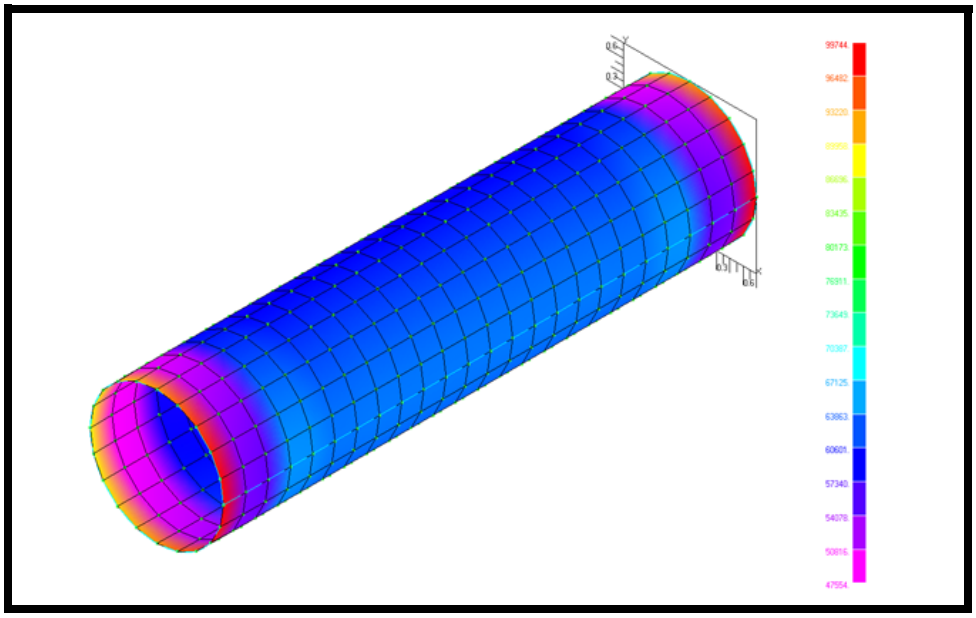
WORKSHOP 7

Thermal Stress Analysis from Directional Heat Loads

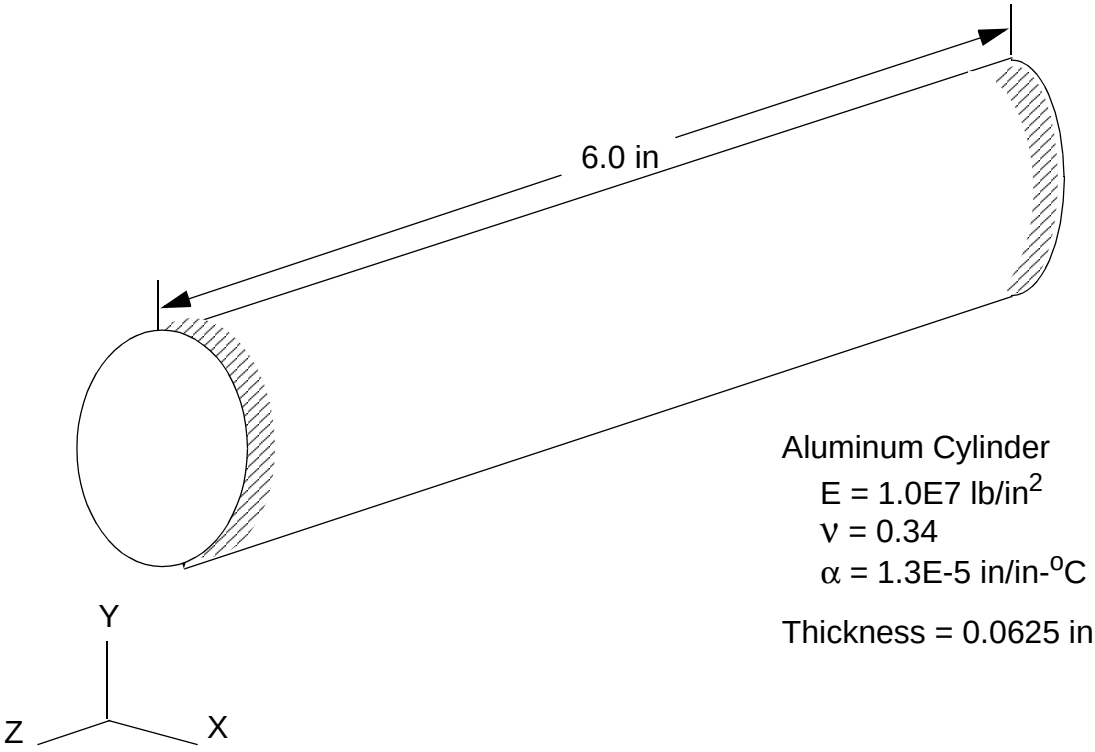


Objectives:

- Open an existing MSC.Nastran for Windows model.
- Create a temperature load based on the previous analysis' output.
- Apply constraints to the ends of the cylinder.
- Run a linear static analysis on the model to obtain stresses due to constraining thermal expansion.

