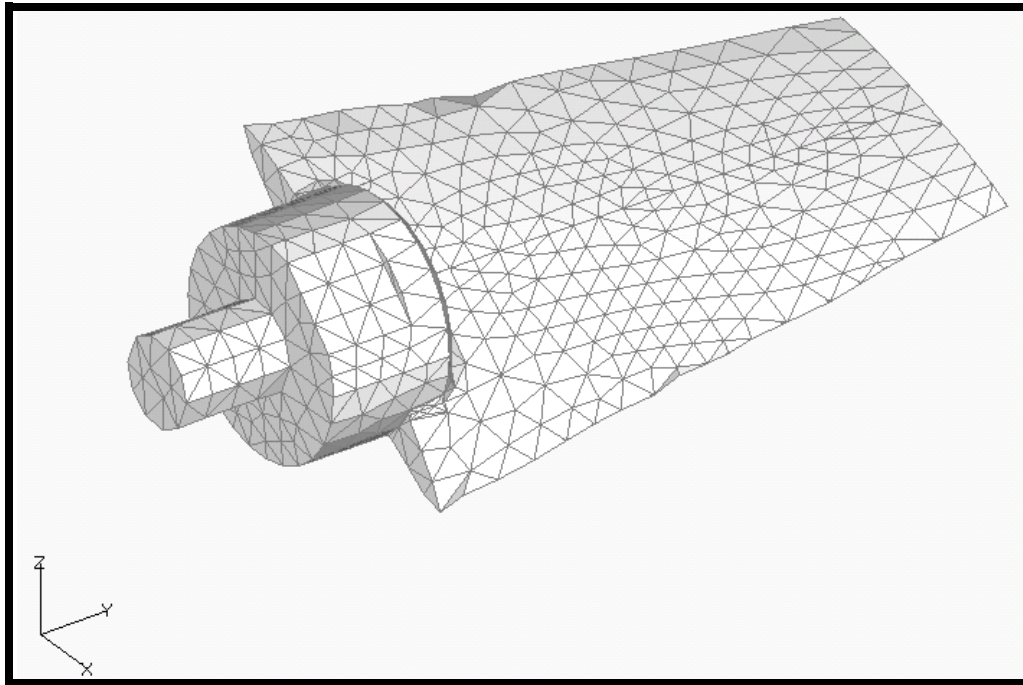

WORKSHOP 30

Thermal Structural Analysis w/ Result Mapping



Objectives:

- Open model from workshop 7.
- Create thermal conditions and obtain a thermal profile.
- Apply the profile to the model for linear static analysis.



Model Description:

This exercise will involve a two step analysis. First step is to apply thermal conditions to the model and obtain a thermal profile. Next, the profile will be applied to the model as a thermal load for static analysis with thermal expansion.

