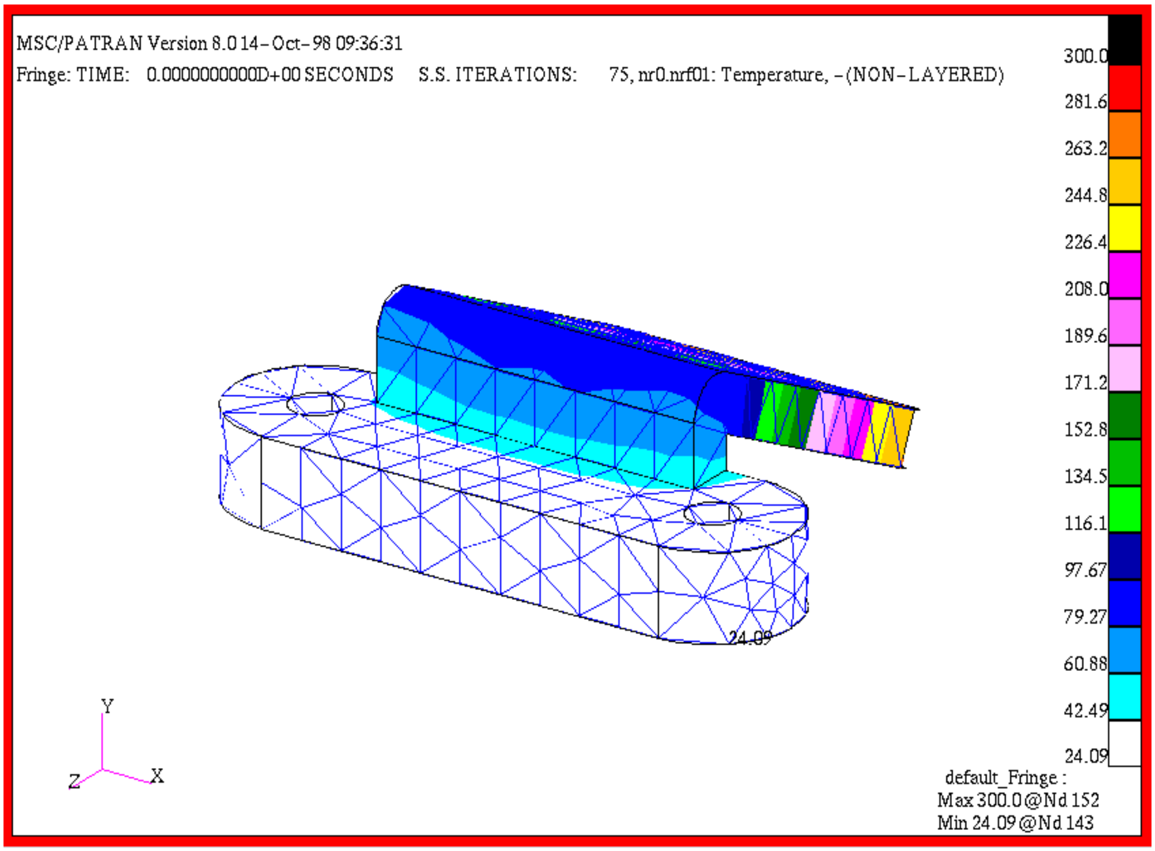
WORKSHOP 5

Thermal Analysis using Imported CAD Geometry



Objective:

- In this exercise you will complete a thermal analysis of a model created from imported CAD geometry.

Model Description:

In this exercise analyze an oven lid clamp. The clamp geometry (in centimeters) will be imported as ProEngineer geometry; from it, create a **B-rep** solid. Use the Auto **TetMesh** *Mesher* to mesh the solid. Apply boundary conditions, complete the analysis and review the results.

