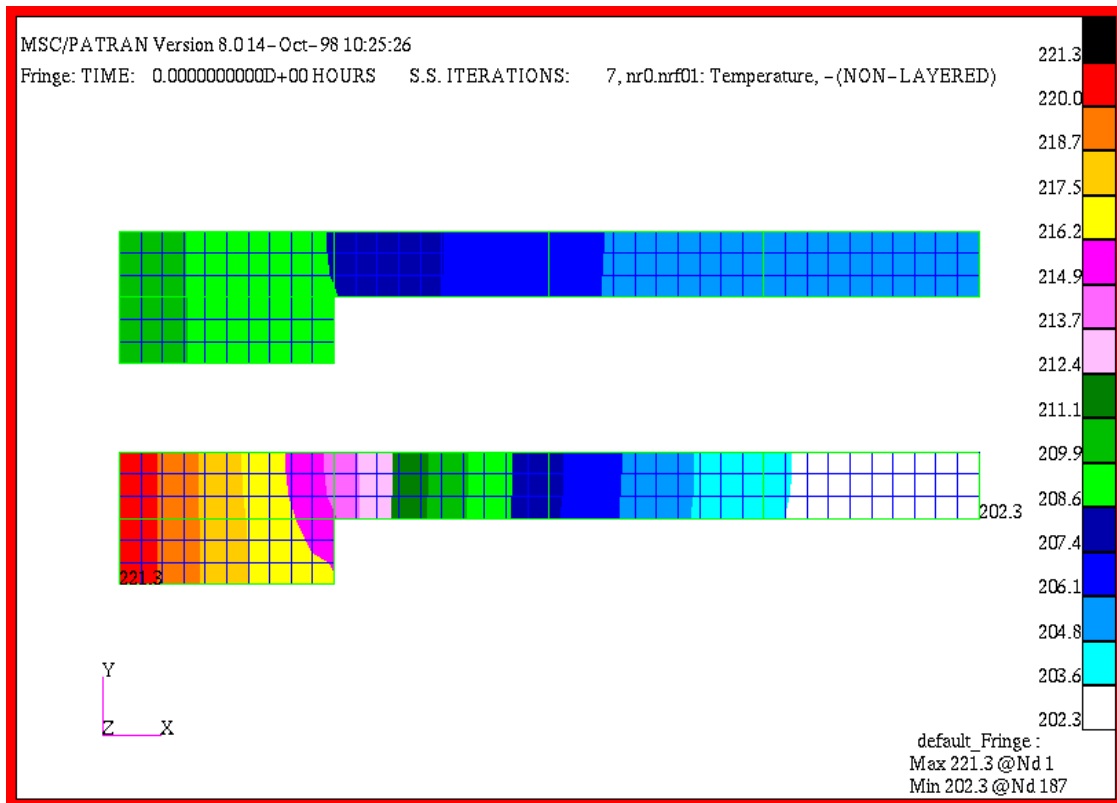


WORKSHOP 6

Comparison of Two Heat Sink Designs



Objective:

