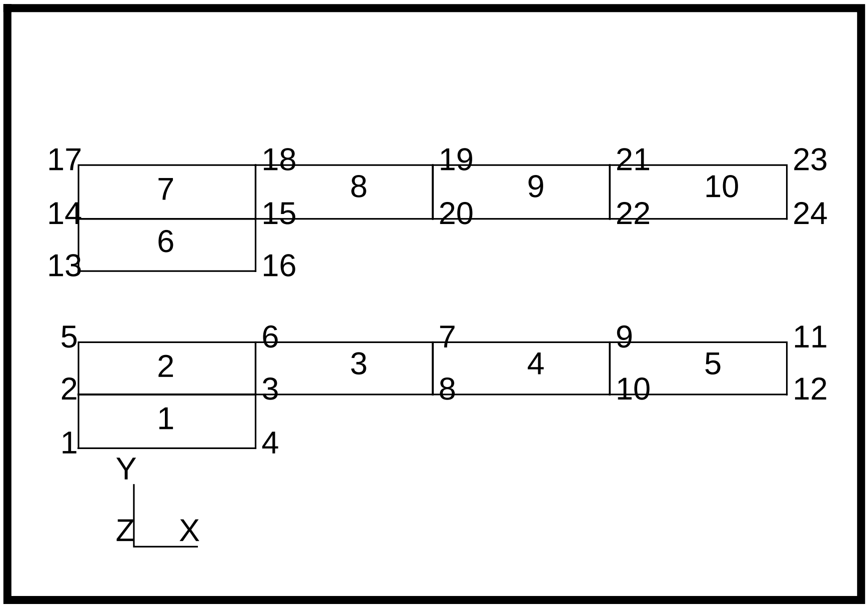
3. Transform the geometry to create the second heat sink.

Duplicate the entire heat sink cross section by transforming all the existing surfaces. Note that the *Repeat Count* is adjusted back to **1** to create the a single copy of the heat sink. The completed geometry is shown below.

◆ **Geometry**

Transform/Surface/Translate	
Translation Vector	**<0 1 0>**
Repeat Count	**1**
Surface List	**<drag a rectangle around all surfaces>**
Apply	

The display should now appear as shown below.



IsoMesh the surfaces

4. Mesh the surfaces with an IsoMesh.

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Accept the *Global Edge Length* of **0.1** and select all surfaces for inclusion in the *Surface List*.

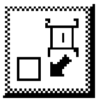
◆ **Finite Elements**

Create/Mesh/Surface

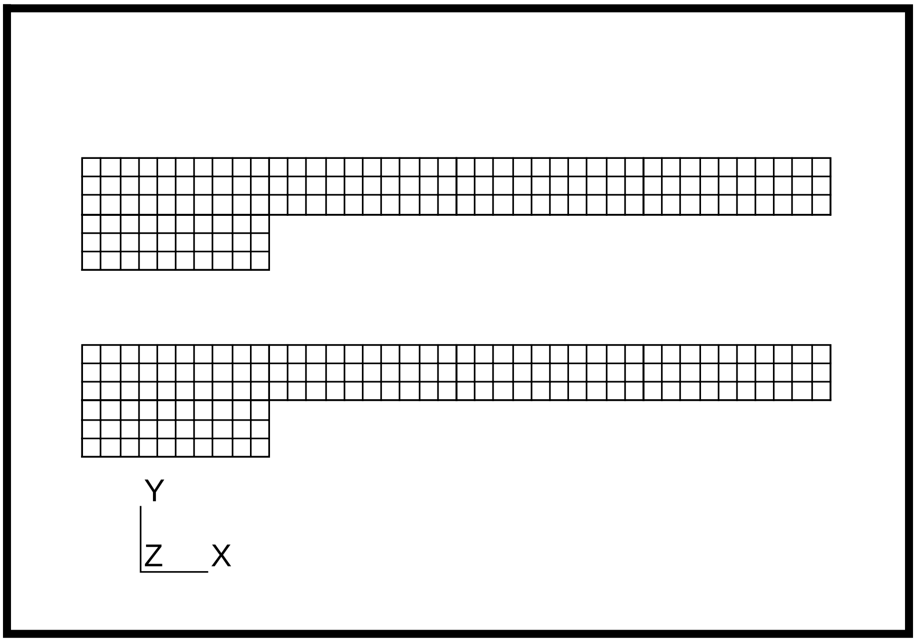
Surface List **<drag a rectangle around all surfaces>**

Apply

Use *Hide Label* Icons to turn off all labels.



The display should now appear as shown below.



5. Create an ambient node **999**.

Create an ambient node

Using the Finite Elements form create a boundary node which is not associated with geometry. The node is numbered **999**. Locate the node at **[6.2 0.9 0]** to the right of and between the two models.

◆ **Finite Elements**

Create/Node/Edit

Node ID List — **999**

☐ **Associate with Geometry**

Node Location List — **[6.2 0.9 0]**

Apply

Increase the display size of nodes to facilitate the application of boundary condition. Use either **Display/Finite Element...** or the associated Tool Bar *Node Size* icon to change the node size. The model should now appear as shown below.