Model Description:

Below is the model created in the previous exercise. The resulting temperature distibution of the steady-state heat transfer analysis is now applied to the model, and the ends of the tube are constrained. This will produce stresses due to constrained thermal expansion. You will run this analysis to determine the deformation and stresses resulting from this constraint.



Exercise Procedure:

1. Open the model created in the previous lesson, **tube.MOD**.

File/Open...

File Name: **tube.MOD**

Open

2. Create a uniform temperature loading for the model.

First, remove the present contour plot from the screen.

View/Select... <F5>

Contour Style: ● **None-Model Only**

OK

Then, a load set must first be created before creating the appropriate model loading.

Model/Load/Set...

ID: **2**

Title: **stress**

OK

Next, apply a uniform default temperature to the model.

Model/Load/From Output...

Nodal Loads: ● **Temperatures**

OK

Output Set: **1.. Case 1 Time 1.**

X Vector: **31..Temperature**

OK

3. Create a constraint set to clamp down the ends of the tube.

First, a constraint set must first be created before creating the appropriate model constraints.

Model/Constraint/Set...

Title: **clamp**

OK

In order to get a better view of the 2 ends, rotate the model to the YZ right position.

View/Rotate... <F8>

YZ Right

OK

Create the fix constraints to the ends.

Model/Constraint/Nodal...

(Select the 2 ends by holding shift and dragging 2 separate boxes around them.)

OK

Fixed

OK

Cancel

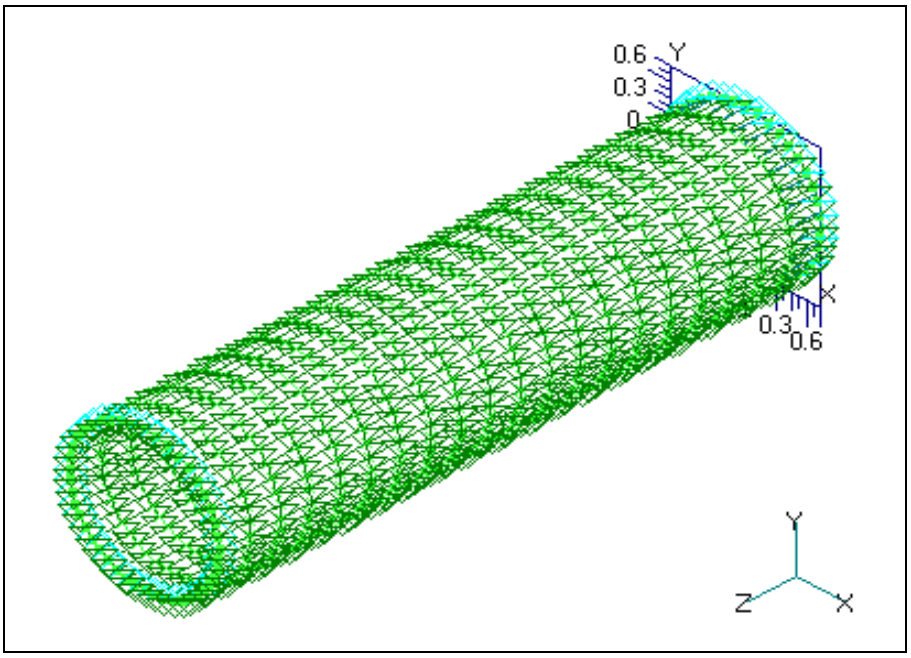
Return the model to its original isometric view. Your model should look like Figure 5-1.

View/Rotate... <F8>

Isometric

OK

Figure 7-1: Model with uniform temperature profile and constraints.



4. Remove the thermal and constraint loading markers from the screen.

View/Options... <Ctrl+Q>

Quick Options...
Load/Constraint Off

Done
Apply
OK

5. Create the input file and run the analysis..

File/Export/Analysis Model...

Analysis Type: **1..Static**

OK

File Name: **static**

Write

☒ *Loads:* **2.. stress**

☒ *Constraints:* **1.. clamp**

☒ **Run Analysis**

OK

When asked if you wish to save the model, respond **Yes**.

Yes

When the MSC.Nastran manager is through running, MSC.Nastran for Windows 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 smoothly, we will not bother with the details this time.

Continue

6. Plot deformation and stress contour on the screen.

View/Select... <F5>

Model Style: ● **Quick Hidden Line**

Deform Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **2..MSC/NASTRAN Case 1**

When the prompt states, Output Vector 31 Does Not Exist, click on **OK.**

OK

Deformation: **1..Total Translation**

Contour: **7033..Plate Top Von Mises Stress**

OK

OK

View the new results and compare them to the results of the previous exercise. The result of static analysis is shown in Figure 7-2.

The stress values in the contour range look valid, but the model's deformation appears out of scale. The default maximum deformation is set to be displayed at 10% of the model's maximum size. For true deformation use **View/Options**.

View/Options... <F6>

Category: ● **PostProcessing**

Options: **Deformed Style**

% of Model (Actual) ☐

OK

When done, exit MSC.Nastran for Windows.

File/Save

File/Exit

This concludes this exercise.

Figure 7-2: Thermal stress on deformed part