Figure 30.1 - Model Geometry

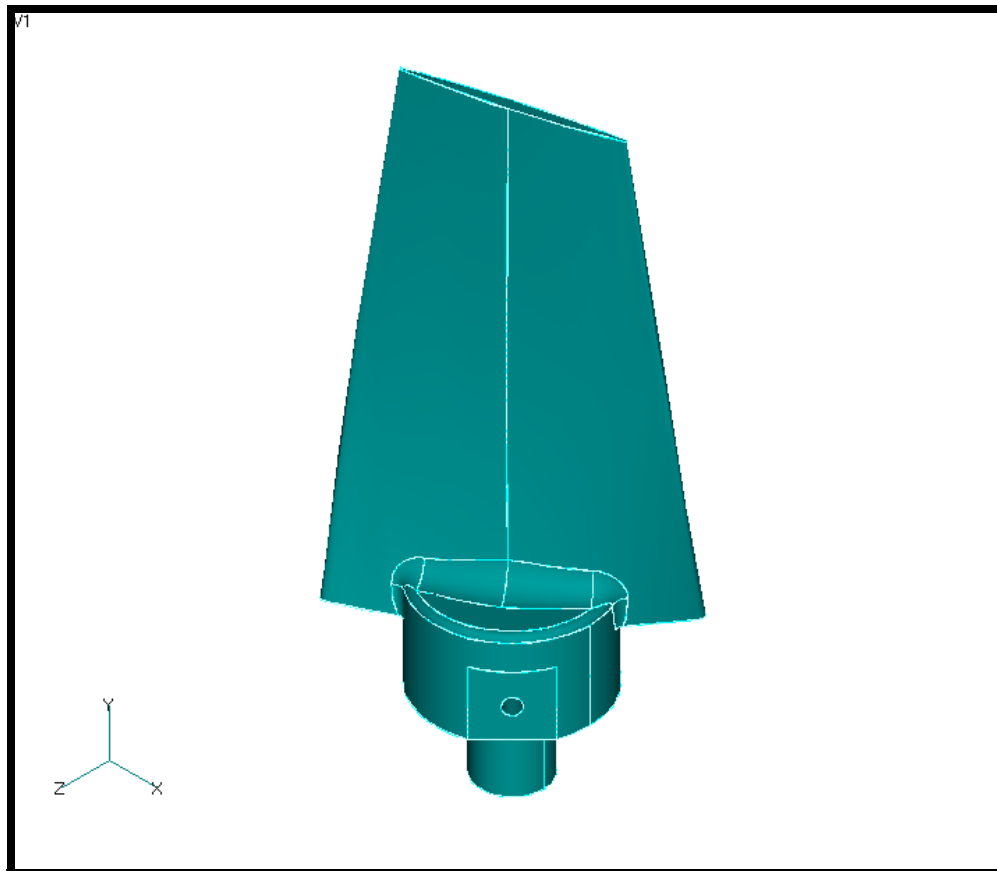


Table 30.1 - Material Property

Young's Modulus	10E6
Poisson's Ratio	0.3
Mass Density:	0.101
Reference Temp:	70
Expansion Coeff, a:	13.1E-6
Conductivity, k:	8.0
Specific Heat, c_p:	0.23

In Figure 30.2 and 30.3, show graphically the thermal loading that will be applied and the thermal expansion constraints.