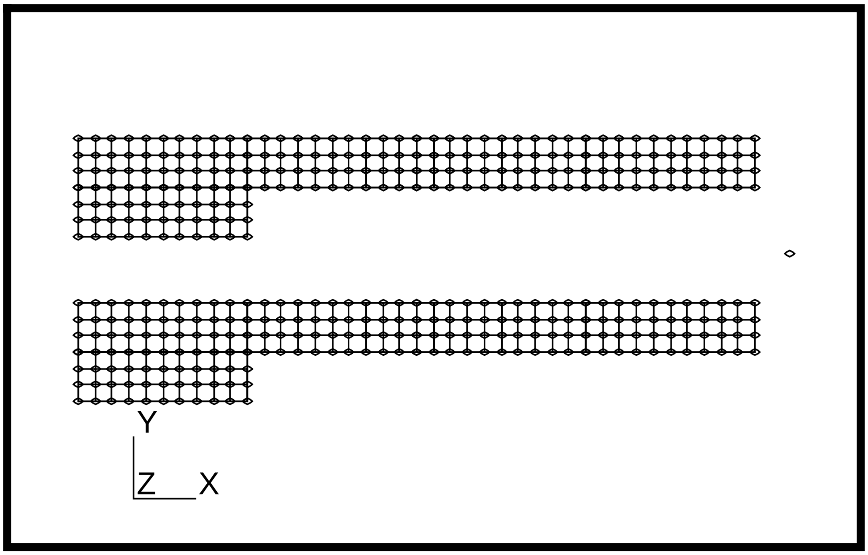
Display

Finite Element...

Node Size — **6**

Apply

Cancel



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

7. Apply element properties to the elements using the two material properties MID’s, **1** and **353**.

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. Use this database to facilitate the analysis.

Select the **Properties***Applications radio button*. Set the *Action*, *Dimension*, and *Method* to **Create/2D/Thermal Axisymmetric**. Enter *Property Set Name* **Steel**. Select the *Input Properties...* box. In the Input Properties form, click in the *Material Name* box and enter **353**. Select **OK** to close the form.Click in the *Select Members* box and drag a rectangle around the lower model in the viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ **Properties**

Create/2D/ Thermal Axisymmetric

Property Set Name	**Steel**
Input Properties...	
Material Name	**353**
Ok	
Select Members	**<drag a rectangle around the lower heat sink model surfaces>**
Add	
Apply	

Perform the same steps for upper model using, Aluminum, for the *Property Set Name*, and 1 for the *Material Name*.

8. Apply the temperature, convection, and flux boundary conditions to the model.

Apply boundary conditions

Begin applying boundary conditions. Select the **Load/BCs** *Applications radio button*. Create a fixed **80.0^{o}F** nodal boundary named **Ambient** In the Input Data form define the fixed temperature. In the Select Applications Region form pick node **999**.

◆ **Load/BCs**

Create/Temperature/Nodal

Option:	**Fixed**
New Set Name	**Ambient**

Input Data...

Fixed Temperature	**80.0**

OK

Select Application Region...

Geometry Filter	◆ **FEM**
Select Nodes	**<select node 999>**

Add

OK

Apply

Create the heat transfer coefficient boundary conditions with the **Use Correlations** option, set name **Air**, and a heat transfer coefficient of **2.0 Btu/^{o}F-hr-ft^2**. Apply the boundary condition to the exposed edges of both finned heat sinks as shown in Figure1. The same boundary condition is applied to both heat sink models.

◆ **Load/BCs**

Create/Convection/Element Uniform

Option:	**Use Correlations**
New Set Name	**Air**
Target Element Type	**2D**

Input Data...

In the Input Data form provide the heat transfer coefficient and fluid node.

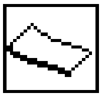
Convection Coefficient	**2.0**
Fluid Node ID	**999**

OK

Select Application Region...

In the Select Applications Region form select the right facing and bottom facing exposed edges of the finned heat sinks. Switch to the *Select an Edge* icon in the *Select Menu* form. When selecting the edges the edge chosen will be highlighted. Hold down the *<shift> key* and use the left mouse button to collect all the edges in the *Select Surfaces or Edges* box.

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



Select Surfaces or Edges	**<using shift-left mouse button select the edges highlighted in Figure 1 for both heat sink models>**

Add

OK

Apply

Create a set name **Flux** of **3400 Btu/hr-ft^2**. Apply the boundary condition to the left facing edges of both finned heat sinks as shown in Figure1. The same boundary condition is applied to both heat sink models.

◆ **Load/BCs**

Create/Heating/Element Uniform

Option:	**Fluxes**
New Set Name	**Flux**
Target Element Type	**2D**

Input Data...

