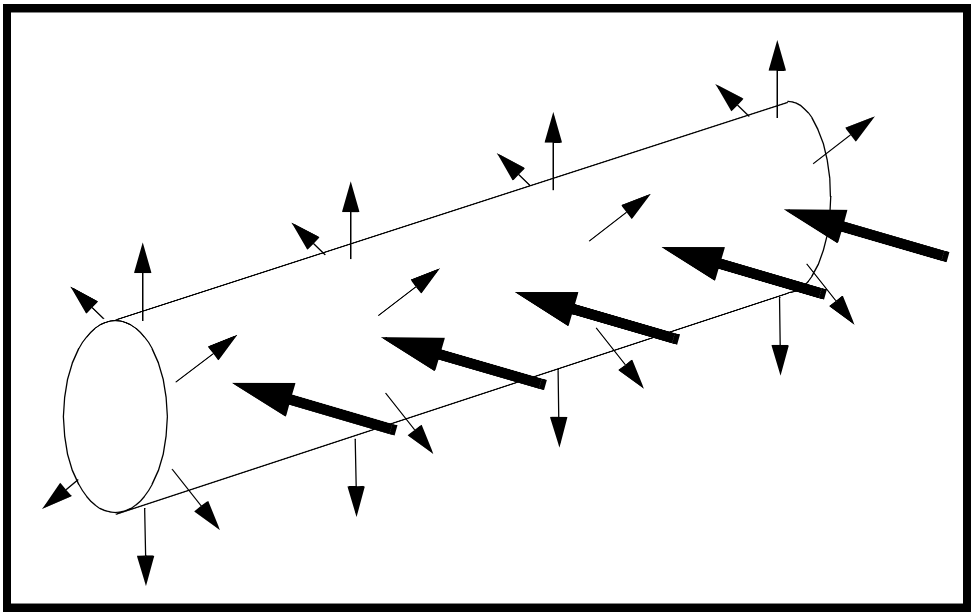
LESSON 4

Directional Heat Loads

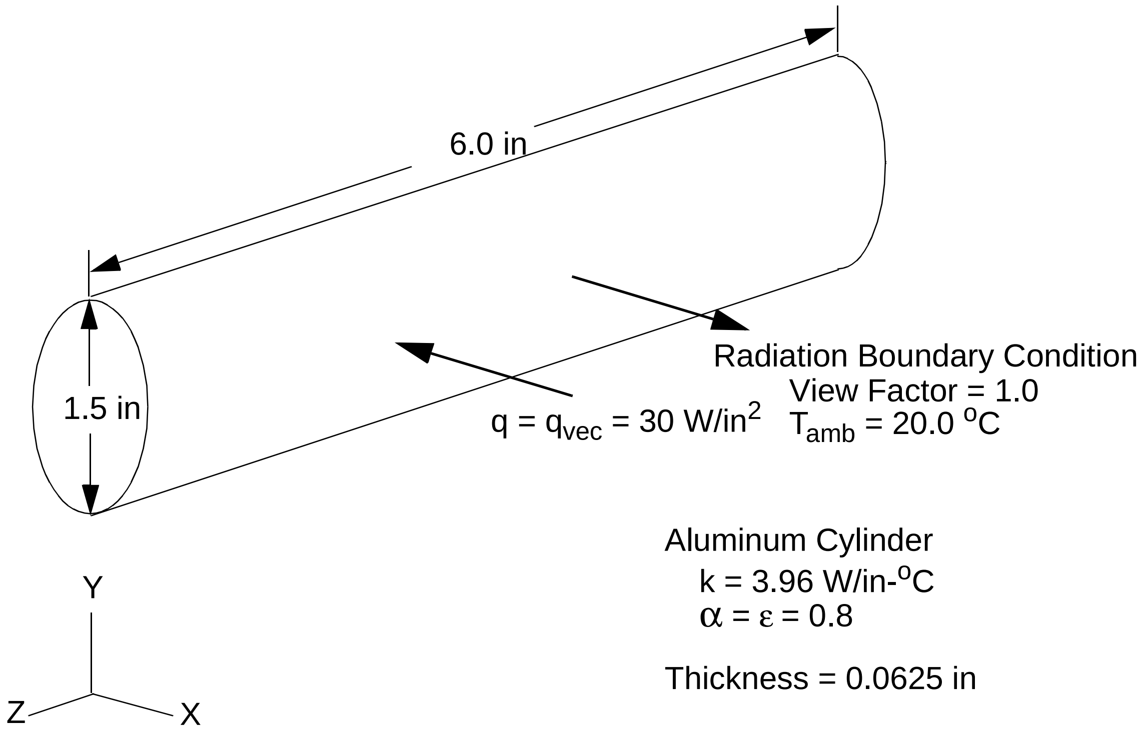


Objectives:

- 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) 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/in^2. In this exercise you will determine the steady state temperature distribution of the model.



Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0.2 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.2.

