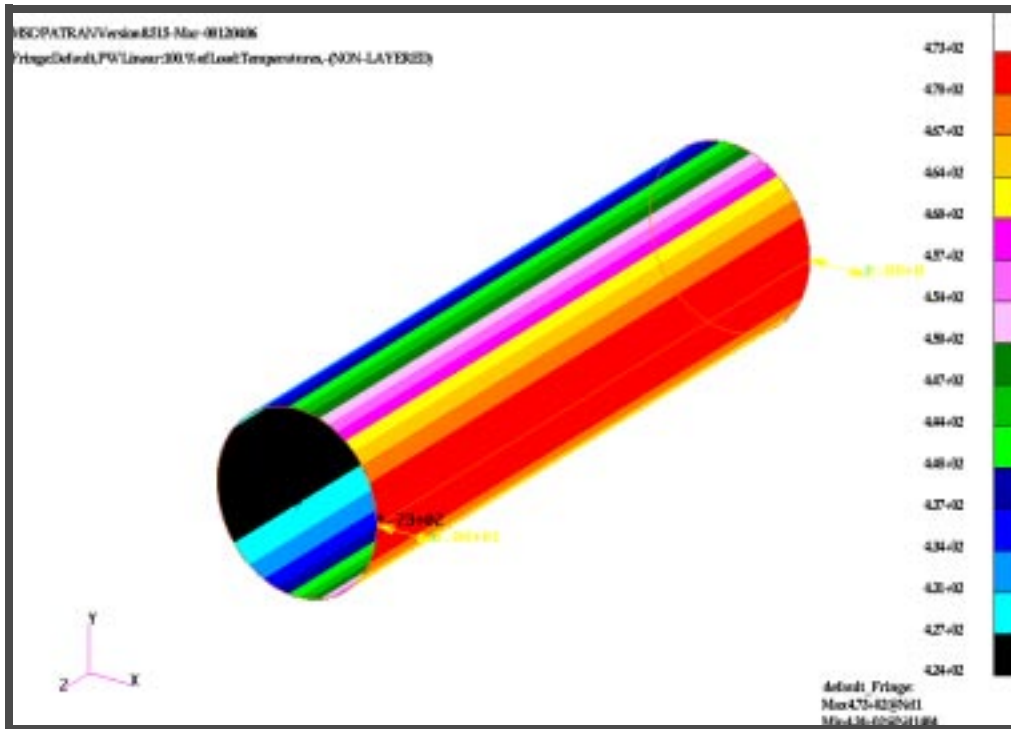


# WORKSHOP 8

## *Directional Heat Loads*



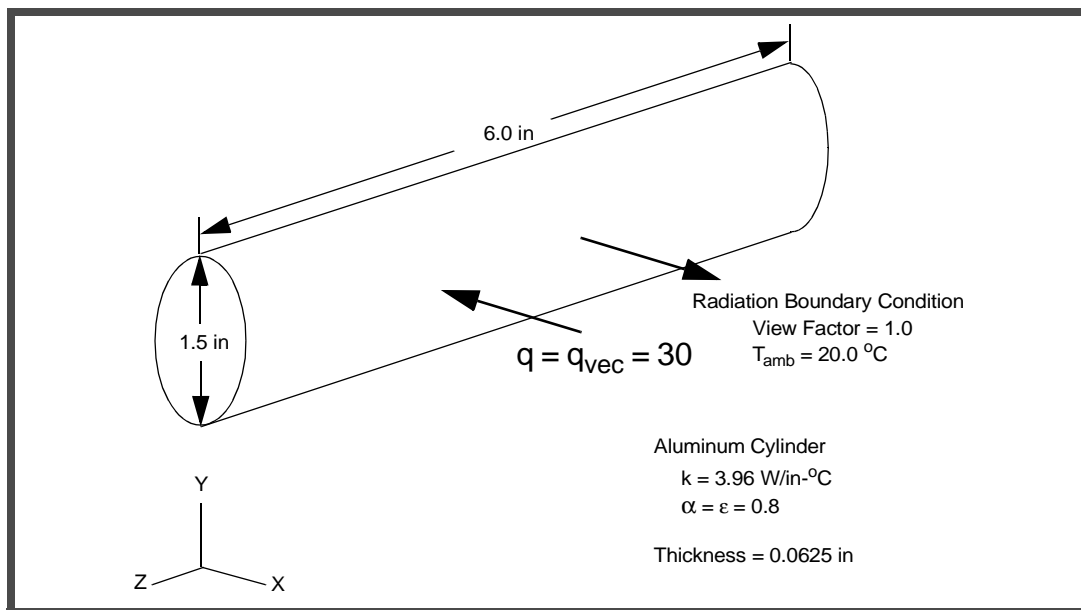


### Model Description:

In this example we will apply a directional heat load on cylinder. We will orient the surface normal from the surface such that the normal vector (Right hand rule) will point away from the surface. This allows the incoming directional heat flux to see the normals, and project the correct energy by forming a dot product with this vector. A typical application of this directional heat load process is in an orbital heating environment.

The dimension of the cylinder is 1.5 inch in diameter with a length of 6 inches. The material is aluminum with a thermal conductivity of 3.96 W/in-°C. The absorptivity and emissivity of the cylinder surface are 0.8. The directional heat load is 30 W/in<sup>2</sup>. The exterior surface of the cylinder loses heat by radiation to space. The radiation view factor is 1.0 and the ambient temperature is 20 °C.

**Figure 8.1**





## Suggested Exercise Steps:

- Create a new database called
- Create the Surfaces of Printed Circuit Board and Electric Components.
- Extrude the Surfaces to Create Solids.
- Mesh the Solids.
- Specify Materials.
