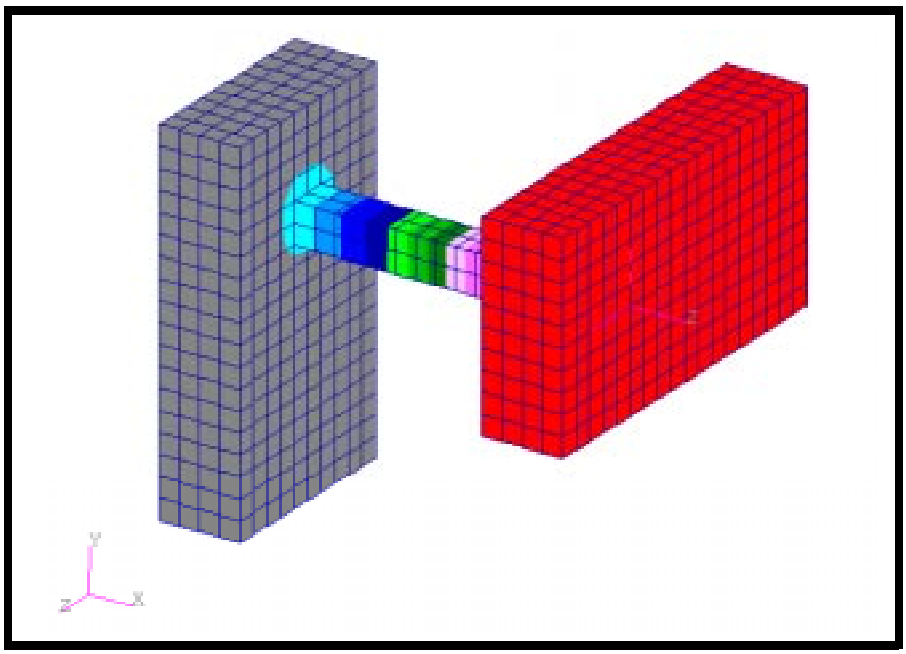
LESSON 15

Thermal - Structural Exercise

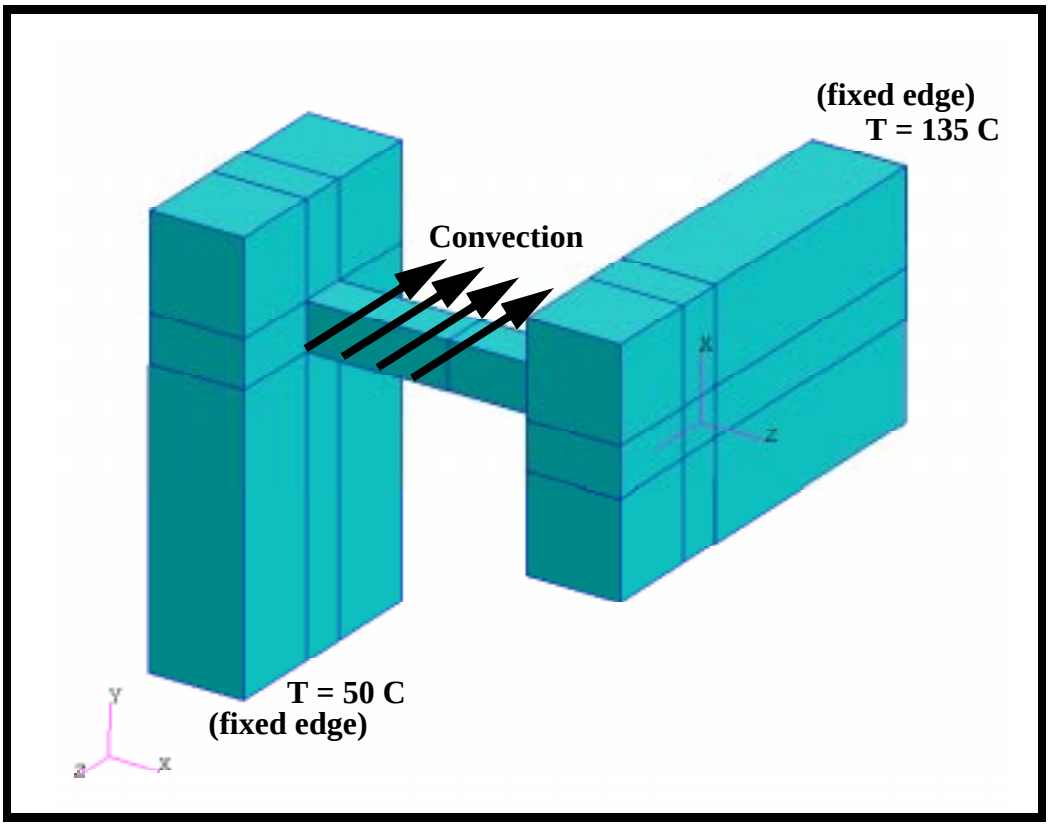


Objectives:

- Demonstrate the use of thermal analysis with temperature loading.
- Demonstrate the use of structural analysis with temperature loading.

Model Description:

In this exercise, the following structure will be subjected to the designated thermal loading and analyzed to determine the steady-state temperature distribution. This temperature distribution will then be applied to the structure, which shall be rigidly fixed at the designated edges and analyzed to determine deformation and stresses due to thermal expansion.



The temperature distribution for the structural analysis shall be read in from the thermal results file.

Exercise Procedure:

1. Open a new database. Name it **thermal_structural.db.**

File/New ...

New Database Name: **thermal_structural.db**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

2. In the *New Model Preference* form pick the following options

Max Model Dimension: **2**

Analysis Code: **MSC/ADVANCED_FEA**

Analysis Type: **Thermal**

OK

3. To create necessary geometry, play the session file named **model.ses.**

File/Session/Play...

Session File List: **model.ses**

Apply

4. Define the temperature dependent material property table for conductivity.

◆ **Fields**

Action: **Create**

Object: **Material Property**

Method: **Tabular Input**

Field Name: **conductivity**

Active Independent Variable: ■ **Temperature (T)**

Input Data...

A table will appear and should be filled in with the data shown below:

Temperature	Conductivity
100	14.6538
600	22.6087
1400	31.8197

OK

Apply

5. Now create the relevant material properties for 17-4 PH stainless steel.

◆ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **stainless 17-4 PH**

Input Properties ...

Conductivity= **conductivity**

Apply

Cancel

6. Apply the steel properties to the model.

◆ **Properties**

Action: **Create**

Dimension: **3D**

Type: **Thermal 3D Solid**

Property Set Name: **prop_thermal**

Input Properties ...