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

Open Model File:	**New Model**

2. Create the NASTRAN geometry for the plate.

Geometry/Surface/Cylinder...

First, define the center and the height.

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**6**

OK

Next, define the direction toward start of surface.

Base:	**0**	**0**	**0**
Tip:	**1**	**0**	**0**

OK

Bottom Outer:	**0.75**

OK

Cancel

3. Use Autoscale and Rotate to better view the model.

View/Rotate... **(or use <F8>)**

Isometric

OK

4. Create a material called **alum**. In anticipation of a future structural analysis, we will take this opportunity to simulataneously describe the material properties of the structure.

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

Model/Material...

Title:	alum
Youngs Modulus, E:	1.0e7
Poisson's Ratio, nu:	0.34
Expansion Coeff, a:	1.3e-5
Conductivity, k:	3.96

OK

Cancel

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

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

Model/Property...

Title:	tube

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

Material:	**1..alum**
Thickness, Tavg or T1:	0.0625

OK

Cancel

6. Define the mesh size on the tube.

Mesh/Mesh Control/Mapped Divisions on Surface

Select Surface 1. To do this, you can either click on the yellow surface and hitting “OK” or enter it manually as shown below.

ID:	**1**

OK

The Messages and Lists window should confirm with “1 Surface(s) Selected”.

Now input the mesh size on surface.

	s:	*t:*
Number of Elements:	**20**	**24**
Bias:	**1.**	**1.**

OK

Cancel

Repeat the foregoing steps to prescribe an equivalent mesh density for *Surface* 2. Note, however, that the parameterizaton of *Surface* 2 is opposite to that of *Surface* 1 (i.e. the number of nodes along the *s* and *t* directions will need to be reversed)

7. Now, create the mesh for the model.

Mesh/Geometry/Surface

Select All

OK

Property:	**1..tube**

OK

8. Remove the labels from the screen.

View/Options...

Quick Options... **(or use <CTRL Q>)**

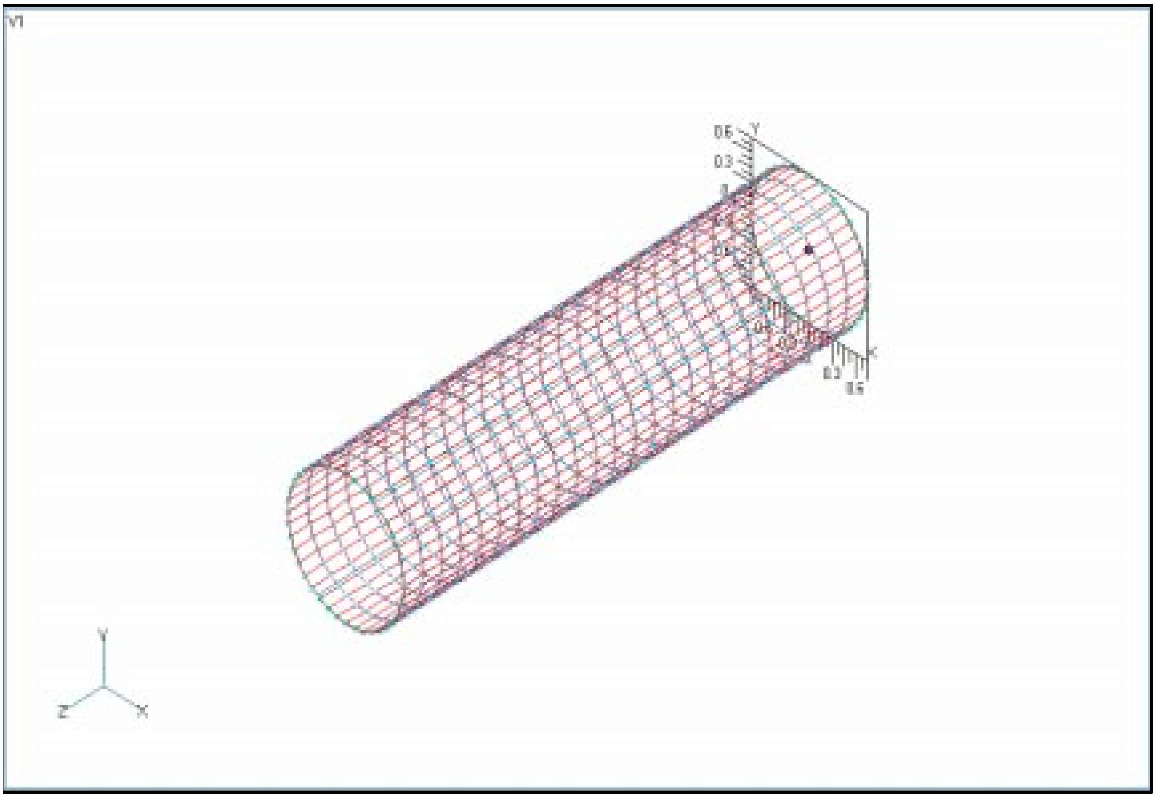
Labels Off

Done

OK

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

Figure 4-1: Meshed Model



9. 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: ☒ **Merge Coincident Entities**

OK

10. 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: **load1**

OK

Next, apply a uniform default temperature to the model.