Figure 30.2 - Loads for Thermal Analysis

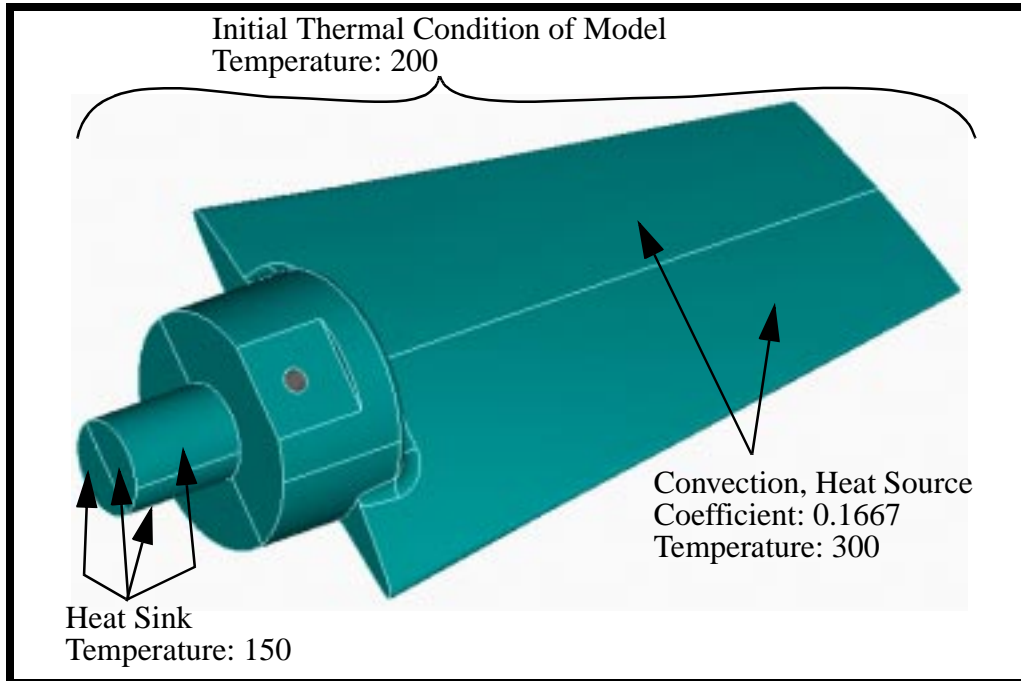
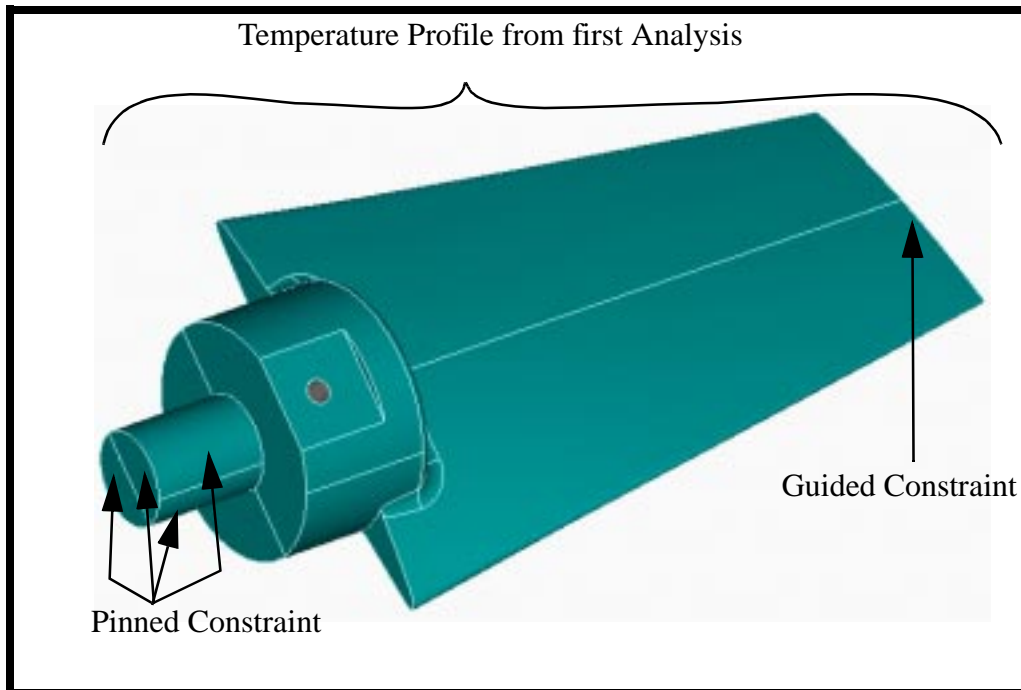


Figure 30.3 - Loads and Constraints for Static Analysis



Suggested Exercise Steps:

- Open the model from Workshop 7.
- Modify the material.
- Delete excess nodes
- Create thermal loads.
- Analyze and display temperature profile.
- Apply the temperature profile as a temperature load.
- Constrain the model.
- Analyze and display thermal expansion stress.

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

Double click on the icon labeled **MSC.Nastran for Windows V4.0**.

On the *Open Model File* form, select **prob4.mod**.

Open Model File:

prob4.mod

Open

2. To apply thermal analysis, the material from Workshop 7 has to be modified.

Modify/Edit/Material...

More

OK

Mass Density:

0.101

Reference Temp:

70

Under the Thermal section:

Expansion Coeff, a:

13.1E-6

Conductivity, k:

8.0

Specific Heat, c_p :

0.23

OK

3. Create the thermal conditions.

Making a load set.

