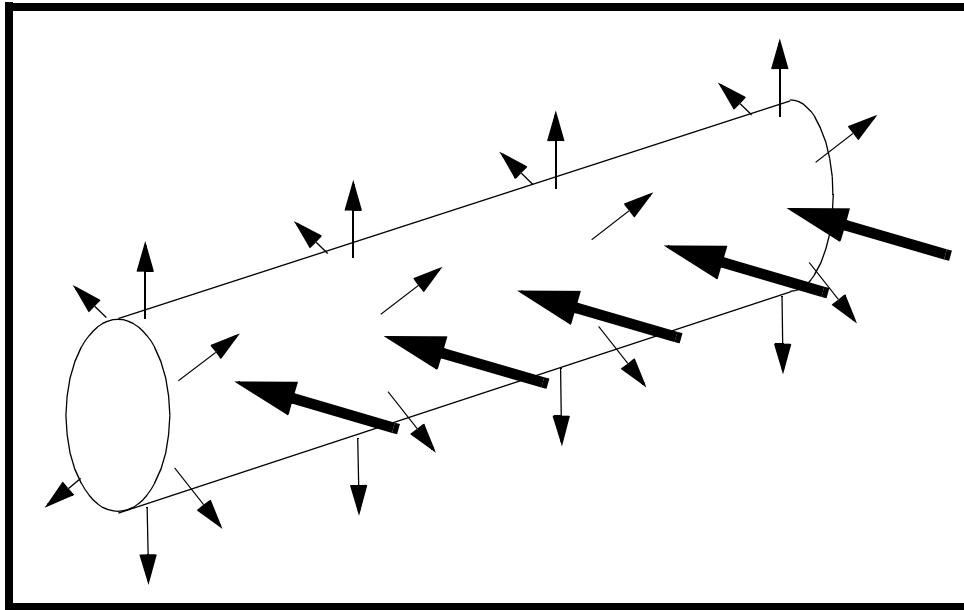

WORKSHOP 5

Directional Heat Loads

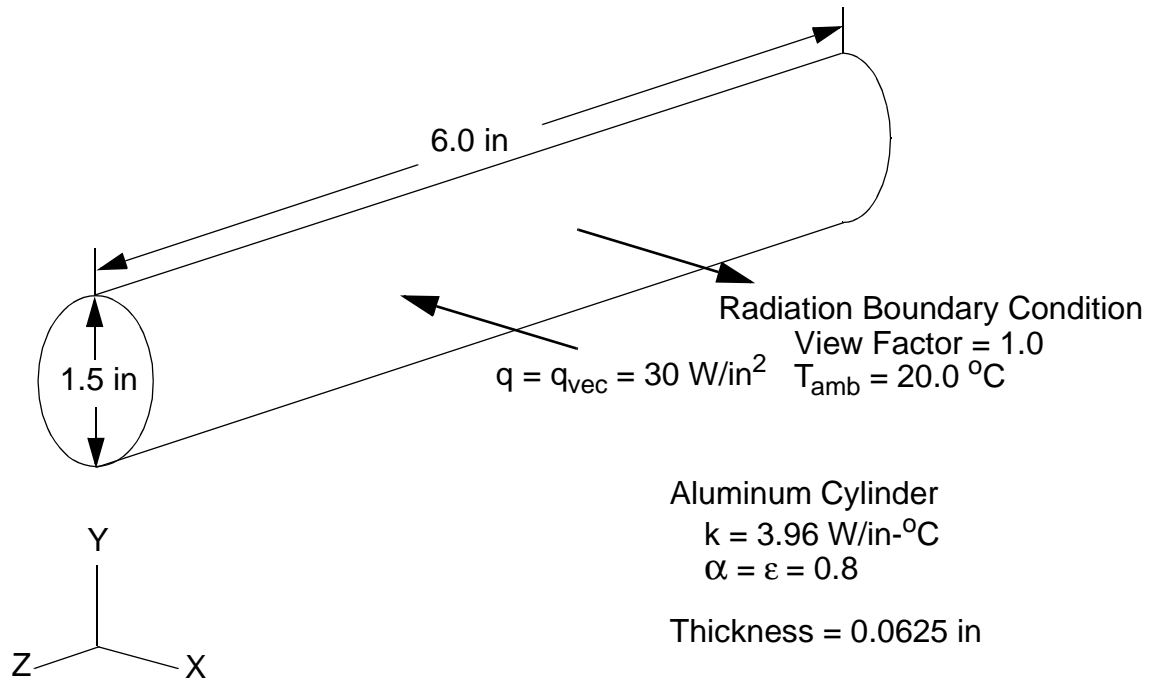


Objectives:

- Create a geometric model of a cylinder.
- Apply ambient radiation to space and a directional heat flux to the model.
- Run a steady-state heat transfer analysis on the model.

Model Description:

Below is a model of a cylinder which radiates heat to space (ambient temperature 20 degrees C, view factor 1.0) in a direction consistent with the surface normal. It has a heat flux passing through it in the global negative x-direction of 30 W/in². In this exercise you will determine the steady state temperature distribution of the model.



Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

On the *Open Model File* form, select **New Model**.

Open Model File:

New Model

2. Create the geometry for the plate.

Tools/Advanced Geometry...

● **Standard**

OK

Geometry/Surface/Cylinder...

First, define the center and the height.

	X:	Y:	Z:
<i>Base:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
<i>Tip:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="6"/>

OK

Next, define the direction toward start of surface.

<i>Base:</i>	<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="0"/>
<i>Tip:</i>	<input type="text" value="1"/>	<input type="text" value="0"/>	<input type="text" value="0"/>

OK

Bottom Outer:

OK

Cancel

View/Rotate... <F8>

3. Create a material called **alum**. In anticipation of a future structural analysis, we will take this opportunity to simultaneously describe the material properties to be used in the structural analysis.

From the pulldown menu, select **Model/Material**.

Model/Material...*Title:**Youngs Modulus, E:**Poisson's Ratio, nu:**Expansion Coeff, a:**Conductivity, k:*

4. Create a property called **tube** to apply to the members of the cylinder itself.

From the pulldown menu, select **Model/Property**.

Model/Property...*Title:*

To select the material, click on the list icon next to the databox and select **alum**.

*Material:**Thickness, Tavg or T1:*

5. Define the mesh size on the tube.

Mesh/Mesh Control/Mapped Divisions on Surface...

Select All

OK

Now input the mesh size on surface.

	<i>s:</i>	<i>t:</i>
<i>Number of Elements:</i>	<input type="text" value="20"/>	<input type="text" value="24"/>
<i>Bias:</i>	<input type="text" value="1."/>	<input type="text" value="1."/>
<input type="button" value="OK"/>		
<input type="button" value="Cancel"/>		

6. Now, create the mesh for the model.

Mesh/Geometry/Surface

Select All

OK

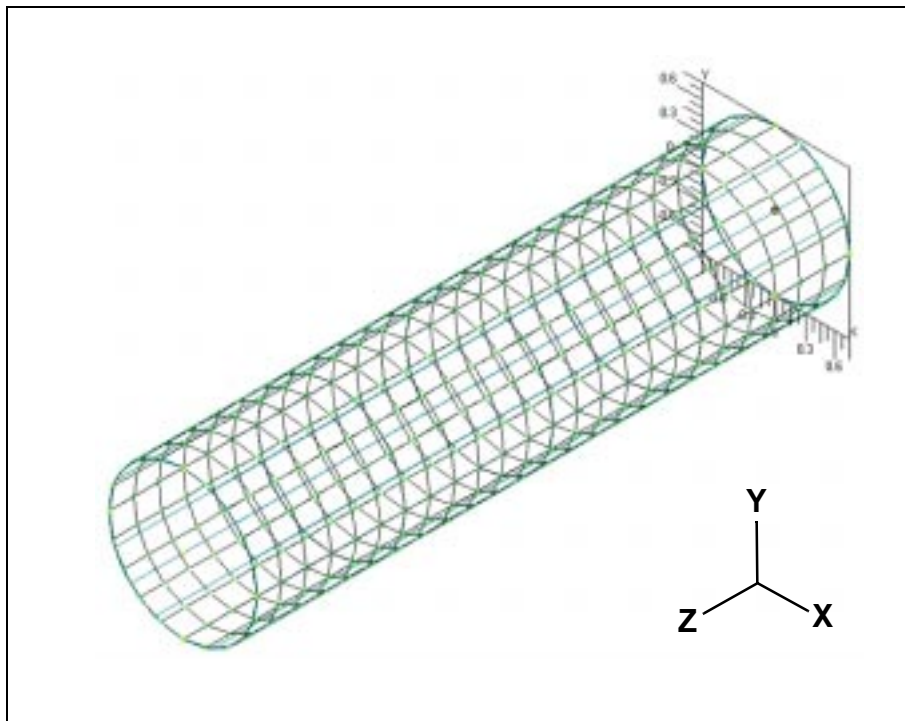
Property:

OK

View/Autoscale... <Ctrl+A>

Compare the topology of your finite element model to Figure 4-1.

Figure 5-1: Meshed Model



7. Now check and merge all coincident nodes.

Tools/Check/Coincident Nodes...

Select All

OK

When asked if it is OK to specify additional range of nodes to merge, respond **No**.

No

Options:

OK

Merge Coincident Entities

8. Create a uniform temperature loading for the model.

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

Model/Load/Set...

Title:

Next, apply a uniform default temperature to the model.

Model/Load/Body...

(next to Thermal options) **Active**
Default Temperature:

Set up heat transfer properties by the following procedure:

Model/Load/Heat Transfer

Temp Offset from Abs Zero:
Stefan-Boltzmann:

9. Verify the direction of the element normal vectors of the shell elements.

View/Options... <F6>

Options:
Normal Style:
Show Direction

Turn the model to a better view to verify the vectors.

View/Rotate... <F8>

OK

If the normal vectors are all pointing at the outward direction, return to the previous view point and turn off the vectors. Otherwise, **reverse** any elements that has a normal pointing in the wrong direction.

View/Rotate...