This stainless steel (MID 364) clamp is used to clamp the perimeter flange on a pressurized processing oven lid. The oven lid surface can reach 300^{o}C for several days. The lid is insulated; the insulation is sometimes pierced by the clamp edge. The clamp mounting boss is fastened with two bolts and thermal grease (total contact h = 0.01 w/^{o}C-cm^{2}) to a room temperature (20^{o}C) water cooled sink.

Determine both that the bracket mounting boss will remain at or below 50^{o}C to ensure safe handling during disassembly and that the spring tab knee and boss transition areas remain at or below 150^{o}C to prevent loss of clamping force due to creep.

This exercise will introduce a different format for guiding data entry, keystrokes, and mouse operations. Though all actions and entries required to accomplish a given step are provided some additional synthesis may be required by the user since exact images of the entry forms are not provided.

Exercise Overview:

- Open a new database named **exercise_05.db**.
- Import Pro/ENGINEER primitive geometry from a file named oventab.geo.
- Create a **B-rep** solid from these surfaces and delete the original surfaces in the process.
- Mesh the solid with the **TetMesh** *Mesher* using **Tet4** elements, a *global edge length* of **2.5.**.
- Define an element property over all the solid elements using a material name of **364**.
- Create a boundary sink node **999** below the mounting boss and not associated with geometry.
- Change the view for application of boundary conditions
- Apply a 20^{o}C fixed temperature to the sink node.
- Apply a fixed temperature of 300^{o}C to the edge of the solid in contact with the lid.

- Apply a convection boundary condition of 0.01 w/^{o}C-cm^2 to the underside of the mounting boss.
- Select the **mpidcgs.bin** file in the P/Thermal Translation Parameters form in order to select the correct material property units.
- Run the analysis and read the results into the database.
- Fringe plot the temperature results and evaluate them against the requirements.
- **Quit** MSC.Patran.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_05.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_05.db to the new database by clicking in the *New Database Name* box and entering **exercise_05** (.db will automatically be appended).

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

File

New...

New Database Name **exercise_05**

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

Import CAD geometry

2. Import Pro/ENGINEER primitive geometry from a file named **oventab.geo**.

Select **File** from the *Menu Bar* and select **Import…** from the drop-down menu. Change the *Object*, *Source*, and *File Type* list boxes as shown below. It may be necessary to select a path and use the *Filter* button to locate the **oventab.geo** file which should be contained in your home directory.

File

Import...

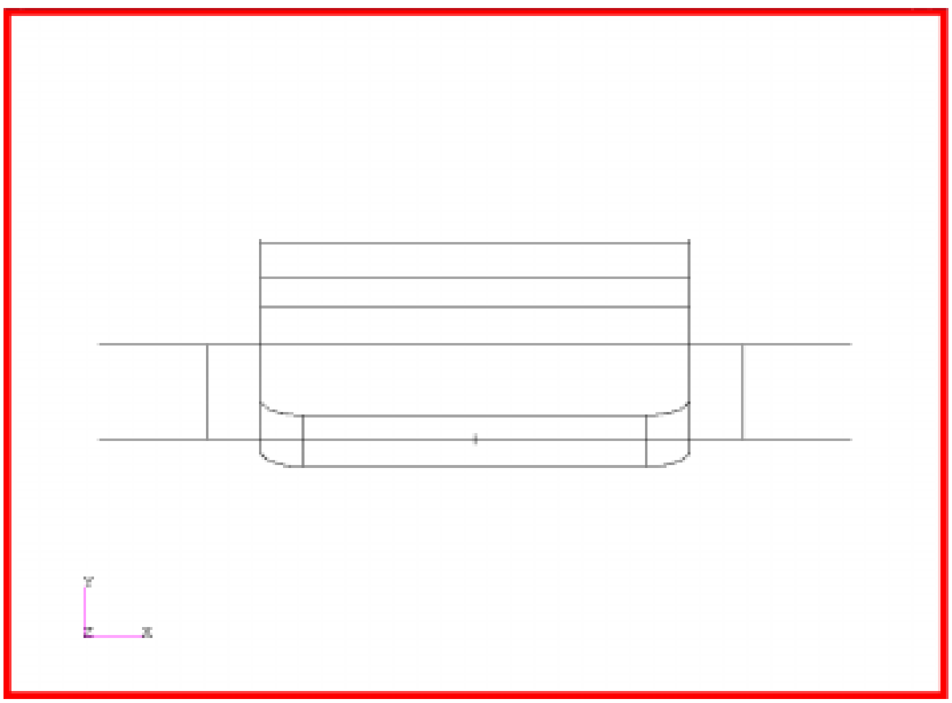
Object	**Model**
Source	**Pro/ENGINEER**
File Type	**Primitive Geometry**
Pro/ENGINEER Files	**oventab.geo**