Model/Load/Set

Title:

Thermal

OK

Rotate view to easily apply thermal loads.

View/Rotate...

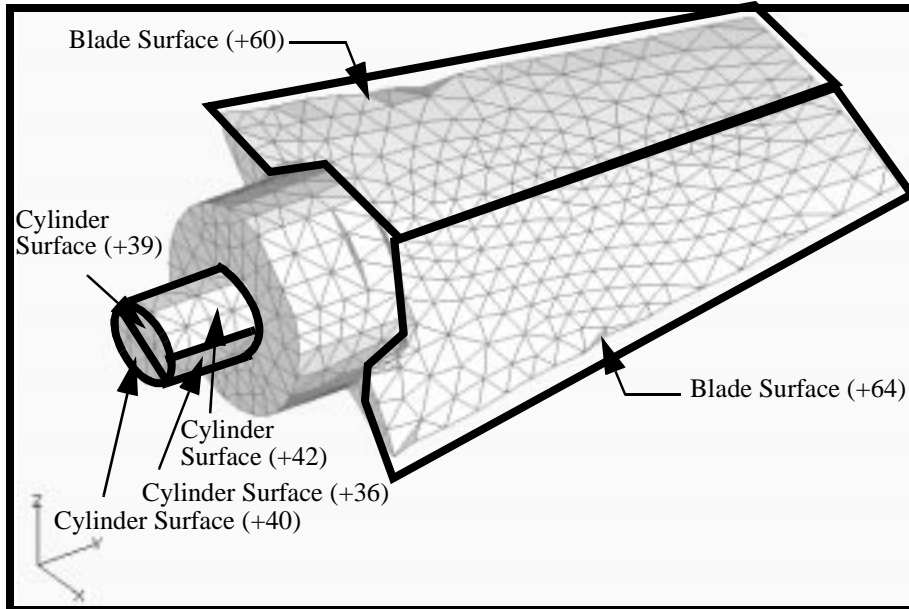
<F8>

Dimetric

OK

The model should be similar to Figure 30.4.

Figure 30.4 - Dimetric View of Elements



Generate the heat source through convection.

Model/Load/Elemental...

Method^

(highlight)

on Surface

Select the two surfaces that form the blade. (refer to Fig. 30.4)

+60
+64

OK

Convection

Under the Load section:

Coefficient:

0.1667

Temperature:

Face:

4. Create the heat sink.

Model/Load/Nodal...

(highlight) **on Surface**

Select all the surfaces of the small cylinder. (refer to Fig 30.4)

Temperature

Under the Load section:

Temperature:

5. Before the thermal analysis, there must be an initial starting temperature.

Model/Load/Body...

Active (for thermal)

Default Temperature T:

6. Save the model.

File/Save As

File name:

prob30

Save

7. Now, the model is ready for analysis.

File/Export/Analysis Model...

Type:

20..Steady-State Heat Transfer

OK

File name:

prob30

Write

Under Additional Info section:

Run Analysis

OK

When prompt, "OK to Save Model Now" choose **Yes**

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran successfully, we will not bother with the details this time.

Continue

8. Display the temperature profile.

Quick Options...

<Ctrl+Q>

All Entities Off

Element

Done

View/Select...

<F5>

NOTE: If Model Style has *Render* turned on, you will not be able to select results. After you display results, you can then turn on *Render* and *Quick Hidden Line* to get a good view of the Contour plot.

Contour Style:

● Contour

Deformed and Contour Data...