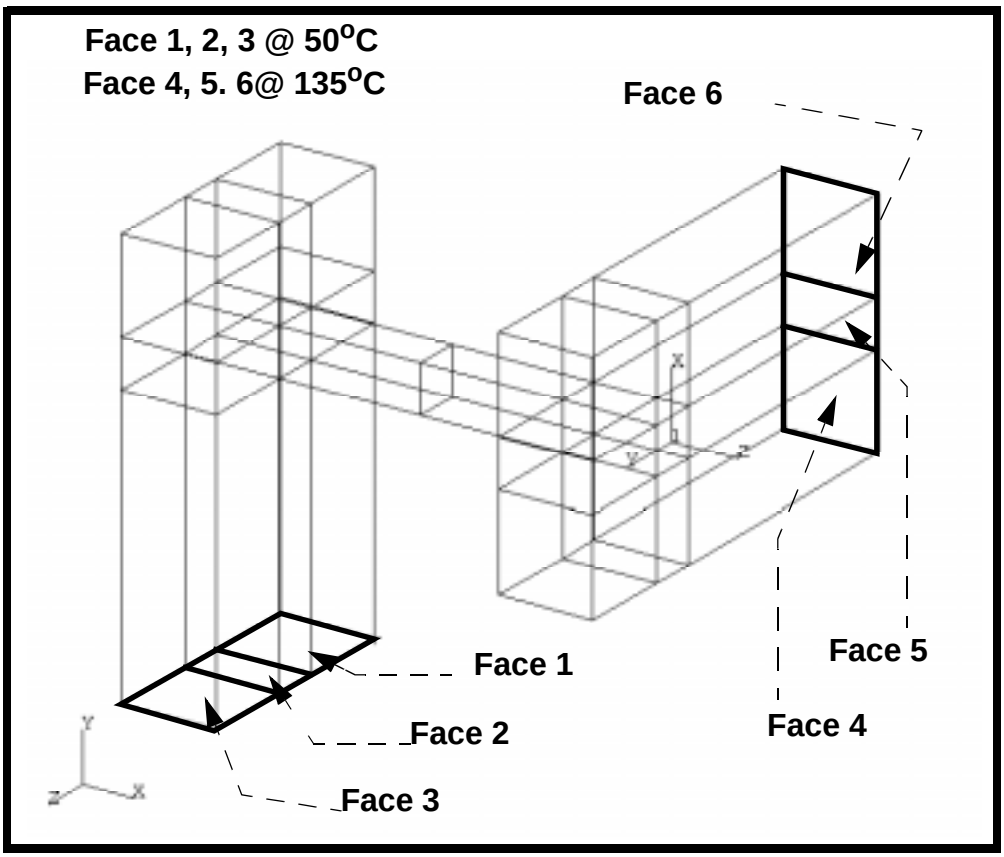
Material Name: **stainless 17-4 PH**

OK	
Select Members:	select all solids
Add	
Apply	

7. Create the temperature loading at the fixed edges.

For guidance on how to apply the temperature boundary conditions, see Figure 15.1:

Figure 15.1 - Solid Faces to Apply Temperatures to



First, create the temperature loading on the left side of the model.

◆Loads/BCs

Action:	**Create**
Object:	**Temperature (pthermal)**
Type:	**Nodal**

Option: **Initial**

New Set Name: **left_edge**

Input Data...

Temperature: **50**

OK

Select Application Region...

In order to select the appropriate solid faces, use the following entity select icon:

Surface or Face

Select Geometric Entities: select Faces 1,2,3

Add

OK

Apply

Next, create the loading for the right side of the model.

Action: **Create**

Object: **Temp(thermal)**

Type: **Nodal**

New Set Name: **right_edge**

Input Data...

Temperature: **135**

OK

Select Application Region...

In order to select the appropriate solid faces, use the following entity select icon:

Surface or Face

Select Geometric Entities:	select Faces 4,5,6

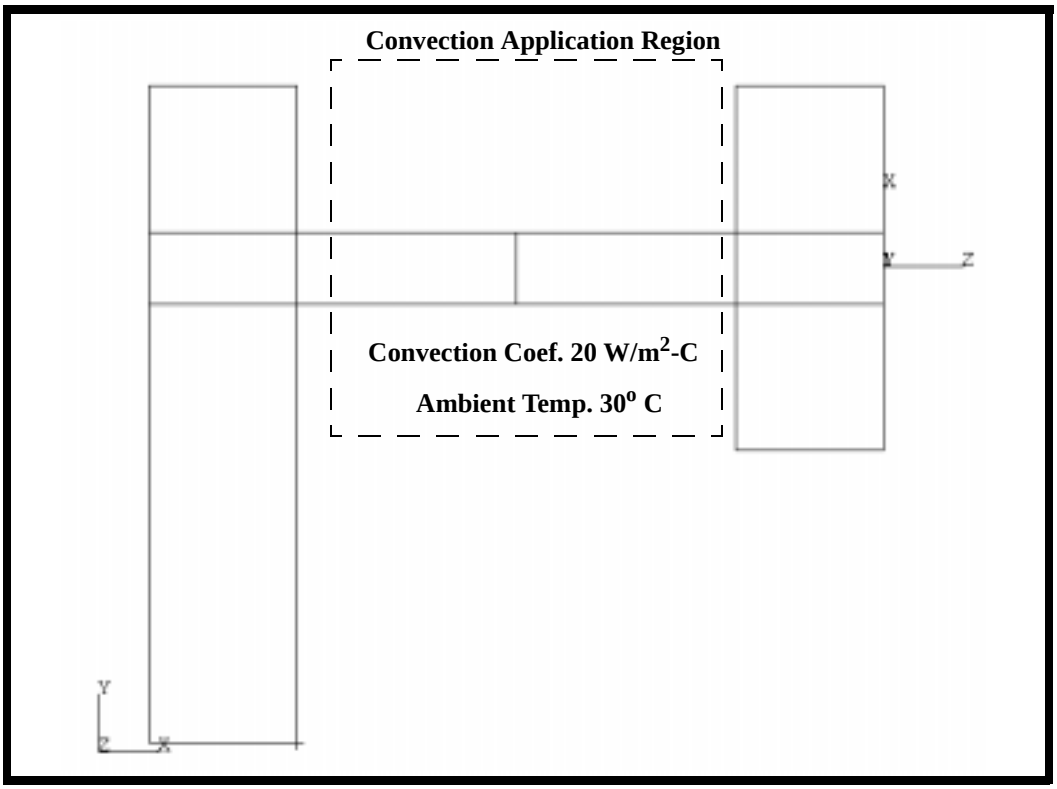
Add

OK

Apply

8. Create the convective loading condition across the middle bar, as shown in Figure 15.2.

Figure 15.2 - Convective Region of Model



◆Loads/BC

Action:	**Create**
Object:	**Convection**
Type:	**Element Uniform**

New Set Name: **convection**

Input Data...

Convection: **20**

Ambient Temperature: **30**

OK

Select Application Region...

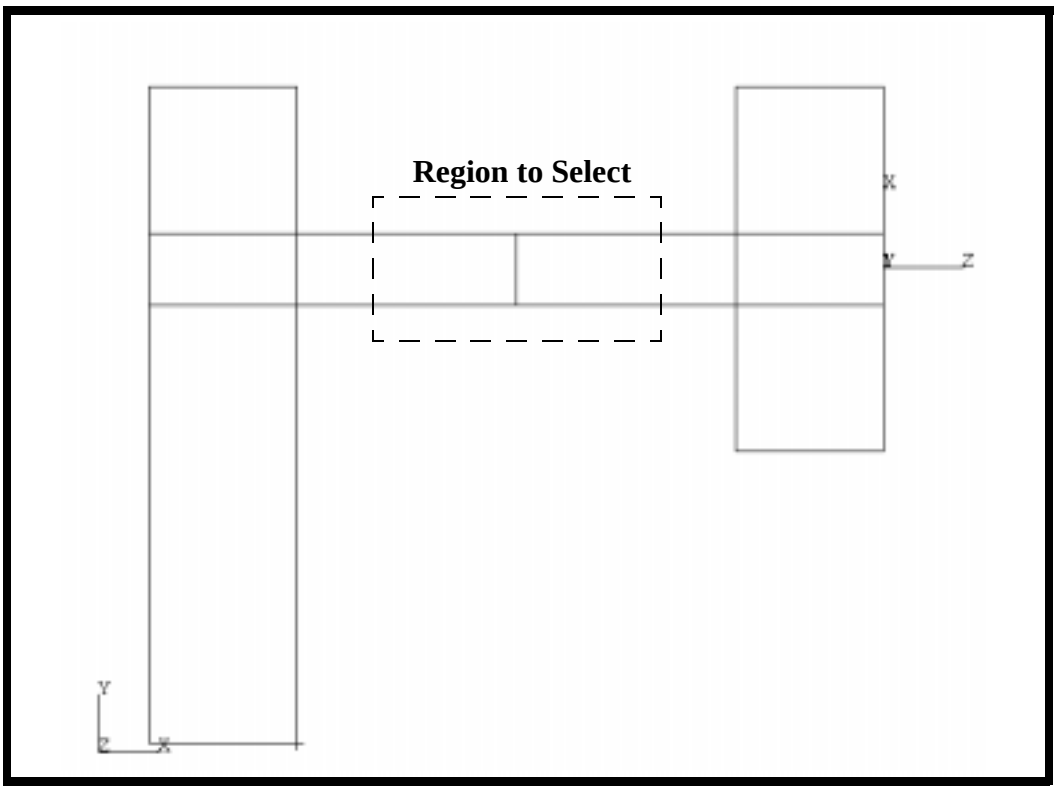
In order to select all 8 faces of the middle bar, temporarily change the picking preferences so that you can select any portion of the entity. Then simply drag the mouse to select a middle portion of the bar.

Preferences/Picking...

◆ **Enclose Any Portion of Entity**

Close

Figure 15.3 - Use 'Pick Any Portion of Entity' to select region



Select Solid Faces: select region shown in Figure 15.3

Add

OK

Apply

Change the picking preference back to the default.

Preferences/Picking...