Model/Load/Body...

(next to Thermal options) ☒ **Active**

Default Temperature: **500**

OK

11. Set up heat transfer properties by the following procedure:

Model/Load/Heat Transfer

Temp Offset from Abs Zero: **273.15**

Stefan-Boltzmann: **3.658e-11**

OK

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

View/Options... **(or use <F6>)**

Options: **Element - Directions**

Show Direction ☒

Normal Style: **1..Normal Vectors**

Apply

OK

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

View/Rotate... **(or use <F8>)**

XY Top

OK

If the normal vectors are all pointing at the outward direction, return to the previous view point and turn off the vectors.

View/Rotate...

Isometric

OK

View/Options...

Show Direction ☐

OK

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

First, create the heat flux.

Model/Load/Elemental...

Select All

OK

Type: ● **Heat Flux**

Load: ☒ **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

14. Create the input file and run the analysis.

File/Analyze

Analysis Type:	**20..Steady-State Heat Transfer**
	☒ **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 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

15. Remove the thermal loading markers from the screen.

View/Options...

Quick Options...

☐ **Load - Heat Flux**

☐ **Load - Radiation**

Done

OK

16. Create a final temperature distribution contour plot.

View/Select...

Model Style: ● **Hidden Line**

Contour Style: ● **Contour**

Deformed and Contour Data...

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

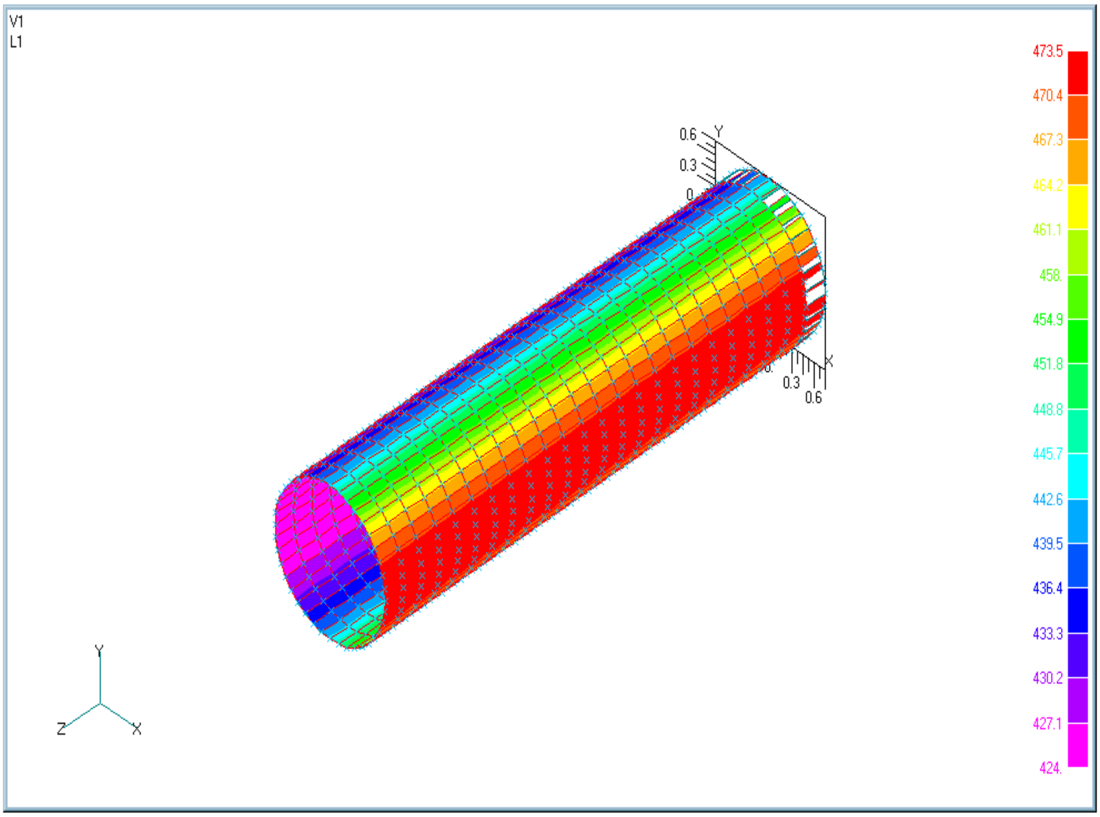
Contour: **31..Temperature**

OK

OK

Notice the effects of the direction of the heat flux and radiation on the temperature distribution in Figure 4-2.

Figure 4-2: Tube modeled with directional heat loads



When done, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.