Output Vectors:

Contour:

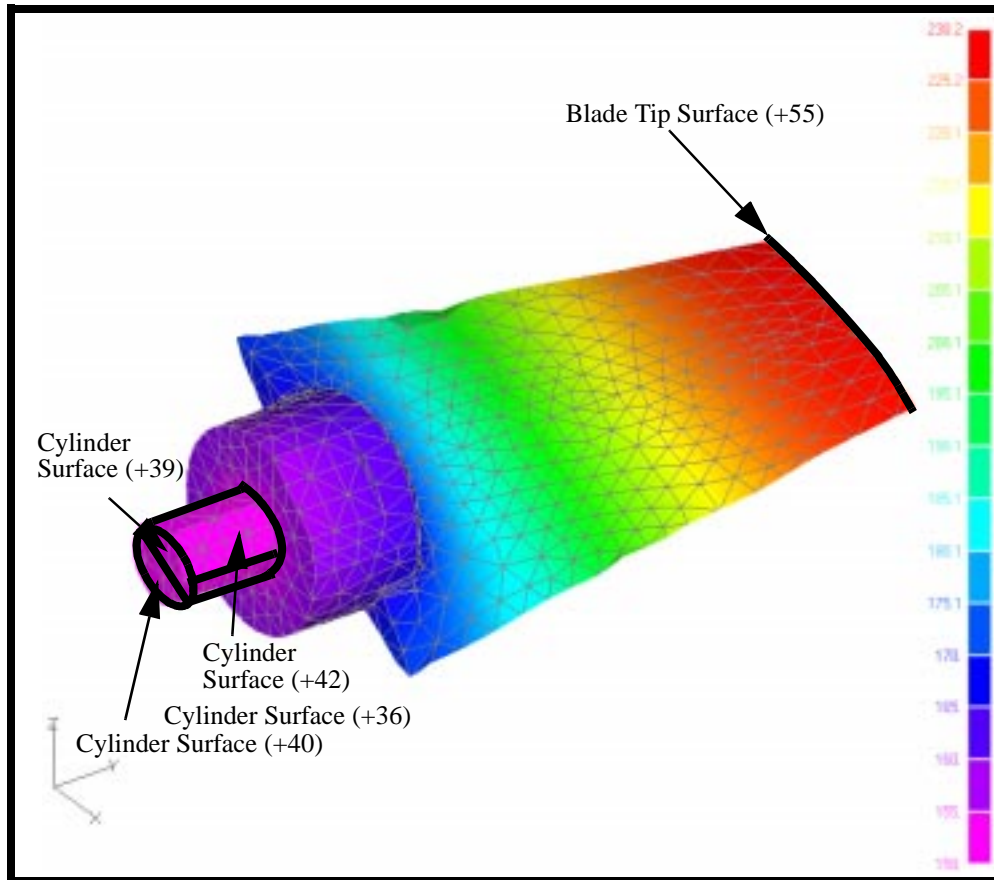
31.. Temperature

OK

OK

The model should be similar to Figure 30.5

Figure 30.5 - Element Temperature Profile



9. Load and constrain the model for static analysis.

First convert the temperature contour into a load profile.

Model/Load/Set...

ID:

Title:

Model/Load/From Output...

Nodal Loads: **Temperatures**

Output Set:

X Vector:

10. Create the constraints on the model.

First create the constrain set.

Model/Constraint/Set...

Title:

Next constraint the model.

Model/Constraint/On Surface...

Select all surfaces on the little cylinder and blade tip.
(refer to Fig. 30.5)

OK

DOF:

Pinned - No Translation

OK

Cancel

11. Now the model is ready for the second analysis.

File/Export/Analysis Model...

Type:

1..Static

OK

File name:

prob30_2

Write

Under Additional Info section:

Run Analysis

Notice that this analysis will use the

Loads 2..profile

Constraints 1..pinned

OK

When prompt, "OK to Save Model Now" choose **Yes**

Yes

When prompt, "OK to read direction cosines from solid element principal stress" choose **Yes**.

Yes

When prompt "OK to read solid element corner stresses" choose **Yes**

Yes

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran successfully, we will not bother with the details this time.

Continue

12. Rotate view to easily see the model.

View/Rotate...

<F8>

Dimetric

OK

13. View static solution results.

View/Select...