Apply

The model geometry will be imported. A Pro/ENGINEER Model Import Summary form will provide statistics on the entity type and quantity imported. Click **OK** to close this form.

OK

The display should appear as shown below.



Select **Viewing** from the *Menu Bar* or use the Tool Bar *Iso 1 View* con to change to an isometric_view.

Preference

Graphics...

Auto Fit View

Apply

Cancel

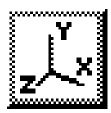
Viewing

Named View Options...

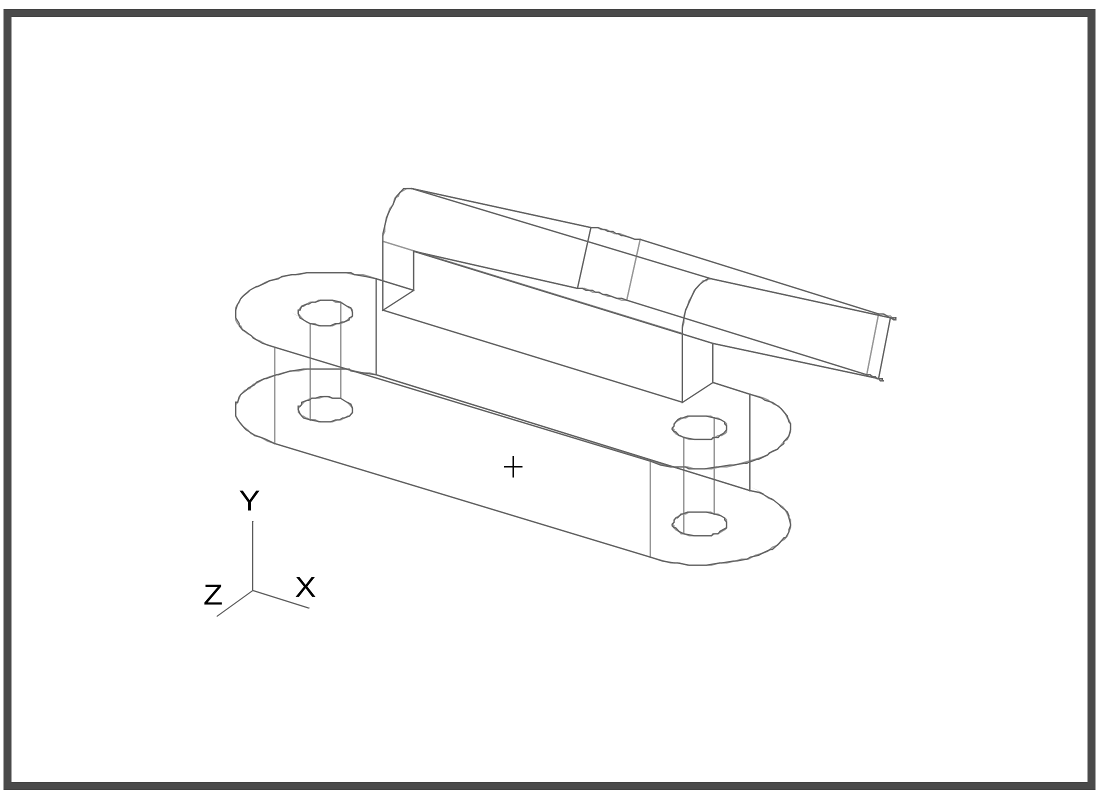
Select Named View **isometric_view**

Close

Or, use the Tool Bar *Iso 1 View* Icon.



