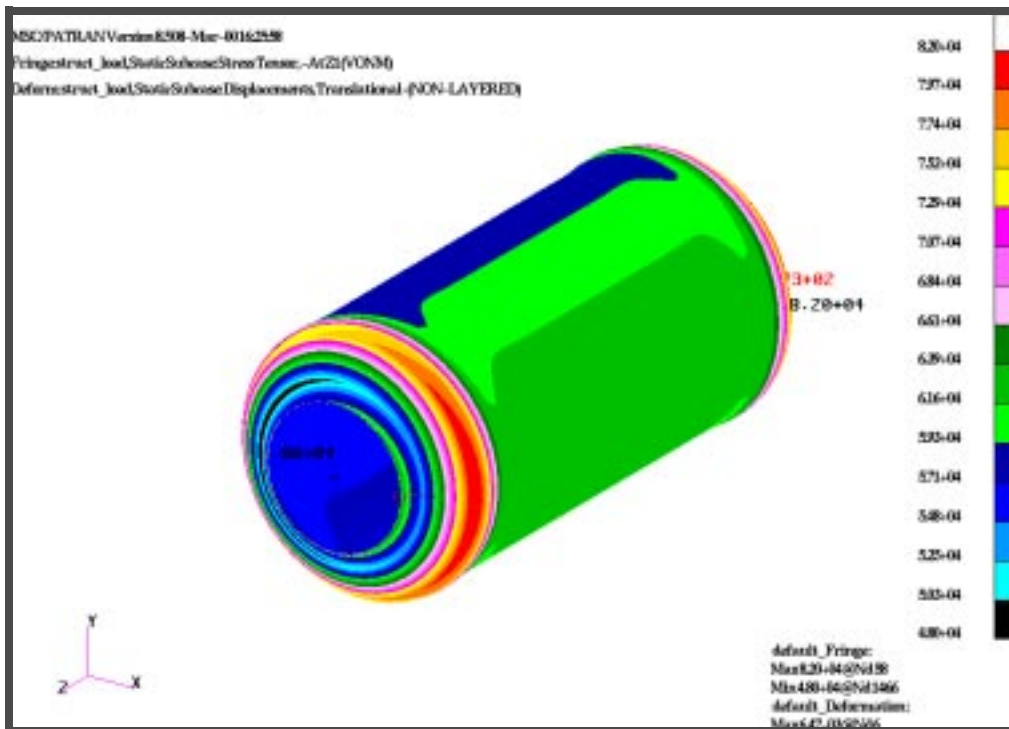


WORKSHOP 9

Thermal Stress Analysis from Directional Heat Loads



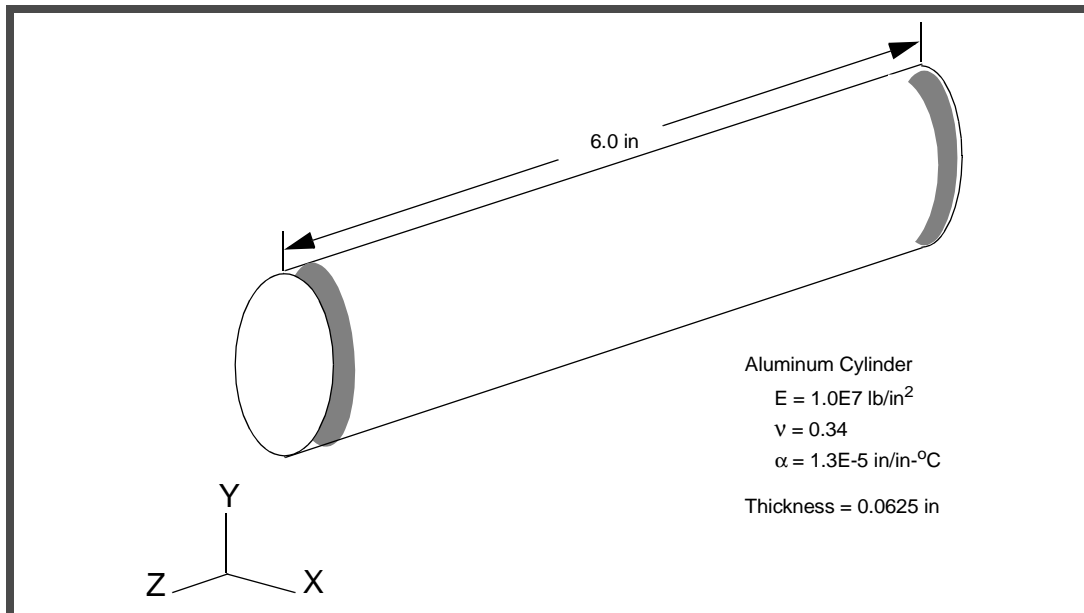


Model Description:

This example demonstrates how to apply the thermal results of Example 8 to perform a stress analysis. We will create the temperature loading for the stress run by using the **Create-Spatial-FEM** command under the Fields Application. You can also use the include punch file option to get the thermal load.