<F5>

Deformed Style:

Deform

Deformed and Contour Data...

Output Vectors:

Output Set:

2.. MSC/Nastran Case 1

When prompt, "Output Vector 31 Does Not Exist" choose **OK**.

OK

Deformation:

1.. Total Translation

Contour:

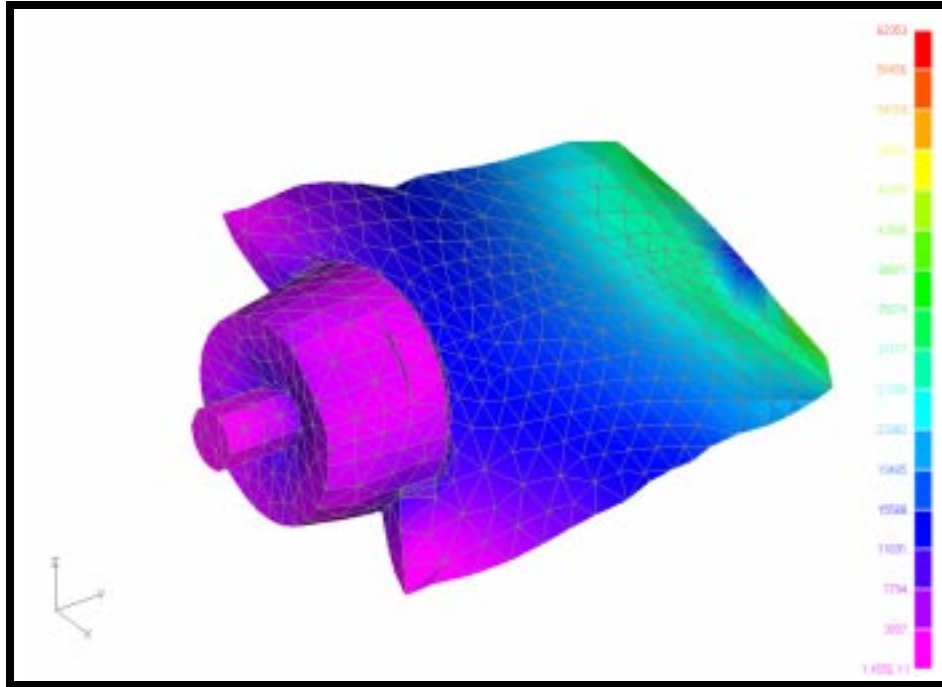
60031.. Solid vonMises Stress

OK

OK

The model should be similar to Figure 30.6.

Figure 30.6 - Thermal Expansion



To only see the deformation results.

View/Select...

