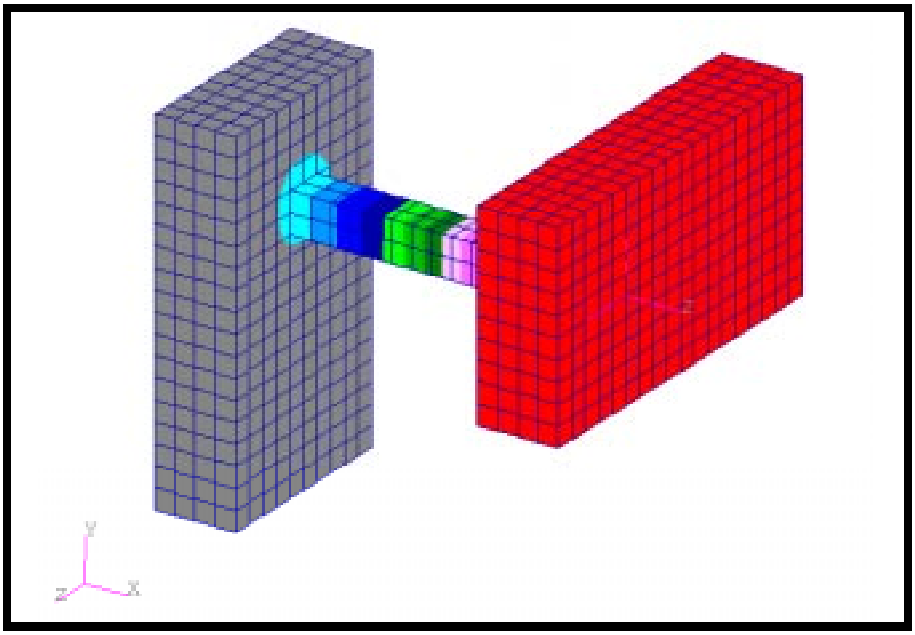
APPENDIX 4

Thermal - Structural Exercise II

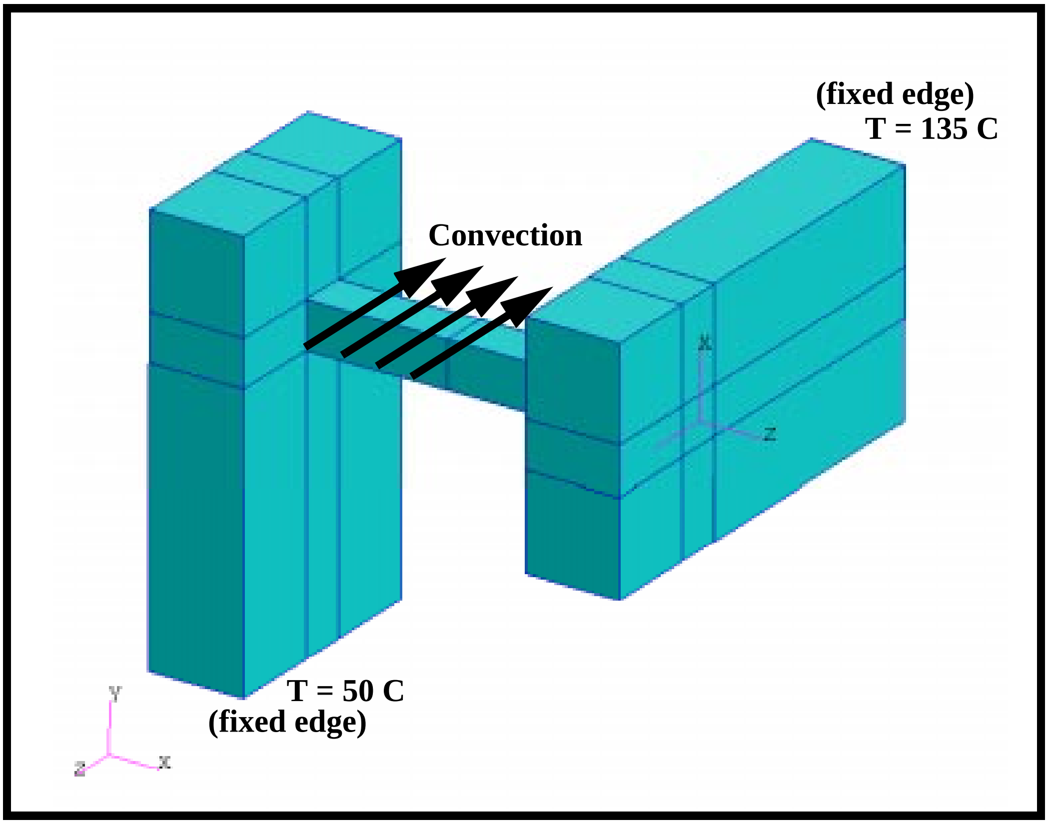


Objectives:

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

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.