- Define Element Properties.
- Merge the Common Nodes.
- Verify the Free Edges.
- Apply a heat load on each device.
- Apply a convection boundary condition on the PCB.
- Perform the Analysis.
- Read the analysis results.
- Display the results.



## Exercise Procedure:

1. Open a new database. Name it **ex8.db**

**File/New...**

*New Database Name:*

**ex8**

**OK**

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC.PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC.Nastran**

*Tolerance:*

◆ **Based on Model**

*Analysis Code:*

**MSC/NASTRAN**

*Analysis Type:*

**Thermal**

**OK**

2. Create the geometry.

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Point**

*Method:*

**XYZ**

*Point ID List:*

**1**

*Refer. Coordinate Frame:*

**Coord 0**

*Point Coordinates List:*

**[0.75 0 0]**

**Apply**

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Curve**

*Method:*

**Revolve**

---

*Curve ID List:*

*Total Angle:*

**Auto Execute**

*Point List:*

◆ **Geometry**

*Action:*

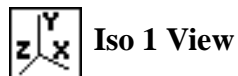
*Object:*

*Method:*

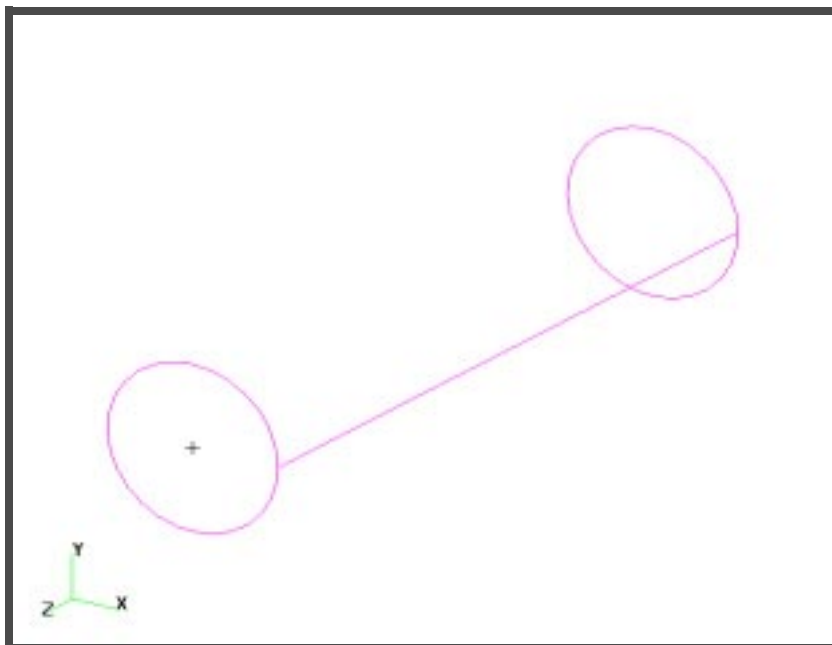
*Translation Vector:*

*Curve List:*

Click on the **Iso 1 View** icon to obtain a 3D view of the cylinder.