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

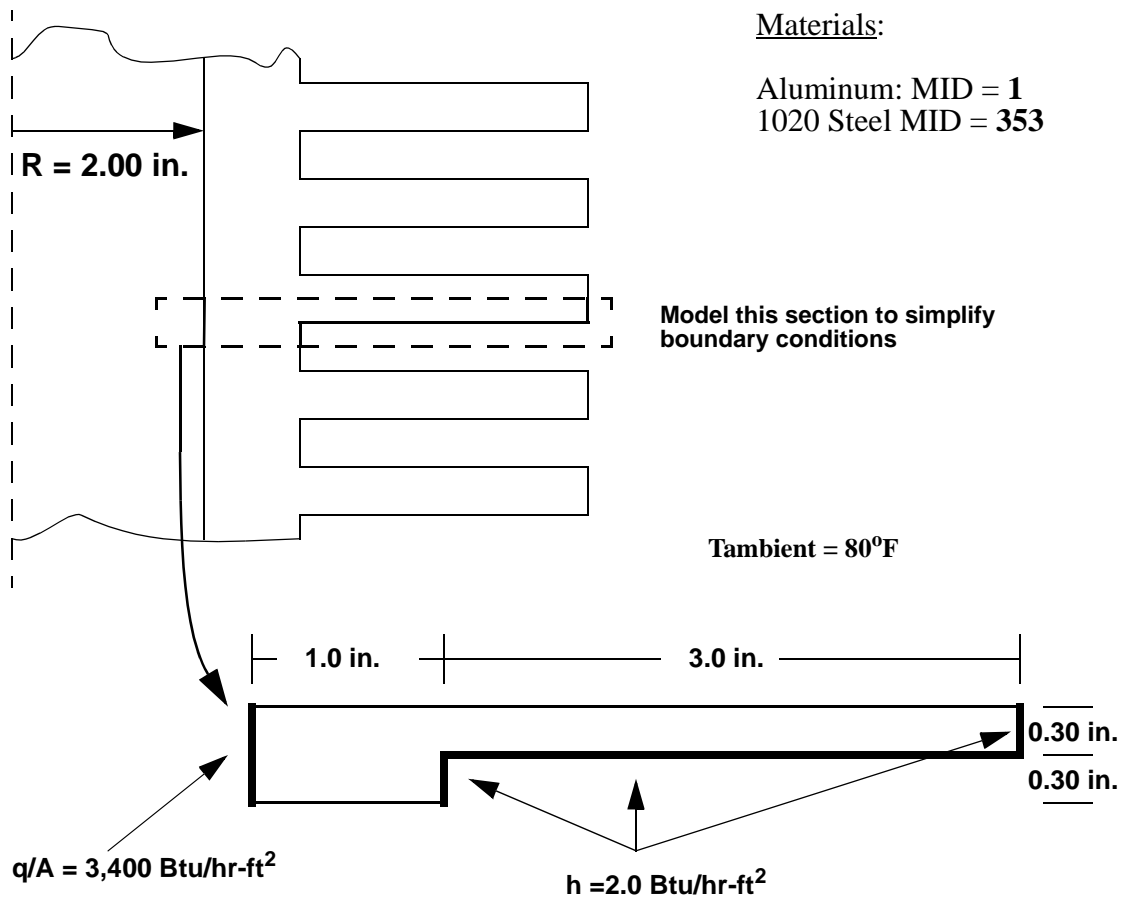


Model Description:

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°F. If both materials keep this surface at or below 212°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°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.



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	

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.

Create the heat sink surfaces

◆ 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

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

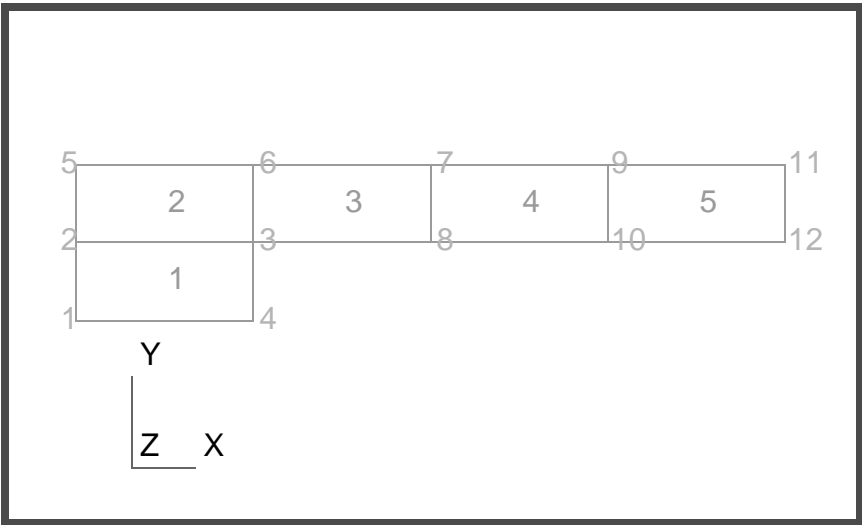
3

Surface List

<select Surface 2 in the viewport>

Apply

The resulting model is shown below.



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.