In order to obtain the temperature distribution, a field containing the results of the thermal analysis shall be created and applied to the structural model.

Exercise Procedure:

1. Open a new database. Name it **thermal_structural2.db.**

File/New ...

New Database Name: **thermal_structural2.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: **Solid**

Property Set Name: **prop_thermal**

Input Properties ...

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

OK

Select Members: select all solids

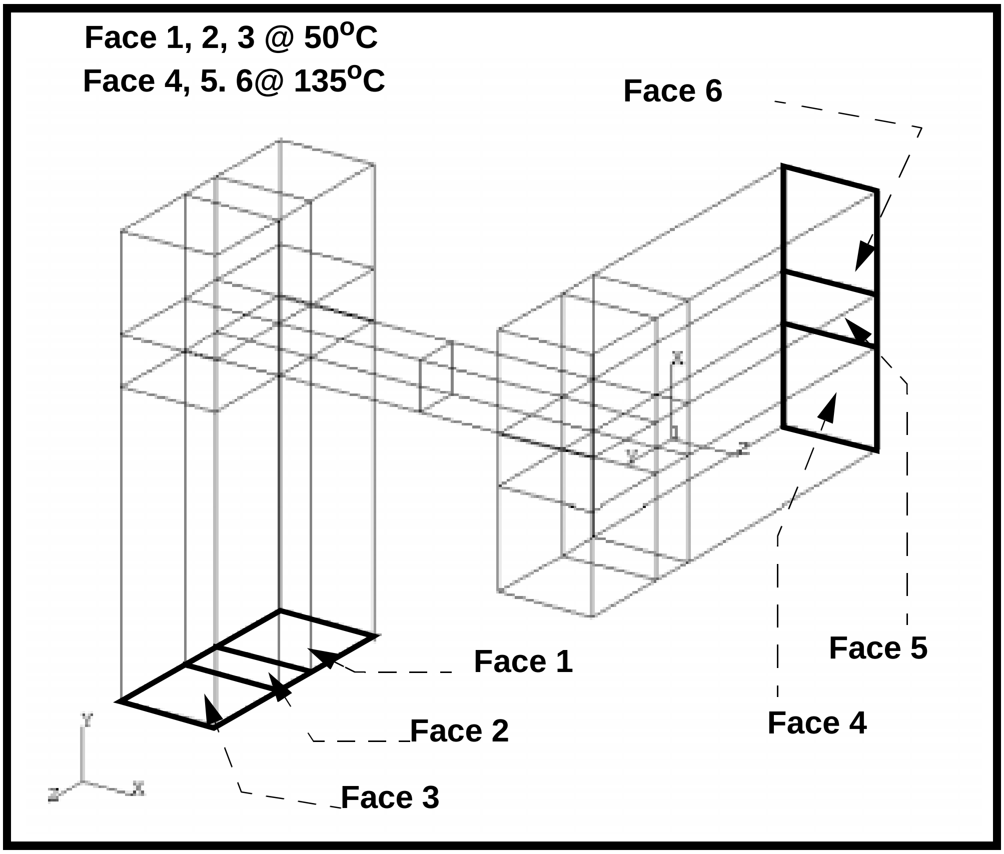
Add

Apply

7. Create the temperature loading at the fixed edges.

For guidance on how to apply the temperature boundary conditions, see the Figure A3.1

Figure A3.1 - Solid Faces to Apply Temperatures to:



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

◆**Loads/BCs**

Action: **Create**

Object: **Temp(thermal)**

Type: **Nodal**

New Set Name: **left_edge**

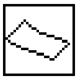
Input Data...

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**

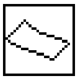
Input Data...