◆ **Enclose Entire Entity**

Close

9. Submit the model for thermal analysis.

◆**Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name: **thermal_1**

Step Creation...

Job Step Name: **thermal**

Apply

Cancel

Step Selection...

Selected Job Steps: **thermal**

Apply

Apply

You can monitor the progression of the job by looking at **thermal_1.msg** and **thermal_1.sta** files using the UNIX command ***tail -lf [filename]***. You can also monitor the analysis in the background using the UNIX command ***ps -a***.

10. Once the analysis is complete read the results back into the database

◆Analysis

Action:	**Read Results**
Object:	**Results Entities**
Method:	**Translate**

Select Results File ...

Available Files: **thermal_1.fil**

OK

Apply

11. Post process the results of the thermal analysis.

Before viewing the results, remove the boundary conditions from the screen using the following main menu icon:

Reset Graphics

To view the steady-state temperature distribution:

◆Results

Action:	**Create**
Object:	**Quick Plot**

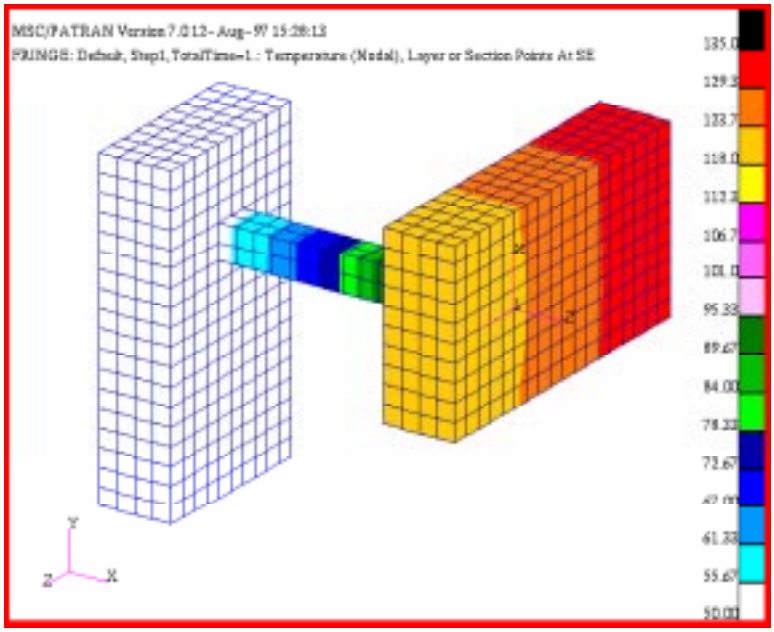
Be sure you are on the **Select Results** form

Select Result Cases:	**Default,Step1,Total Time =1**
Select Fringe Results:	**Temperature(Nodal)**

Apply

You should now see a the steady state temperature distribution in the viewport window, as shown in Figure 15.4:

Figure 15.4 - Steady State Temperature Distribution



12. In order to perform the structural portion of the analysis, change the preference from thermal to structural.

First, remove the results from the screen using the following main menu icon:

Reset Graphics

Preference/Analysis...

Analysis Type: **Structural**

OK

Answer **OK** to" A matching element type with some differences was found for property set "prop_thermal".

OK

This message is to remind us that an applicable property needs to be created for this analysis.

13. Create a new material with the structural properties of 17-4 PH stainless steel.

◆Materials

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**stainless 17-4 PH structural**

Input Properties ...

Elastic Modulus:	**19.65e10**
Poisson Ratio:	**.27**
Reference Temperature:	**30**
Thermal Expansion Coefficient:	**10.8e-6**

Apply

Cancel

14. Now create the structural property for the model.

◆Properties

Action:	**Create**
Dimension:	**3D**
Type:	**Solid**
Property Set Name:	**prop_structural**

Input Properties...

Material:	**stainless 17-4 PH structural**

Select Application Region...

Select Members:	select all solids

Add

Apply

When asked if you wish to overwrite the existing property association, respond with **Yes for All**.

Yes for All

15. Create the boundary constraints which fix the model.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **support**

Input Data...