The diameter of the cylinder is 1.5 inch with a length of 6 inches. The material is aluminum. The heat transfer problem solved in Example 8 resulted in a temperature solution which we would now like to apply to a thermal stress analysis.

Figure 9.1





Suggested Exercise Steps:

- Create a new database called **ex9**.
- Create Spatial FEM based on the Temperature Profile.
- Specify the material properties after changing the Analysis Type to Structural.
- Define element properties using 2D shell.
- Create new load case and applied fixed boundary conditions on the end of the cylinder.
- Apply boundary conditions to the structural load case and define temperature load to the model.
- Analyze the model
- Read and display the results.



Exercise Procedure:

1. Open the database **ex8.db** from the previous exercise.

File/Open...

Existing Database Name:

ex8

OK

2. Create a Spatial FEM based on the Temperature Profile.

◆ Fields

Action:

Create

Object:

Spatial

Method:

FEM

Field Name:

tempload

FEM Field Definition:

◆ **Continuous**

Field Type:

◆ **Scalar**

Mesh/Results Group Filter:

◆ **Current Viewport**

Select Group:

default_group

Apply

3. Change the Analysis Type to Structural.

Preferences/Analysis...

Analysis Type:

Structural

OK

4. Specify the Structural Materials.

◆ Materials

Action:

Create

Object:

Isotropic

Method:

Manual Input

<i>Material Name:</i>	<input type="text" value="alum_st"/>
Input Properties...	
<i>Constitutive Model:</i>	<input type="text" value="Linear Elastic"/>
<i>Elastic Modulus:</i>	<input type="text" value="1.0e7"/>
<i>Poisson Ratio:</i>	<input type="text" value="0.34"/>
<i>Thermal Expan. Coeff:</i>	<input type="text" value="1.3e-5"/>
<i>Reference Temperature:</i>	<input type="text" value="0.0"/>
Apply	
Cancel	

5. Assign Element Properties.

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="2D"/>
<i>Type:</i>	<input type="text" value="Shell"/>
<i>Property Set Name:</i>	<input type="text" value="alum_st"/>

Input Properties...

<i>Material Name:</i>	<input type="text" value="m:alum_st"/>
<i>Thickness:</i>	<input type="text" value="0.0625"/>

OK

<i>Select Members:</i>	<input type="text" value="Surface 1"/>
------------------------	--

Add

Apply

When asked, “Surface 1 already has been associated to an element property region. Overwrite the association?”, answer Yes.

Yes

6. Create a New Load Case.

We will create a new load case consisting of the structural thermal loading and apply the fixed boundary conditions on the ends of the cylinder.

◆ **Load Cases**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Load Case Name:</i>	<input type="text" value="struct_load"/>
<i>Load Case Type:</i>	<input type="text" value="Static"/>
<input type="button" value="Apply"/>	

7. Apply the Clamped Boundary Conditions.

◆ **Load/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Displacement"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>Analysis Type:</i>	<input type="text" value="Structural"/>
<i>Current Load Case:</i>	<input type="text" value="struct_load"/>
<i>New Set Name:</i>	<input type="text" value="clamp_bc"/>

<i>Load/BC Set Scale Factor:</i>	<input type="text" value="1.0"/>
<i>Translations <T1 T2 T3></i>	<input type="text" value="< 0., 0., 0.>"/>
<i>Rotations <R1 R2 R3></i>	<input type="text" value="< 0., 0., 0.>"/>

Geometry Filter:

◆ **Geometry**

Click on the **Curve or Edge** icon.



<i>Select Geometry Entities:</i>	<input type="text" value="Curve 1 Surface 1.3"/>
----------------------------------	--

Add
OK
Apply

8. Define a Temperature Load.

◆ **Load/BCs**

Action: **Create**
Object: **Temperature**
Type: **Nodal**
Analysis Type: **Structural**
Current Load Case: **struct_load**
New Set Name: **temp_load**

Input Data...

Load/BC Set Scale Factor: **1.0**
Temperature: **f:tempload**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Click on the **Surface or Face** icon.