Create the second heat sink

◆ **Geometry**

Transform/Surface/Translate

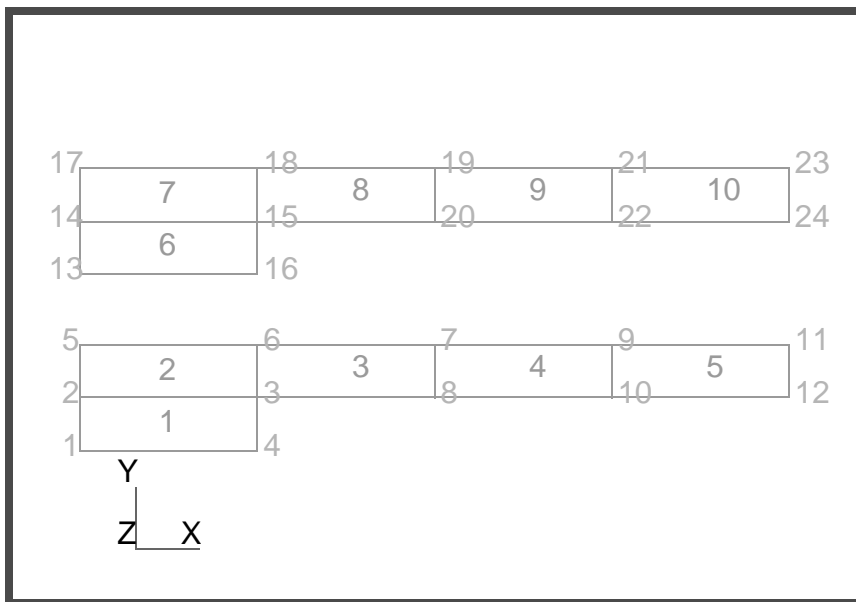
Translation Vector

Repeat Count

Surface List

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

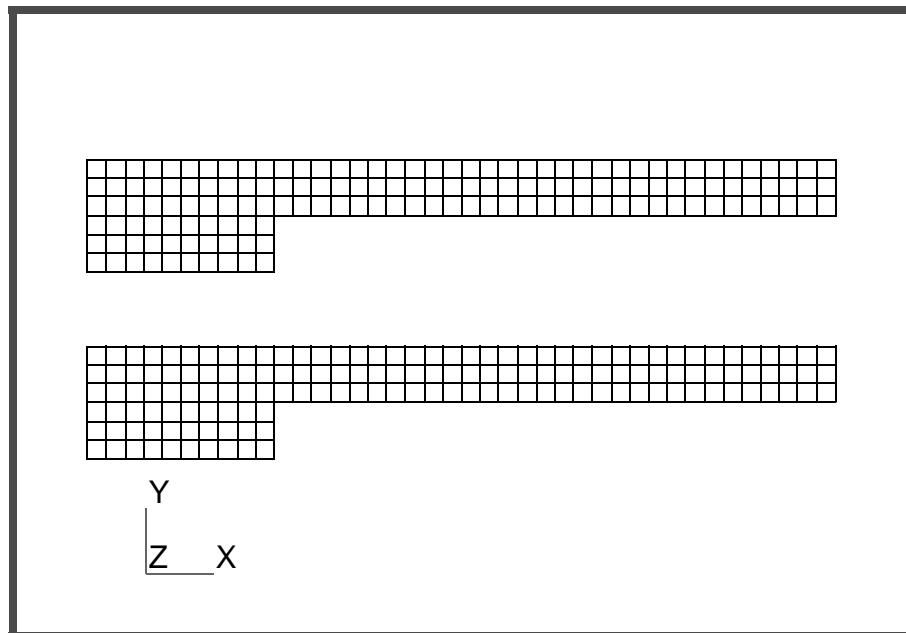
<drag a rectangle
around all surfaces>

Apply

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



The display should now appear as shown below.



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

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.

Create an ambient node

◆ Finite Elements

Create/Node/Edit

Node ID List

Associate with Geometry

Node Location List

Apply

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

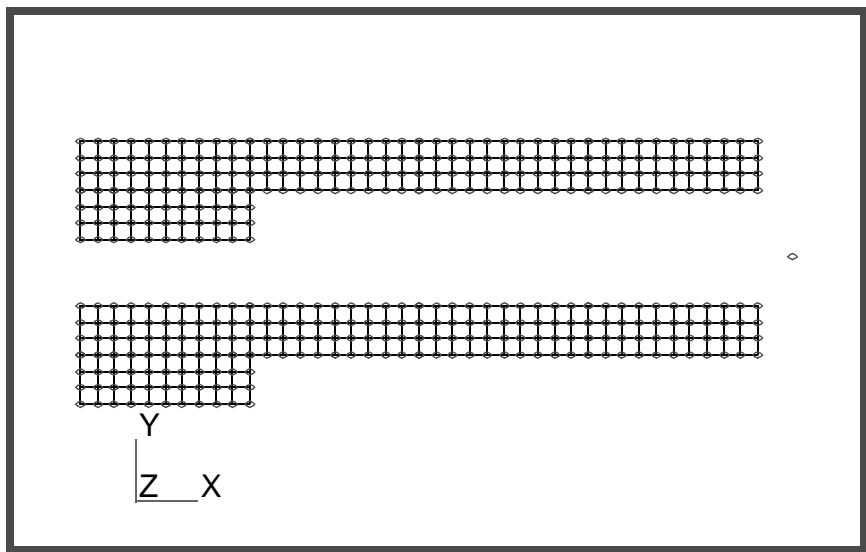
Display

Finite Elements...