Your model should look like the following figure.



The surface normal direction is important in this problem, because the incoming heat flux vector will form a dot product with the normal vector for the surface generating the correct projected surface area for application of the heat load. Therefore, when we created the cylinder using geometry, we should verify that the normal vector points outward. This is accomplished by using:

◆ **Geometry**

<i>Action:</i>	<input type="text" value="Show"/>
<i>Object:</i>	<input type="text" value="Surface"/>
<i>Method:</i>	<input type="text" value="Normal"/>
<input type="checkbox"/> <b>Auto Execute</b>	
<i>Surface List:</i>	<input type="text" value="Surface 1"/>

Click on the **Front View** icon.




Select Surface 1 to make sure that the normal vector indicated by the red arrow points outward from the cylinder. If the normal vector is pointing inward, then you can reverse the surface normal by using the following command:

◆ **Geometry**

<i>Action:</i>	<input type="text" value="Edit"/>
<i>Object:</i>	<input type="text" value="Surface"/>
<i>Method:</i>	<input type="text" value="Reverse"/>
<input type="checkbox"/> <b>Auto Execute</b>	
<i>Surface List:</i>	<input type="text" value="Surface 1"/>

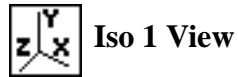
---

3. Create Finite Elements.

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Surface"/>
<i>Global Edge Length:</i>	<input type="text" value="0.1"/>
<i>Element Topology:</i>	<input type="text" value="Quad4"/>
<i>Surface List:</i>	<input type="text" value="Surface 1"/>
<input type="button" value="Apply"/>	

Click on the **Iso 1 View** icon.

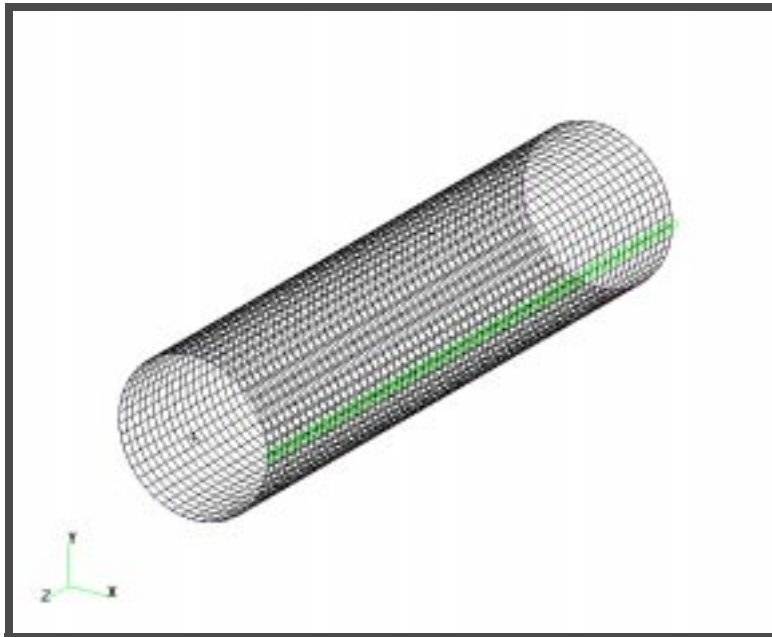


4. Remove Coincident Nodes.

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Equivalence"/>
<i>Object:</i>	<input type="text" value="All"/>
<i>Type:</i>	<input type="text" value="Tolerance Cube"/>
<i>Equivalencing Tolerance:</i>	<input type="text" value="0.005"/>
<input type="button" value="Apply"/>	

Your model should look like the following figure.