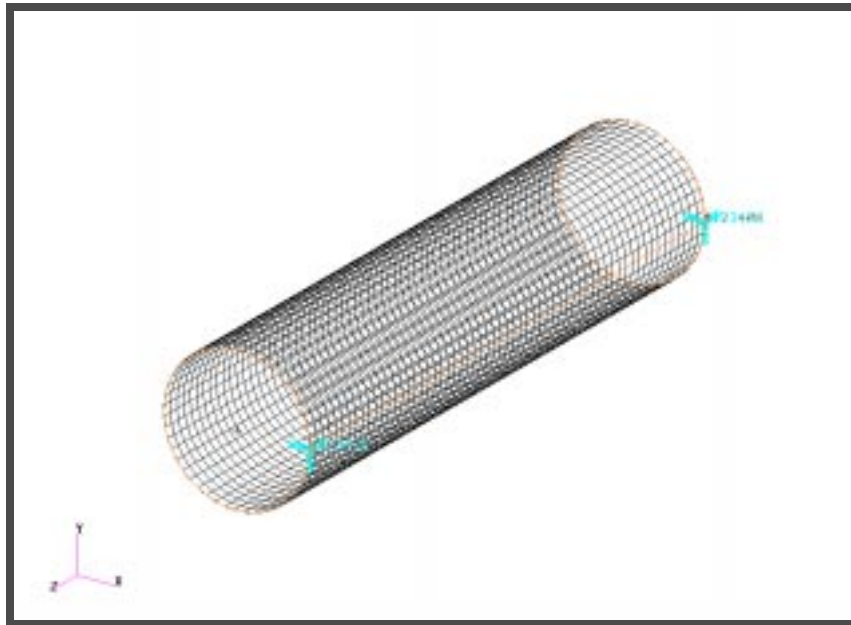


Surface or Face

Select Geometry Entities: **Surface 1**

Add
OK
Apply

Your model should look like the following figure.



9. Perform the Analysis.

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name:

ex9

Subcase Select...

Subcases For Solution Sequence: 101

struct_load

Subcases Selected:

Default

OK

Apply

An MSC.Nastran input file called **ex9.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Submitting the Input File for Analysis:

10. Submit the input file to MSC.Nastran for analysis.
 - 10a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex9.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 10b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex9 scr=yes**. Monitor the run using the UNIX **ps** command.
11. When the run is completed, edit the **ex9.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing **WARNING** messages indicate modeling errors.

12. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.
13. Proceed with the Reverse Translation process, that is, attaching the **ex9.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

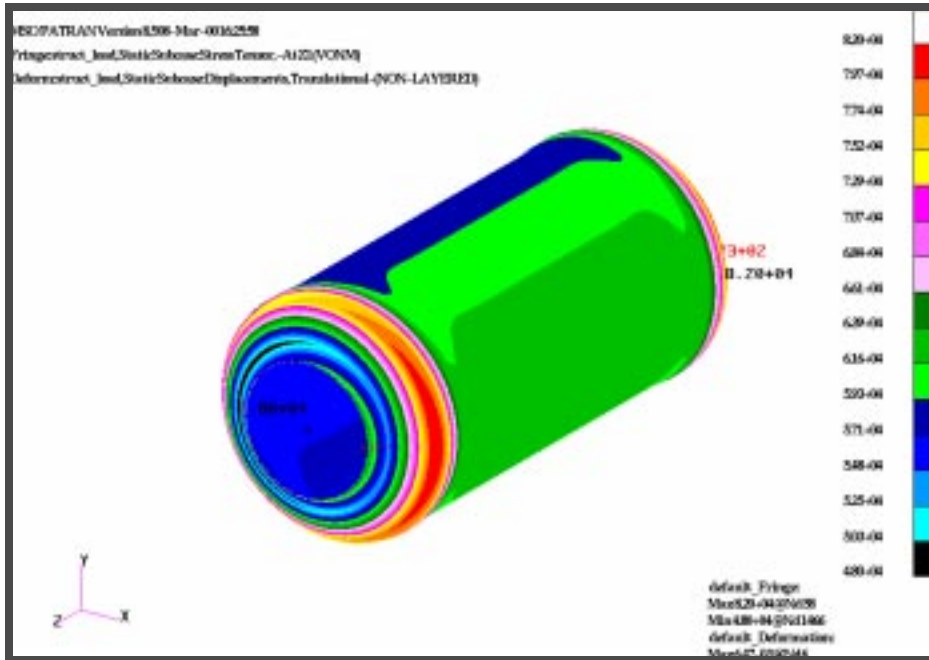
<i>Action:</i>	Attach XDB
<i>Object:</i>	Result Entities
<i>Method:</i>	Local
Select Results File	
<i>Select Results File</i>	ex9.xdb
OK	
Apply	

14. Display the Results.

◆ **Results**

<i>Select Results Cases:</i>	struct_load, Static Subcase
<i>Select Fringe Result:</i>	Stress Tensor
<i>Result Position:</i>	At Z1
<i>Result Quantity:</i>	von Mises
<i>Select Deformation Result:</i>	Displacements, Translational
Apply	

Your model should look like the following figure.



For output we plot the von Mises stress for the fixed end cylinder undergoing the directional thermal load. Peak stresses occur near the fixed end points (recall the points are fixed in X, Y, and Z directions). Thermal expansion causes growth in the axial and radial directions with a circumferential variation due to the directional nature of the thermal load. Near the cylinder mid-plane, in an axial sense, we find the maximum stress at the location which is normal to the directional load vector. The minimum is on the opposite side of the cylinder in the shadow.

Quit MSC.Patran when you have completed this exercise