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

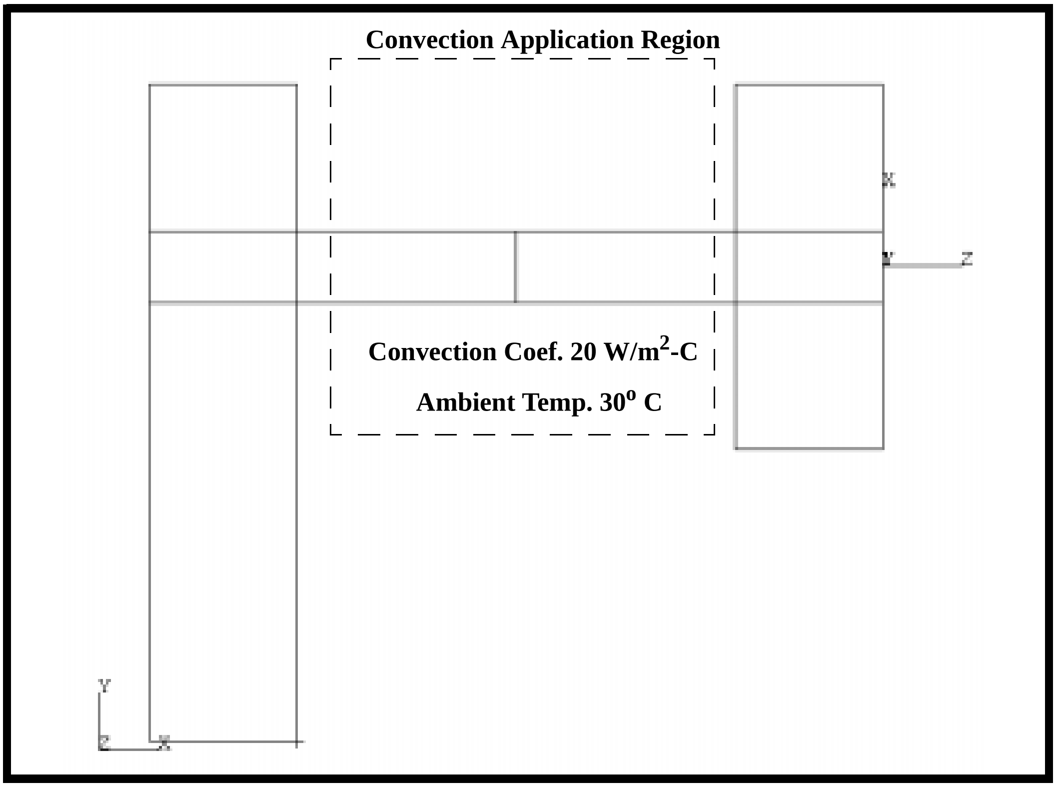
Add

OK

Apply

8. Create the convective loading condition across the middle bar, as shown in Figure A3.2.

Figure A3.2 - 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...

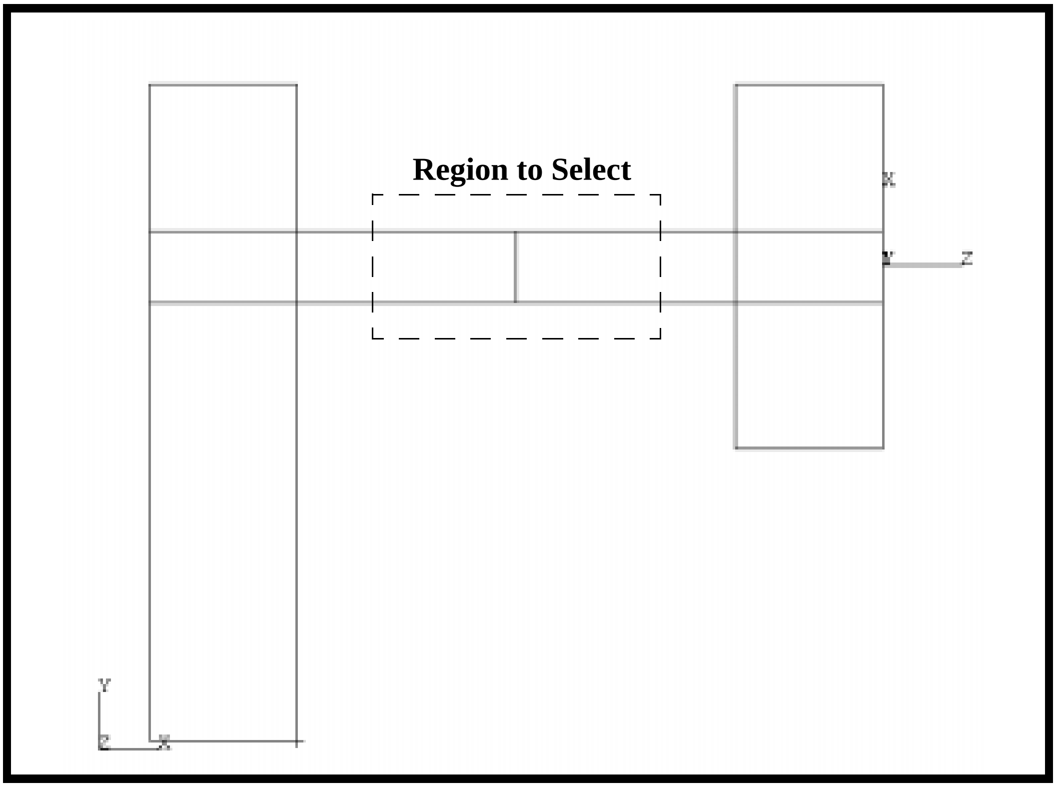
In order to select all 8 faces of the middle bar, temporarily change the picking preferences so that you can select any portion of the entity. Then simply drag the mouse to select a middle portion of the bar.