Translations: **<0, 0, 0>**

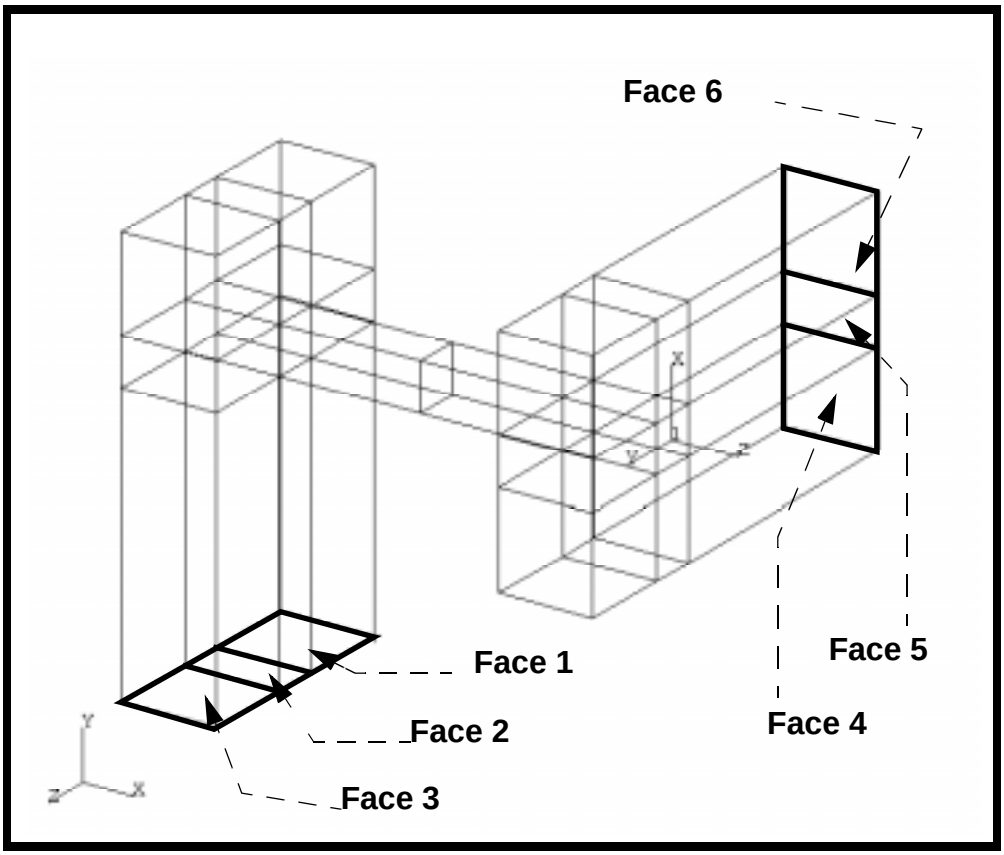
OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select both right & left edges by choosing the solid faces shown in Figure 15.5 (the surface select icon should still be highlighted):

Figure 15.5 - Solid Faces to Apply 'support' LBC to



Select Geometric Entities:	select Faces 1 through 6

Add

OK

Apply

16. In order to individually select the supporting LBC, create a load case including just the support LBC.

◆Load Cases

Action:	**Create**
Load Case Name:	**structure**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet: **Displ_support**

OK

Apply

17. Submit a structural analysis, using the previous thermal analysis results file to obtain the input temperature:

◆Analysis

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name: **structural_1**

Step Creation...

Job Step Name: **structural**

Solution Parameters...

Read Temperature File

Since this is a linear static problem, you must pick the last step and increment of the thermal analysis for both the starting and ending conditions. This will allow the analysis to continue assuming the final steady state temperature distribution.

Start Step Number: **1**

Start Increment Number: **6**

End Step Number: **1**

End Increment Number: **6**

Available Files: **thermal_1.fil**

OK

OK

Select Load Cases...

Available Load Cases: **structure**

OK

Apply

Cancel

Step Selection...

Selected Job Steps: **structural**

Apply

Apply

Once again, you can monitor the progression of the job by looking at **structural_1.msg** and **structural_1.sta** files using the UNIX command ***tail -lf [filename]***. You can also monitor the analysis in the background using the UNIX command ***ps -a***.

18. When the analysis has finished, read in the results of the structural analysis.

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Available Files: **structural_1.fil**

OK

Apply

19. Postprocess the results of the structural analysis.

First, unpost the geometry from the viewport by doing the following:

Display/Plot/Erase ...