The model should appear as shown below.



3. Create a B-rep solid from these surfaces and delete the original surfaces in the process.

Create a B-rep solid

Select the **Geometry** *Applications radio button*. Create a B-rep solid using the following *Action*, *Object*, and *Method***.**

◆ **Geometry**

Create/Solid/B-rep

■ **Delete Original Surfaces**

□ **Auto Execute**

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

Apply

A message window will request confirmation of deletion. Select **Yes**.

Yes

B-rep solid 1 is displayed as white in the viewport.

4. Mesh the solid with the **TetMesh** *Mesher* using **Tet4** elements, a *global edge length* of **2.5**.

TetMesh the B-rep solid

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Solid**. The Isomesh Mesher is used on regular parametric solids. In order to mesh this B-rep solid use the **TetMesh** *Mesher*. Use the default **Tet4** topology and adjust the *Global Edge Length* and **TetMesh Parameters** to reduce the mesh resolution for this analysis.

◆ **Finite Elements**

Create/Mesh/Solid

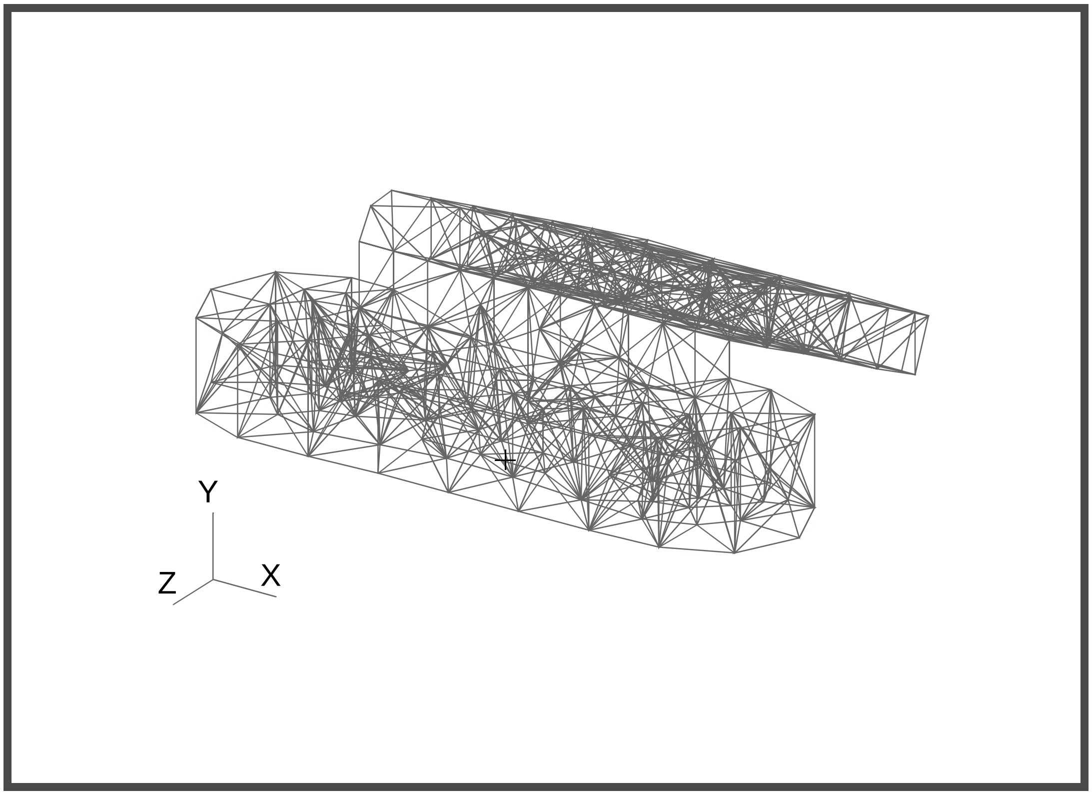
Global Edge Length **2.5**

Mesher ◆ **TetMesh**

Input List **<select Solid 1>**

Apply

Your model should appear as shown below.