Preferences/Picking...

◆ **Enclose Any Portion of Entity**

Close

Figure A3.3 - Use 'Pick Any Portion of Entity' to select region



Select Solid Faces:	select region shown in Figure A3.3

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_2**

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_2.msg** and **thermal_2.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 ...

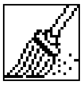
Available Files:	**thermal_2.fil**

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:

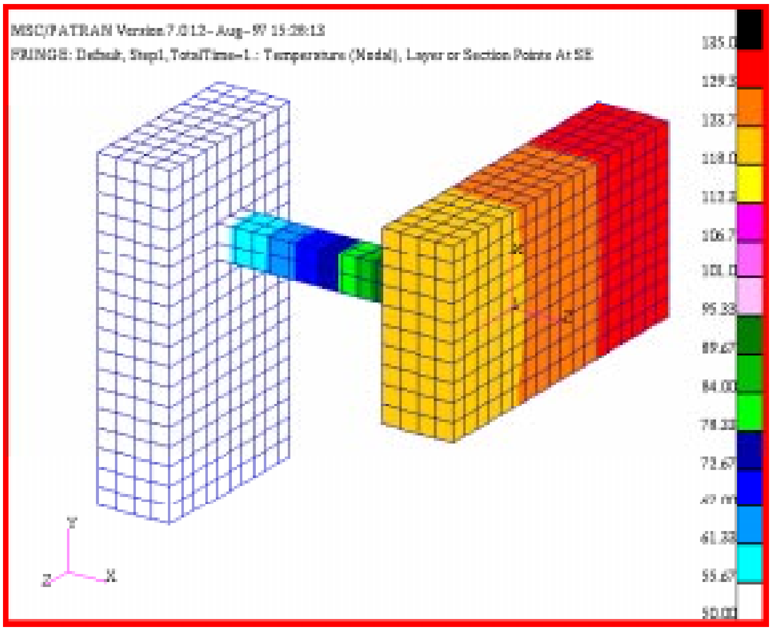
◆**Results**

Form Type:	**Basic**
Select Result Cases:	**Default,Step1,Total Time =1**

Apply

You should now see a the steady state temperature distribution in the viewport window, as shown in Figure A3.4:

Figure A3.4 - Steady State Temperature Distribution



12. Before constructing the structural model, you must first create a FEM field of the resultant temperature distribution. You will later map this temperature field into loading conditions for the structural analysis.

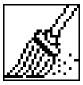
◆**Fields**

Action:	**Create**
Object:	**Spatial**
Method:	**FEM**
Field Name:	**thermal_results**
FEM Field Definition:	◆ **Continuous**
Field Type:	◆ **Scalar**
Mesh/Results Group Filter:	◆**Current Viewport**
Select Group:	**all_entity**

Apply

13. 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.