5. Specify Material Properties.

◆ **Materials**

*Action:*

**Create**

*Object:*

**Isotropic**

*Method:*

**Manual Input**

*Material Name:*

**alum**

**Input Properties...**

*Constitutive Model:*

**Solid properties**

*Thermal Conductivity:*

**3.96**

**Apply**

**Cancel**

6. Define Element Properties.

◆ **Properties**

*Action:*

**Create**

*Object:*

**2D**

Type:	<input type="text" value="Shell"/>
Property Set Name:	<input type="text" value="alum"/>
<b>Input Properties...</b>	
Material Name:	<input type="text" value="m:alum"/>
Thickness:	<input type="text" value="0.0625"/>
<input type="text" value="OK"/>	
Select Members:	<input type="text" value="Surface 1"/>
<input type="text" value="Add"/>	
<input type="text" value="Apply"/>	

7. Apply a Directional Heat Load.

◆ Load/BCs

Action:	<input type="text" value="Create"/>
Object:	<input type="text" value="Applied Heat"/>
Option:	<input type="text" value="Directional Fluxes"/>
Type:	<input type="text" value="Element Uniform"/>
New Set Name:	<input type="text" value="vector_flux"/>
Target Element Type:	<input type="text" value="2D"/>
<b>Input Data...</b>	
Surface Option:	<input type="text" value="Top"/>
Top Surf Absorptivity:	<input type="text" value="0.8"/>
Top Surf Heat Flux:	<input type="text" value="30"/>
Incident Thermal Vector:	<input type="text" value="&lt;-1 0 0&gt;"/>
<input type="text" value="OK"/>	
<b>Select Application Region...</b>	

Geometry Filter:	◆ Geometry
Select Surface or Edges:	<input type="text" value="Surface 1"/>
<input type="text" value="Add"/>	

<b>OK</b>
<b>Apply</b>

8. Apply a Radiation Boundary Condition.

◆ **Load/BCs**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Radiation</b>
<i>Type:</i>	<b>Element</b> <b>Uniform</b>
<i>Option:</i>	<b>Ambient Space</b>
<i>New Set Name:</i>	<b>rad_space</b>
<i>Target Element Type:</i>	<b>2D</b>

**Input Data...**

<i>Surface Option:</i>	<b>Top</b>
<i>Top Surf Emissivity:</i>	<b>0.8</b>
<i>Top Surf Absorptivity:</i>	<b>0.8</b>
<i>Ambient Temperature:</i>	<b>20</b>
<i>View Factor:</i>	<b>1.0</b>

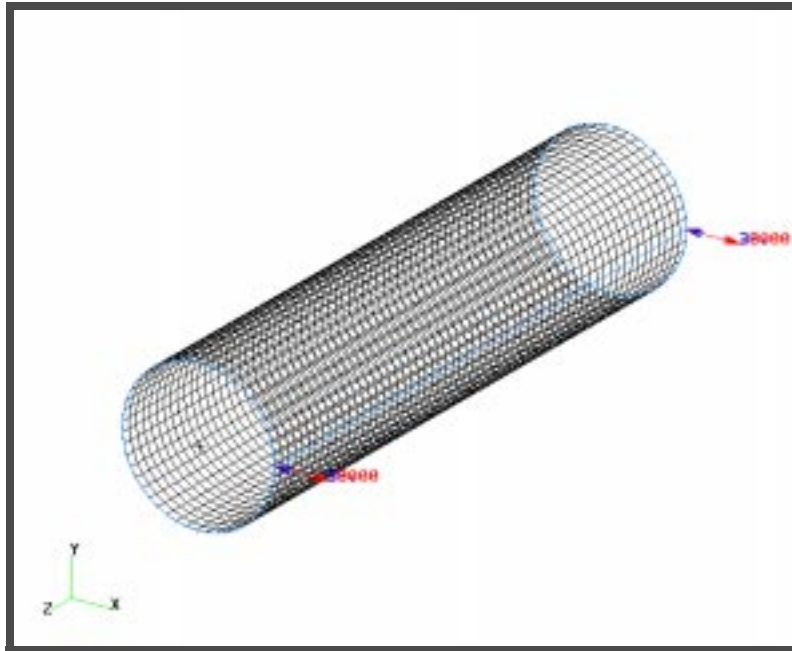
<b>OK</b>
-----------

**Select Application Region...**

<i>Geometry Filter:</i>	◆ <b>Geometry</b>
<i>Select Surface or Edges:</i>	<b>Surface 1</b>

<b>Add</b>
<b>OK</b>
<b>Apply</b>

Your model should look like the following figure.