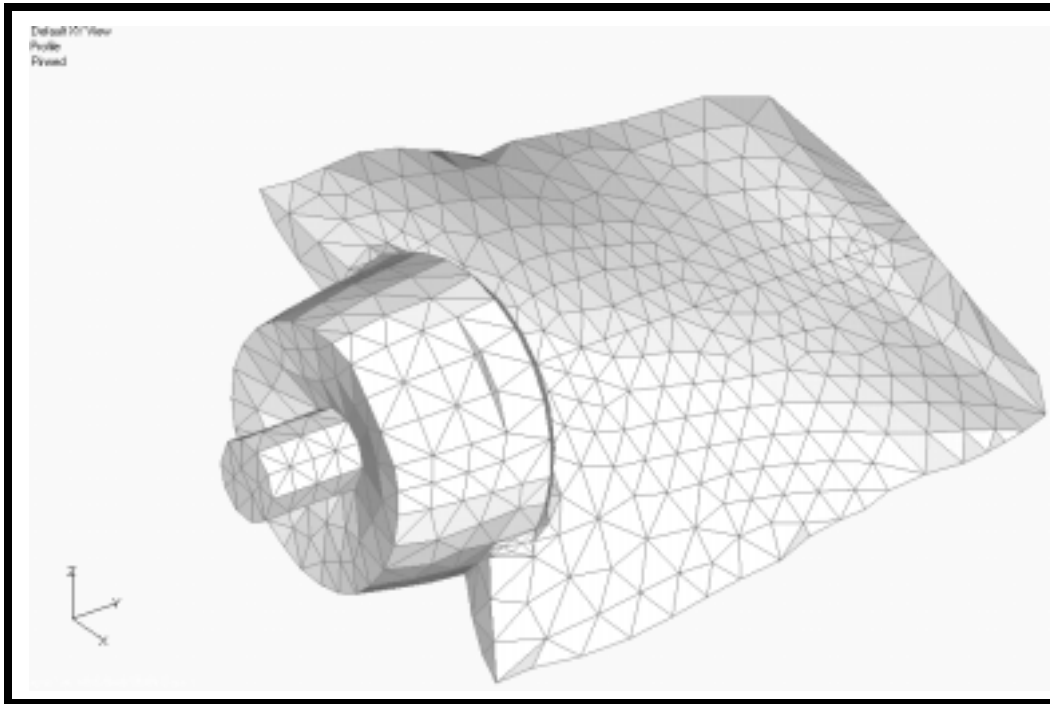
<F5>

Contour Style:

● None - Model Only

OK

The model should be similar to Figure 30.7.

Figure 30.7 - Linear Static Thermal Expansion, Deformation only

To only see the contour results.

View/Select...

<F5>

Deformed Style:

None - Model Only

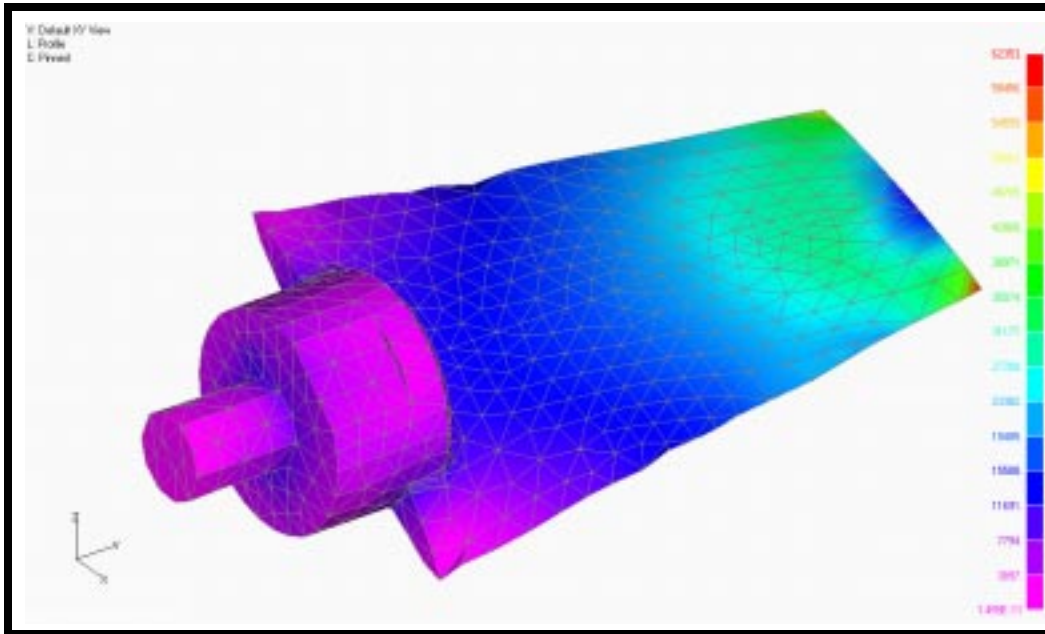
Contour Style:

Contour

OK

The model should be similar to Figure 30.8.

Figure 30.8 - Linear Static Thermal Expansion, Contour only



This concludes this exercise.

File/Save...

File/Exit