14. 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

15. 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

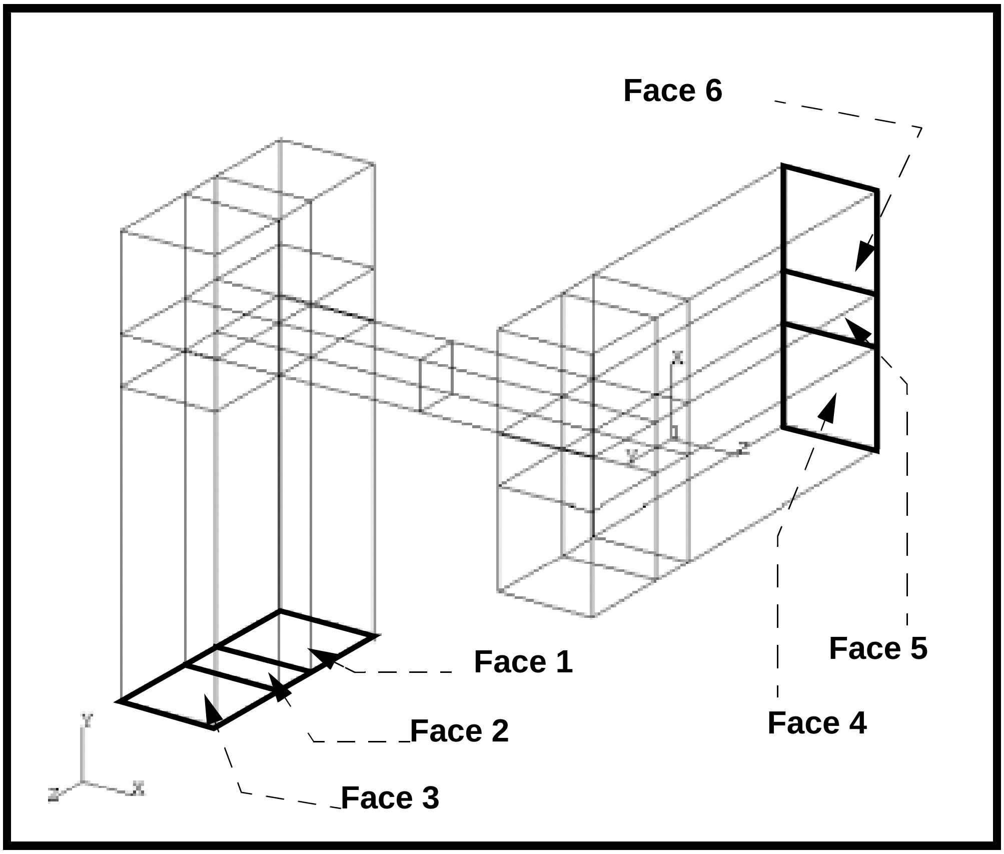
16. Create the boundary constraints which fix the model.

◆ **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 A3.5 (the surface select icon should still be highlighted):

Figure A3.5 - Solid Faces to Apply 'support' LBC to



Select Geometric Entities:	select Faces 1 through 6

Add

OK

Apply

17. Finally, create the appropriate temperature boundary condition.

◆**Loads/BCs**

Action:	**Create**
Object:	**Temperature**

Type: **Nodal**

New Set Name: **applied_temperature**

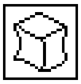
Input Data...

Temperature: **thermal_results**

OK

Select Application Region...

In order to select all the solids of the model, you will have to select the solid entity select icon, shown below:

 **Solid**

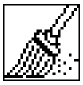
Select Geometric Entities: select all solids

Add

OK

Apply

With all of the loads and boundary conditions applied, the screen is rather cluttered. You might want to clean it up by using the following main menu icon:

 **Reset Graphics**

18. You can verify the applied temperature distribution by plotting the temperature contours.

◆Loads/BCs

Action: **Plot Contours**

Object: **Temperature**

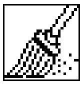
Existing Sets: **applied_temperature**

Select Data Variable: **Temperature**

Select Groups: **all_entity**

Apply

What you should now see is the original steady state temperature distribution resulting from the thermal analysis. After viewing this distribution, clean up the display using the following main menu icon:



Reset Graphics

19. Create a load case in order to isolate only the LBCs you wish to apply to this analysis

◆**Load Cases**

Action:	**Create**
Load Case Name:	structure
Assign/Prioritize Loads/BCs	
Select LBCs to Add to Spreadsheet:	**Displ_support** **Tempe_applied_temperature**
OK	
Apply	

20. Submit a structural analysis of the model.

◆**Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	structural_2
Step Creation...	
Job Step Name:	structural
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_2.msg** and **structural_2.sta** files using the UNIX command **tail -lf [filename]**. You can also monitor the analysis in the background using the UNIX command **ps -a**.