Apply element properties to the elements

5. Define an element property over all the solid elements using a material MID of **364**.

Select the **Properties** *Applications radio button*. Set the *Action*, *Dimension*, and *Type* to **Create/3D/Thermal 3D Solid**. In the Input Properties form enter an MID of **364** for the *Material Name* and select **Solid 1** as the *Select Member* region.

◆ Properties	
Create/3D/Thermal 3D Solid	
Property Set Name	**Stainless_steel**
Input Properties...	
Material Name	**364**
OK	
Select Members	**<select Solid 1>**
Add	
Apply	

6. Create a boundary sink node 999 below the mounting boss and not associated with geometry.

Create a boundary sink node

Select the **Finite Elements** *Applications radio button*. Create a boundary node which is not associated with geometry. The node is numbered **999**. Locate the node at **[0 -5 0]** centered below the mounting boss.

◆ Finite Elements

Create/Node/Edit

Node ID List **999**

☐ Associate with Geometry

Node Location List **[0 -5 0]**

Apply

7. Increase node display size and change the view to a Y-Z, **side_view**. Rotate the view to show the bottom surface of the mounting boss.

Increase node size and change to a Y-Z view

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

Display

Finite Elements...

Node Size (use slider bar) **6**

Apply

Cancel

Select **Viewing** from the *Menu Bar* to change to a **side_view** of the model. Alternately, this step can be completed using the Tool Bar *Right Side View* icon.

Viewing

Named View Options...

Select Named View **side_view**

Close

Using **Viewing/Transformations...** from the drop down menu to change the view point by tilting the 15^{o} around the -Z axis to show the bottom surface of the mounting boss.

Viewing

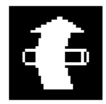
Transformations...

Options...