Erase All Geometry

OK

Clean up the graphics using the following main menu icon:

Reset Graphics

◆Results

Action:	**Create**
Object:	**Quick Plot**

Click on the **Select Results** form

Select Result Cases:	select the last result case
Select Fringe Result:	**Stress, Components**
Select Deformation Result:	**Deformation, Displacements**

Apply

◆ Results

Form Type:	**Basic**
Select Result Cases:	select the last result case
Select Fringe Result:	**Stress, Components**
Select Deformation Result:	**Deformation, Displacements**

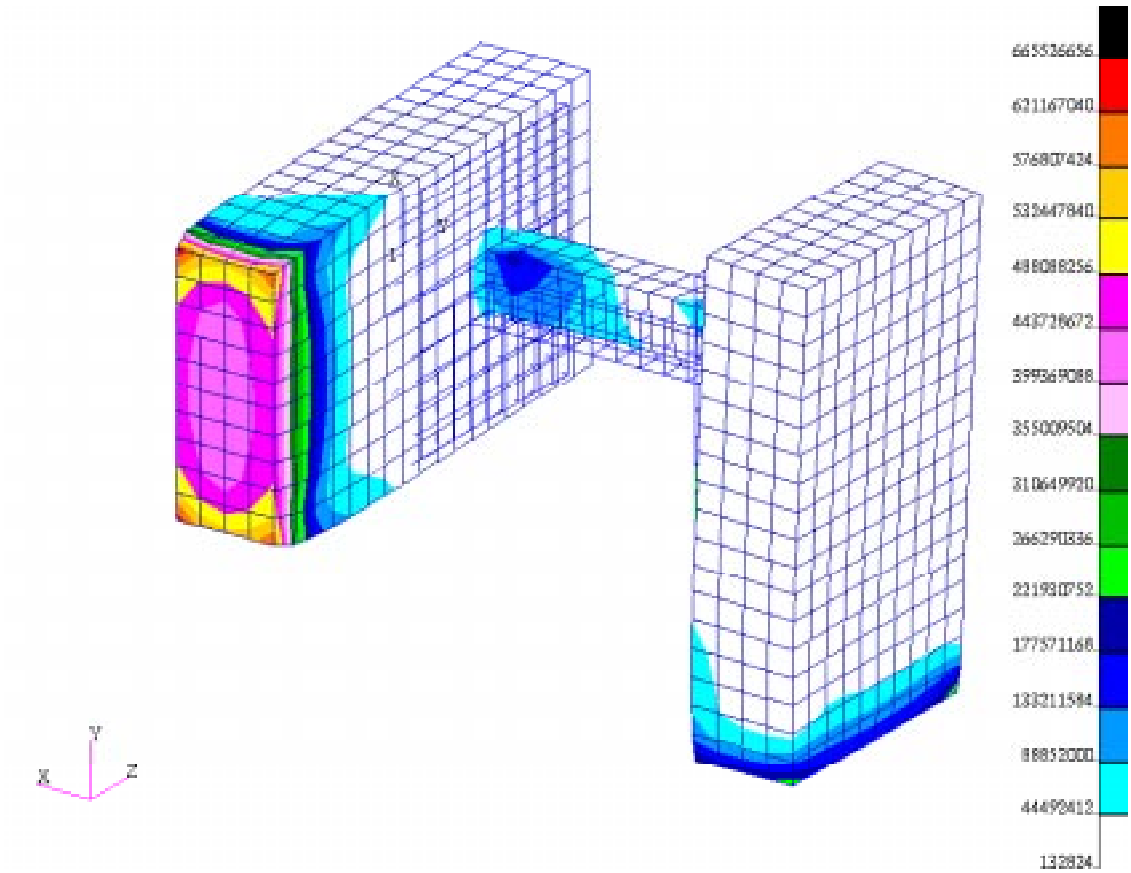
Apply

Notice how the thermal expansion of the model, when restrained by the fixity boundary conditions, induces stresses. To get a better view of the stresses, select the following toolbar icon:

Iso 4 View

The picture shown in Figure 15.6 should be seen in your viewport:

Figure 15.6 - Deformation due to Thermal Expansion



Close the database and quit PATRAN.

This concludes this exercise.