21. 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_2.fil**

OK

Apply

22. Postprocess the results of the structural analysis.

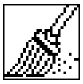
First, unpost the geometry from the viewport by doing the following:

Display/Plot/Erase ...

Erase All Geometry

OK

Clean up the graphics using the following main menu icon:



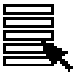
Reset Graphics

◆ **Results**

Action: **Create**

Object: **Quick Plot**

Click on the **Select Results** icon



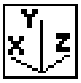
Select Result Cases: select the last result case

Select Fringe Result: **Stress, Components**

Select Deformation Result: **Deformation, Displacements**

Apply

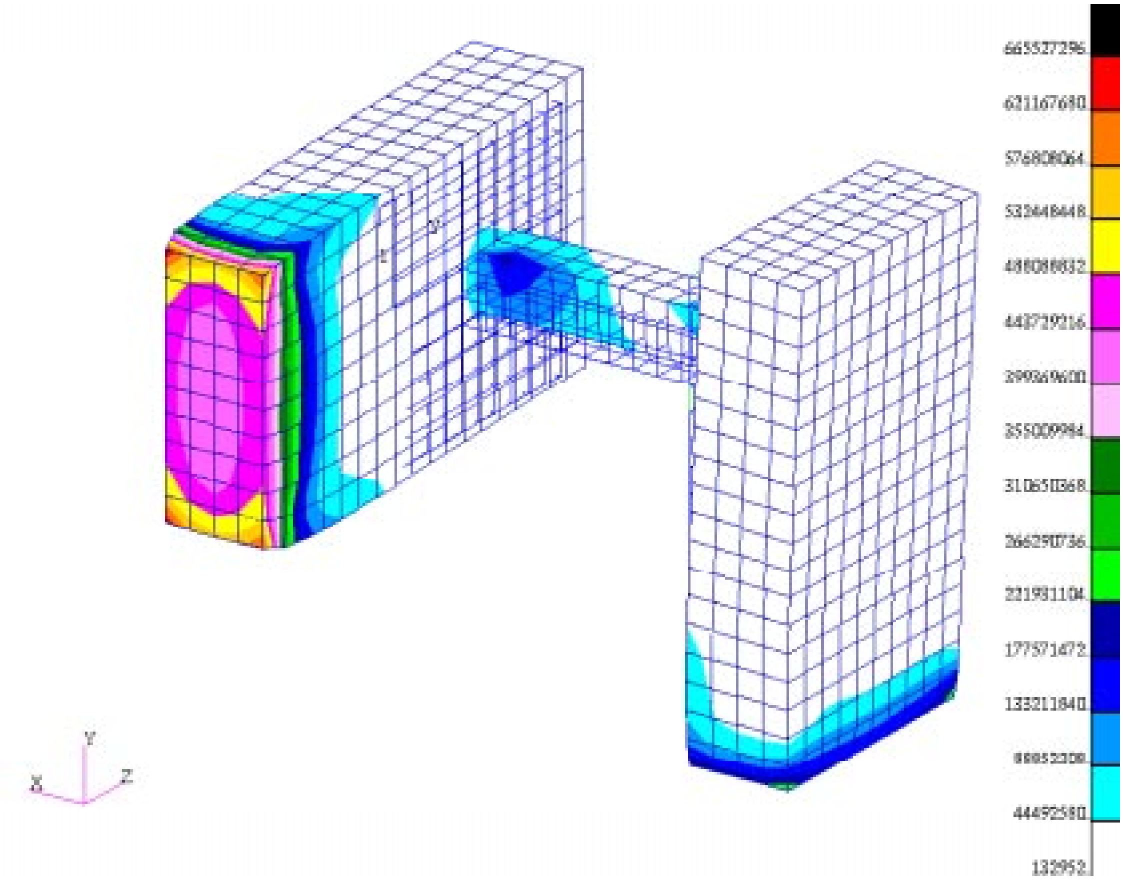
Notice how the thermal expansion of the model, when restrained by the fixity boundary conditions, induces stresses. To get a better view of the stresses, select the following toolbar icon:



Iso 4 View

The picture shown in Figure A3.6 should be seen in your viewport:

Figure A3.6 - Deformation due to Thermal Expansion



Close the database and quit PATRAN.

This concludes this exercise.