Rotation increment (deg) | **15 <use slider bar>**

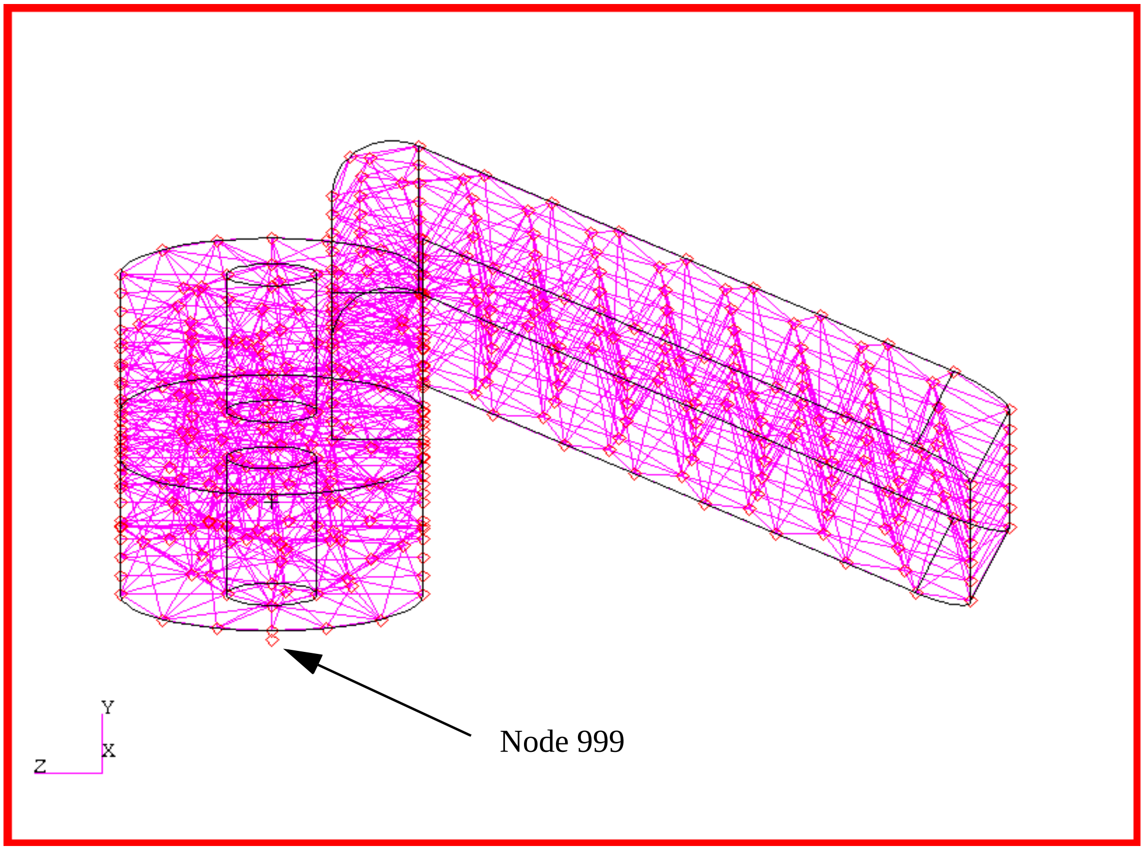
OK

<select this icon once for a 15^{o} -Z rotation>



OK

The model should appear as shown below. Note location of Node 999.



8. Fix the boundary node temperatures at 20.0^{o}C.

Fix nodal boundary temperature

Begin applying boundary conditions. Select the **Loads/BCs** *Applications radio button*. Create a fixed **20.0^{o}C** nodal boundary named **Sink**.

◆ **Loads/BCs**	
Create/Temperature/Nodal	
Option:	**Fixed**
New Set Name	**Sink**
Input Data...	

In the Input Data form define the fixed temperature.

Fixed Temperature	**20.0**
OK	
Select Application Region...	

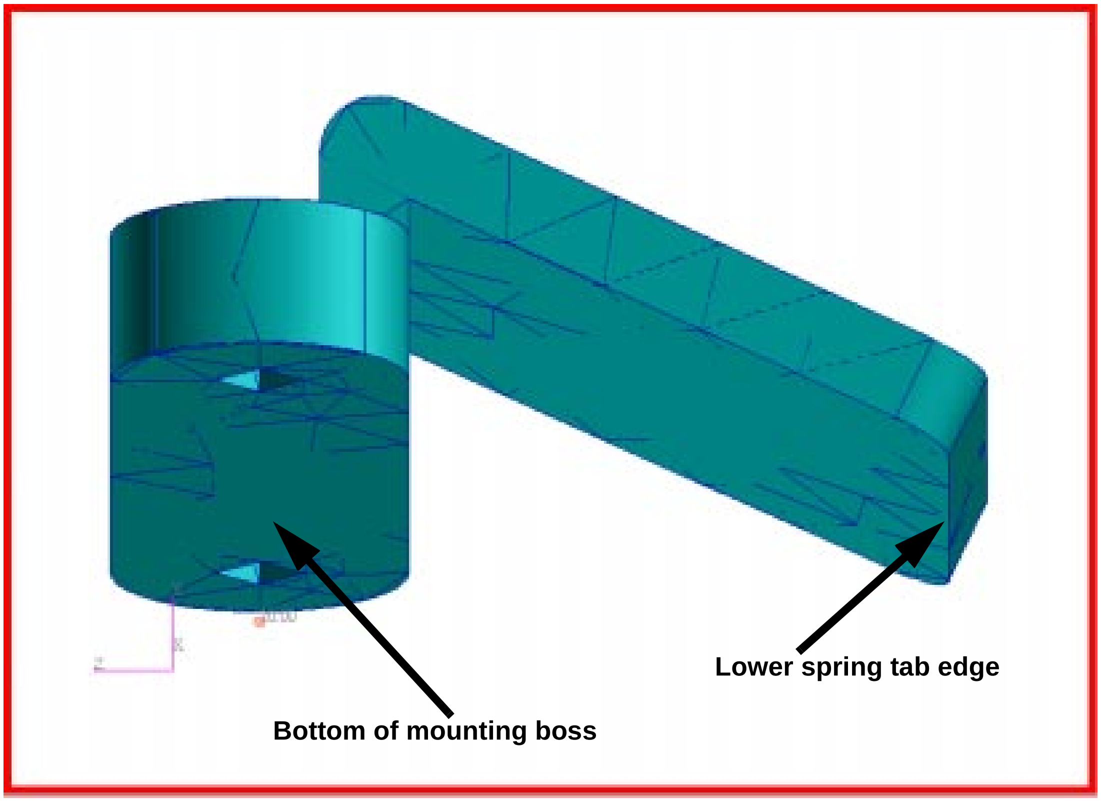
In the Select Applications Region form pick node **999**.