Heat Flux	**3400**

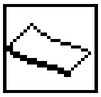
OK

In the Select Applications Region form select the left facing exposed edges of the finned heat sinks. Switch to the *Select an Edge* icon, if necessary, in the *Select Menu* form. When selecting the edges the edge chosen will be highlighted. Hold down the *<shift> key* and use the left mouse button to collect all the edges in the *Select Surfaces or Edges* box.

Select Application Region...

Geometry Filter — ◆**Geometry**

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



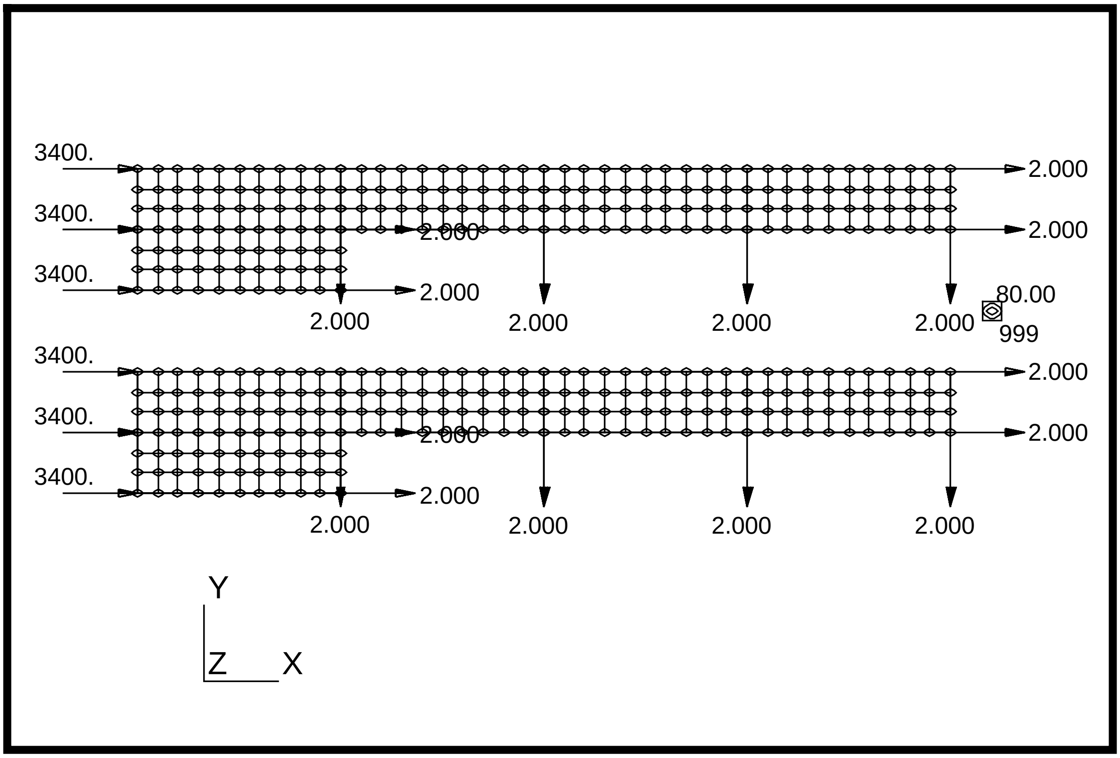
Select Surfaces or Edges — **<using shift-left mouse button select the edges shown in Figure 1 for both heat sink models>**

Add

OK

Apply

With boundary conditions applied the model should appear as shown below



9. 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.

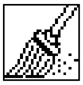
◆ Analysis	
Analyze/Full Model/Full Run	
Translation Parameters...	
Model Dimensionality	**◆ Axisymmetric Geometry, R Z Co-ordinates**
Radial, R Co-ordinate	**◆ X axis**
Centerline, Z Co-ordinate	**◆ Y axis**
Perform Geometry Units Conversion	
From Units	**inches**
To Units	**feet**
File to Extract Undefined Materials:	**3,mpidfph.bin (Btu-feet-lbm..**
OK	
Solution Type...	
OK	
Solution Parameters...	
Calculation Temperature Scale	**◆ Fahrenheit**
Solver Option	**2, Direct Solver**
Run Control Parameters...	
Initial Temperature	**212.0**
Initial Temperature Scale	**◆ Fahrenheit**
OK	
OK	
Output Requests...	
Units Scale for Output Temperatures	**◆ Fahrenheit**
Units Definition for Time Label	**Hours**
OK	

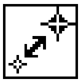
Submit Options...

OK

Apply

While waiting for the analysis to finish. *Reset Graphics* and reduce node size.





10. 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_06.db** database. The analysis created a new subdirectory with the same name as the *Job Name*; exercise_06/. 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_06**

Filter

Available Files **nr0.nrf.01**

OK

Select Rslt Template File...

Files **pthermal_1_nodal.res_tmpl**

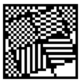
OK

Apply

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

◆ 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.

Which material will be chosen for the cylinder casing?

Quit MSC/ Patran

11. **Quit** MSC/PATRAN

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