Isometric

OK

View/Options... <F6>

Show Direction

OK

10. Apply the loading conditions to the surface of the model.

First, create the heat flux.

Model/Load/Elemental...

Select All

OK

Type:

Heat Flux

Directional

Flux:

30

Absorptivity:

0.8

OK

X:

-1

OK

Face:

1

OK

Next, apply radiation to space at ambient temperature of 20 degrees C.

Select All

OK

Type:

Radiation

Emissivity:

0.8

Absorptivity:

0.8

Temperature:

20

View Factor:

1

OK

Face:

1

OK

Cancel

11. Create the input file and run the analysis.

File/Export/Analysis Model...

Type:

20..Steady-State Heat Transfer

OK

File Name:

Directional

Run Analysis

OK

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

Yes

File Name:

tube

Save

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

12. Remove the thermal loading markers from the screen.

View/Options... <Ctrl+Q>

Quick Options...

Load - Heat Flux

Load - Radiation

Done

OK

13. Create a final temperature distribution contour plot.

View/Select... <F5>

Model Style:

Quick Hidden Line

Render

Contour Style:

Contour

Deformed and Contour Data...

Output Set:

1..Case 1 Time 1

Contour:

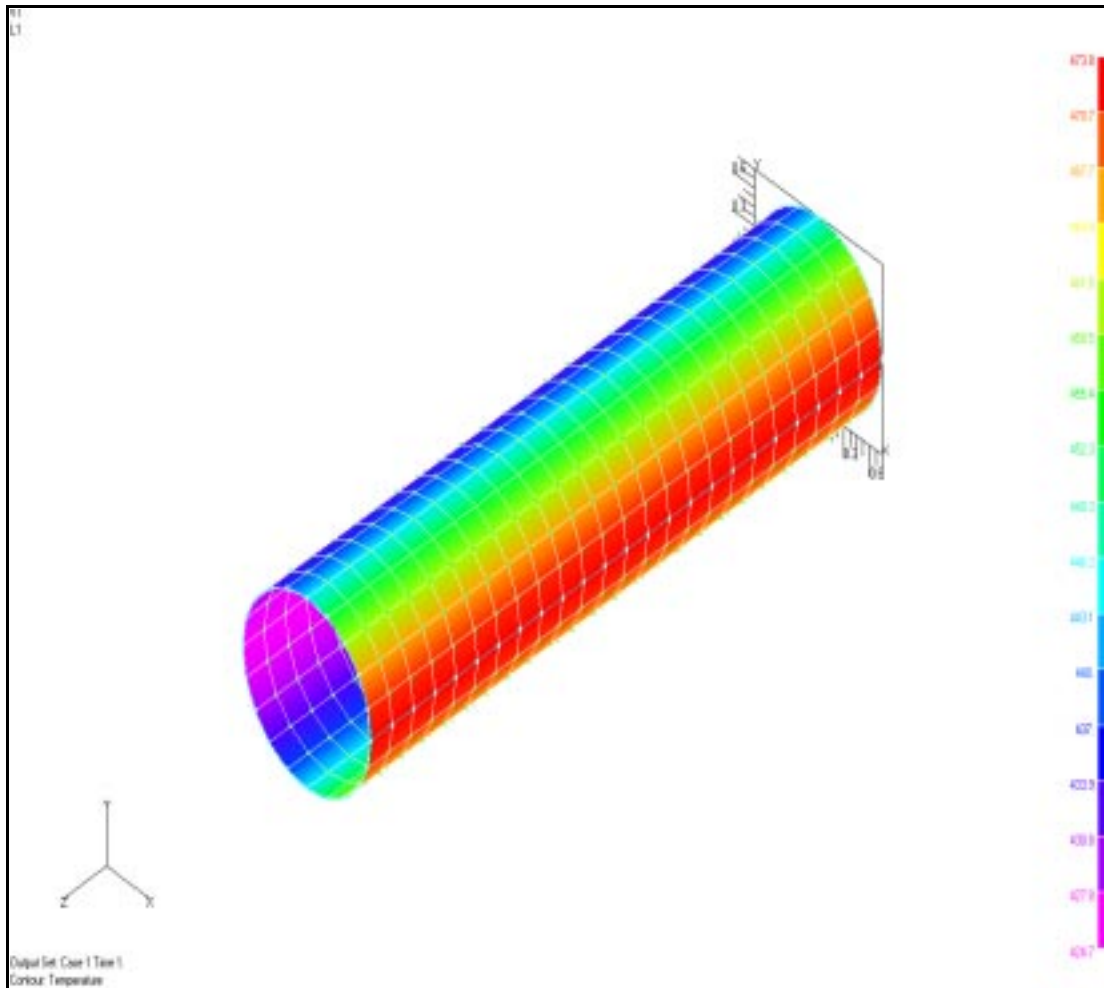
31..Temperature

OK

OK

Notice the effects of the direction of the heat flux and radiation on the temperature distribution in Figure 5-2.

Figure 5-2: Tube modeled with directional heat loads



When done, exit MSC.Nastran for Windows.

File/Save

File/Exit

This concludes this exercise.