◆ **FEM**	
Select Nodes	**<select node 999>**
Add	
OK	
Apply	

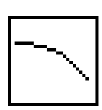
In order to facilitate applying the next two boundary conditions change the display. Select **Display** then **Entity Color/Label/Render ...** Change *Render Style* to **Shaded/Flat** or use the Tool Bar *Smooth Shaded* icon to affect the change.

Display	
Entity Color/Label/Render...	
Render Style	**Shaded/Flat**
Apply	
Cancel	

The display should appear as shown below. The lower contact edge of the spring tab, and the bottom of the mounting boss should now be visible.



Apply the fixed edge temperature. Enter a *New Set Name* **Edge** with a fixed temperature of **300.0°C** applied to lower edge of the spring tab.

New Set Name	**Edge**
Input Data...	
Fixed Temperature	**300.0**
OK	
Select Application Region...	
◆ **Geometry**	
Select Geometry Entities	<Choose *Select a Curve* icon, shown, from the *Select Menu***>**
	**<select lower spring tab linear edge, Solid 1.15.3.>**
Add	
OK	
Apply	

The display should highlight each node and append the fixed temperature. On some displays the symbol and value may be difficult to discern.

Apply contact heat transfer coefficient

9. Apply contact heat transfer coefficient.

Create the contact heat transfer coefficient boundary conditions with the **Template, Convection** option and the heat transfer coefficient provided, 0.01 w/^{o}C-cm^2. Name the set **contact** and apply the boundary condition to the surface on the bottom of the mounting boss.

◆ **Loads/BCs**

Create/Convection/Element Uniform

Option:	**Template, Convection**
New Set Name	**contact**
Target Element Type	**3D**

Input Data...

In the Input Data form provide the heat transfer coefficient and fluid node. Leave the *Template ID* field blank.

Convection Coefficient	**0.01**
Fluid Node ID	**<select Node 999>**

OK

Select Application Region...

In the Select Applications Region form select the bottom face of the mounting boss. When selecting the surface the surface chosen will be highlighted. If the incorrect surface is selected simply reselect closer to the centroidal location of the bottom mounting boss surface. The centroid is located between the mounting holes and centered on the width of the surface.