9. Specify Radiation Parameters and Perform the Analysis.

◆ **Analysis**

*Action:*

Analyze

*Object:*

Entire Model

*Method:*

Analysis Deck

*Job Name:*

ex8

Solution Type...

◆ **STEADY STATE ANALYSIS**

Solution Parameters...

Radiation Parameters...

*Absolute Temperature Scale:*

273.15 Degree Celsius

*Stefan-Boltzmann Constant:*

3.6580e-11 WATTS/IN2/K4

- OK
- OK
- OK
- Apply

An MSC.Nastran input file called **ex8.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 ex8.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 ex8 scr=yes**. Monitor the run using the UNIX **ps** command.
11. When the run is completed, edit the **ex8.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 **ex8.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

*Action:*

**Attach XDB**

*Object:*

**Result Entities**

*Method:*

**Local**

**Select Results File**

*Select Results File*

**ex8.xdb**

**OK**

**Apply**

- 14. Display the Results.

◆ **Results**

*Select Results Cases:*

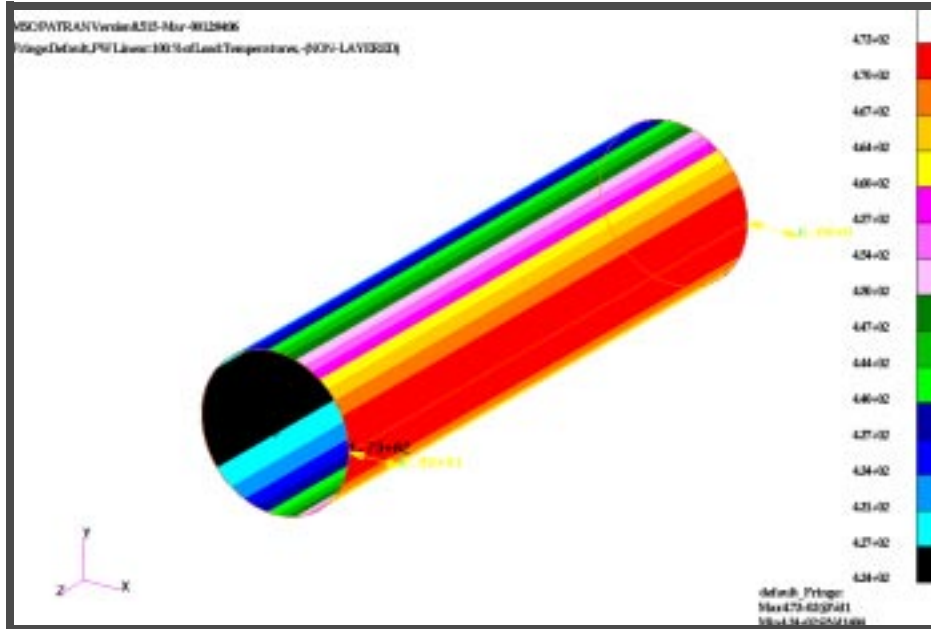
**Default, PW Linear: 100. % of Load**

*Select Fringe Result:*

**Temperatures**

**Apply**

Your model should look like the following figure.



Example 8 demonstrates an aluminum cylinder in radiative equilibrium. The heat source is directional (light source oriented), and the radiation boundary condition is equal for all directions. The cylinder's maximum temperature ( $\sim 473^{\circ}\text{C}$ ) is attained on the side subject to the solar heat load. The minimum temperature ( $\sim 424^{\circ}\text{C}$ ) occurs in the shadow region. The high conductivity of the cylinder helps to equilibrate the temperatures. If the conductivity were very low, the maximum temperature would approach  $740^{\circ}\text{C}$  with the minimum approximately  $20^{\circ}\text{C}$ .

Quit MSC.Patran when you have completed this exercise.