Node Size

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 step 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*.

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

Apply boundary conditions

Begin applying boundary conditions. Select the **Loads/BCs Applications** radio button. Create a fixed **80.0°F** nodal boundary named **Ambient** in the Input Data form define the fixed temperature. In the Select Applications Region form pick node **999**.

◆ Loads/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 **Template, Convection** option, set name **Air**, and a heat transfer coefficient of **2.0 Btu/°F-hr-ft²**. 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.

◆ Loads/BCs	
Create/Convection/Element Uniform	
Option:	Template, Convection
New Set Name	Air
Target Element Type	2D
Input Data...	

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

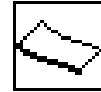
Convection Coefficient

Fluid Node ID

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

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>**

Create a set name **Flux** of **3400 Btu/hr-ft²**. Apply the boundary condition to the left facing edges of both finned heat sinks as shown in Figure 1. The same boundary condition is applied to both heat sink models.

◆ Loads/BCs

Option:

New Set Name

Target Element Type

Heat Flux

In the Select Application 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

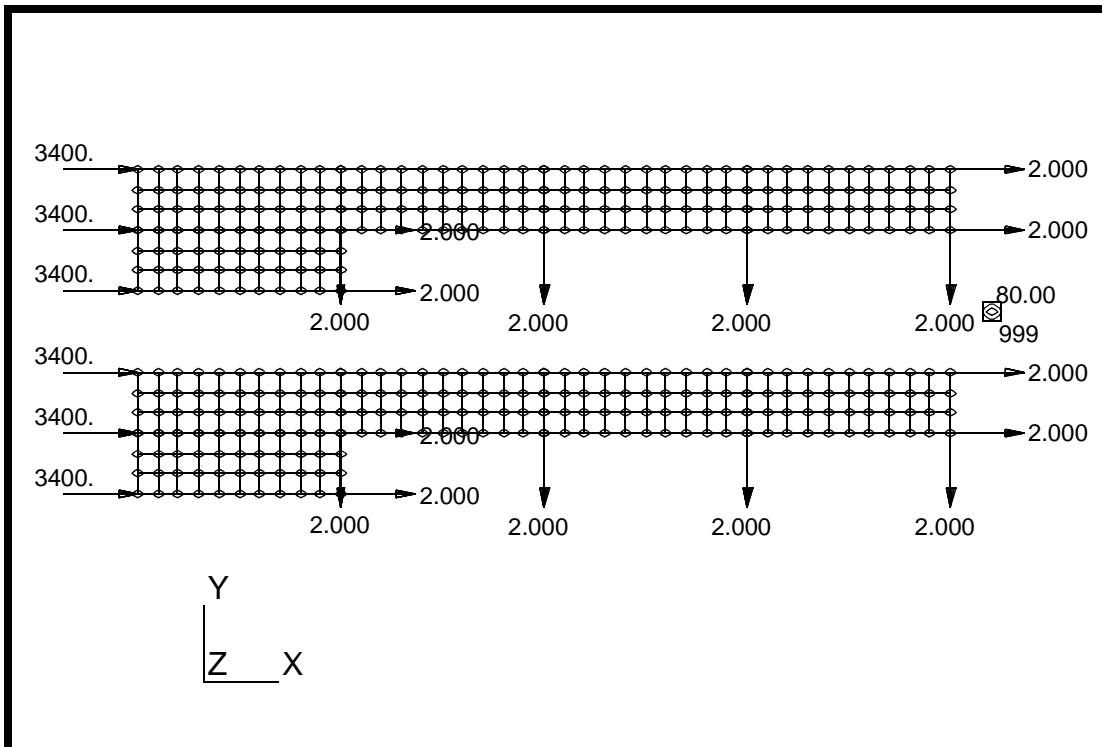
**<using shift-left mouse button
select the edges shown in Figure 1
for both heat sink models>**

Add

OK

Apply

With boundary conditions applied the model should appear as shown below



Prepare and run analysis

9. Prepare and submit the model for 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 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.

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_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.

Read and plot results

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories

Filter

Available Files

OK

Select Rslt Template File...

Files

OK

Apply

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

◆ **Results**

Create/Quick Plot

Select Result Cases

TIME: 0.000000000D+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?

11. Quit MSC.Patran

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

**Quit
MSC.Patran**