◆**Geometry**

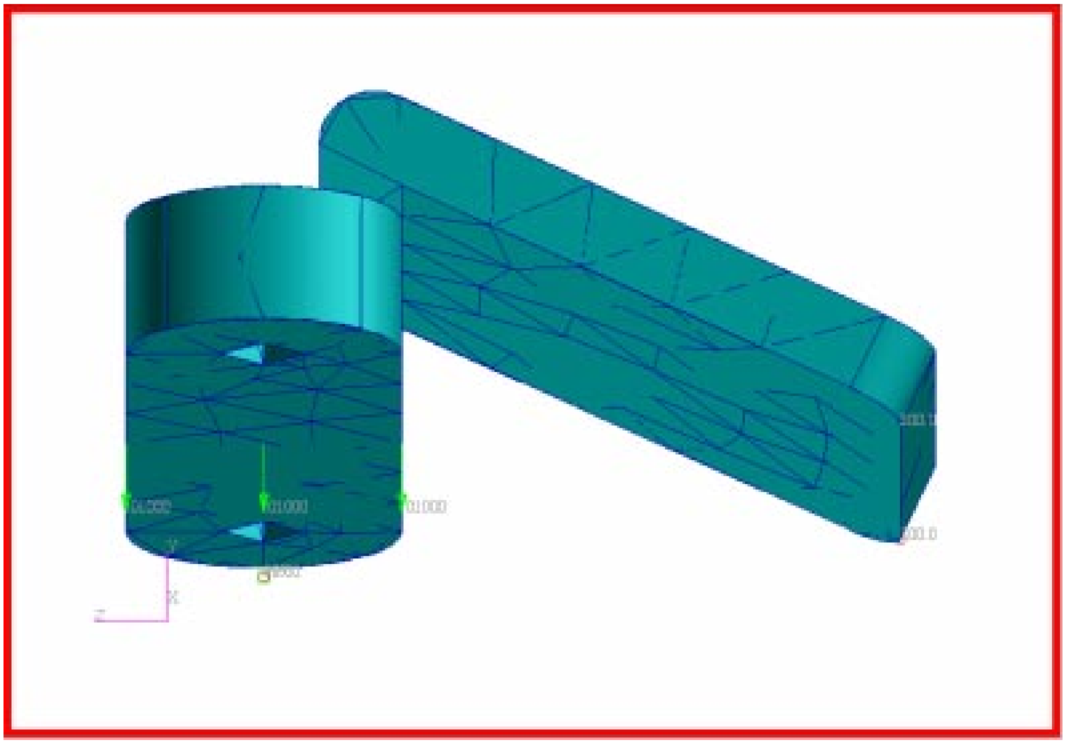
Select Solid Faces	**<select the bottom surface of the mounting boss, Solid 1.1>**

Add

OK

Apply

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



Prepare and run analysis

10. Prepare and submit the model for analysis.

Reset the model to an **isometric_view**. Select **Viewing** from the *Menu Bar* to change to a **isometric_view** of the model. Alternately, this step can be completed using the Tool Bar *Iso 1 View* icon.

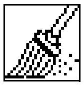
Viewing

Named View Options...

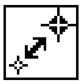
Select Named View **isometric_view**

Close

Reset the graphics using the *Reset Graphics* icon.



Reduce node size using the *Node Size* icon.



Select the **Analysis** *Applications radio button* to prepare the analysis. There are five parameter forms. Change the **Translation Parameters...** as shown below. The analysis will be submitted by selecting **Apply** in the Analysis form.

◆ **Analysis**

Analyze/Full Model/Full Run

Translation Parameters...

File to Extract Undefined Materials:	**4, mpidcgs.bin (CGS Units)**

OK

Solution Parameters...

Calculation Temperature Scale	◆ **Celsius**

OK

Output Requests...

Units Scale for Output Temperatures	◆ **Celsius**

OK

Apply

11. Read results file and plot 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.

Recall that p3 was initiated from a working directory which contained the exercise_05.db database file. The analysis, initiated from within MSC.Patran, created a new subdirectory with the same name as the *Job Name*; it should be named **exercise_05/**. 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 the results file.

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories	**<path>/exercise_05**

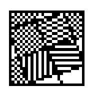
Filter

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.

What is the maximum reported temperature on the mounting boss? Is it at or below the required maximum of 50^{o}C?

Do the spring tab knee and mounting boss transition temperatures meet the requirement of 150^{o}C?

12. Quit MSC.Patran

Quit MSC.Patran

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