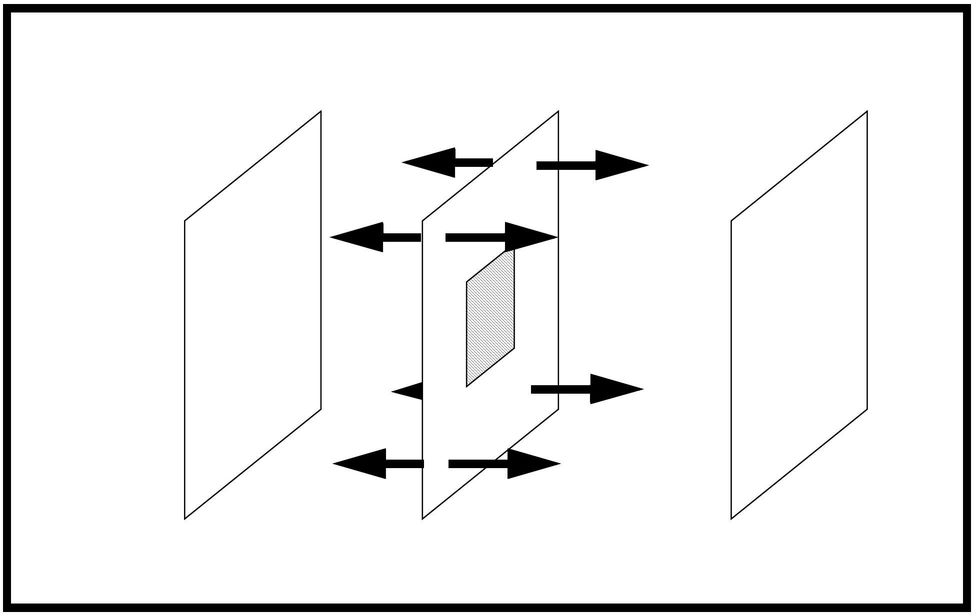
LESSON 3

Radiation Enclosures

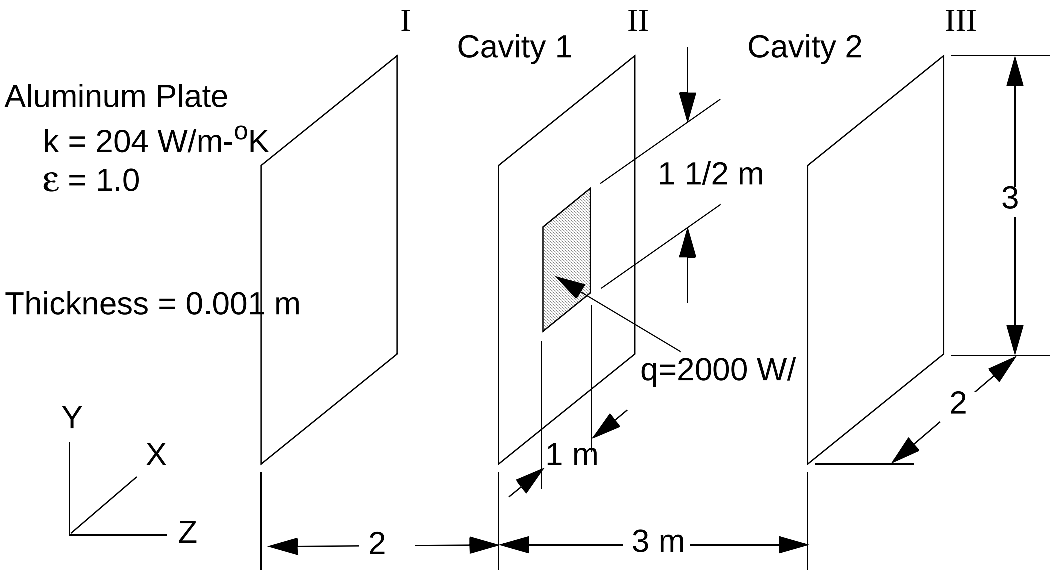


Objectives:

- Create a geometric representation of three plates.
- Aply a thermal load of heat flux to a patch in the middle surface.
- Apply thermal load of shaded radiation to the outer surfaces.
- Run a steady-state heat transfer analysis of the plates.

Model Description:

Below is shown a model consisting of 3 plates. Each plate measures 2 x 3 meters, and has a thickness of 1mm. The plates are model with an emissivity of 1.0, making them perfect blackbodies. The center plate has an applied heat flux of 2000 W/m^2. The middle plate (II) will radiate heat to the other two plates, and will result in a steady state temperature distribution for all three plates.



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 middle plate.

Geometry/Surface/Plane...

OK

Width (along Plane X):	**2**
Height (along Plane Y):	**3**

OK

Cancel

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale... **(or <CTRL A>)**

3. Copy the surface to create the second plate.

Geometry/Copy/Surface...

Select All

OK

OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**2**

OK

4. Copy the surface to create the last plate.

First, rotate the view to better see the model.

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

Isometric

OK

Next, copy the surface to create the last plate.

Geometry/Copy/Surface...

*(select surface on left - **Surface 2**)*

OK

OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**3**

OK

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale...

5. Create a material called **alum**.

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

Model/Material...

Title: **alum**

Conductivity, k: **204**

OK

Cancel

6. Create a property called **cquad8** to apply to the members of the plates.

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

Model/Property...

Title: **cquad8**

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

Material: **1..alum**

Elem/Property Type...

☒ **Parabolic Elements**

OK

Thickness: **0.001**

OK

Cancel

7. Set the default size for the mesh.

Mesh/Mesh Control/Default Size...

Size: **0.5**

OK

8. Create the mesh for the model.

Mesh/Geometry/Surface

Select All

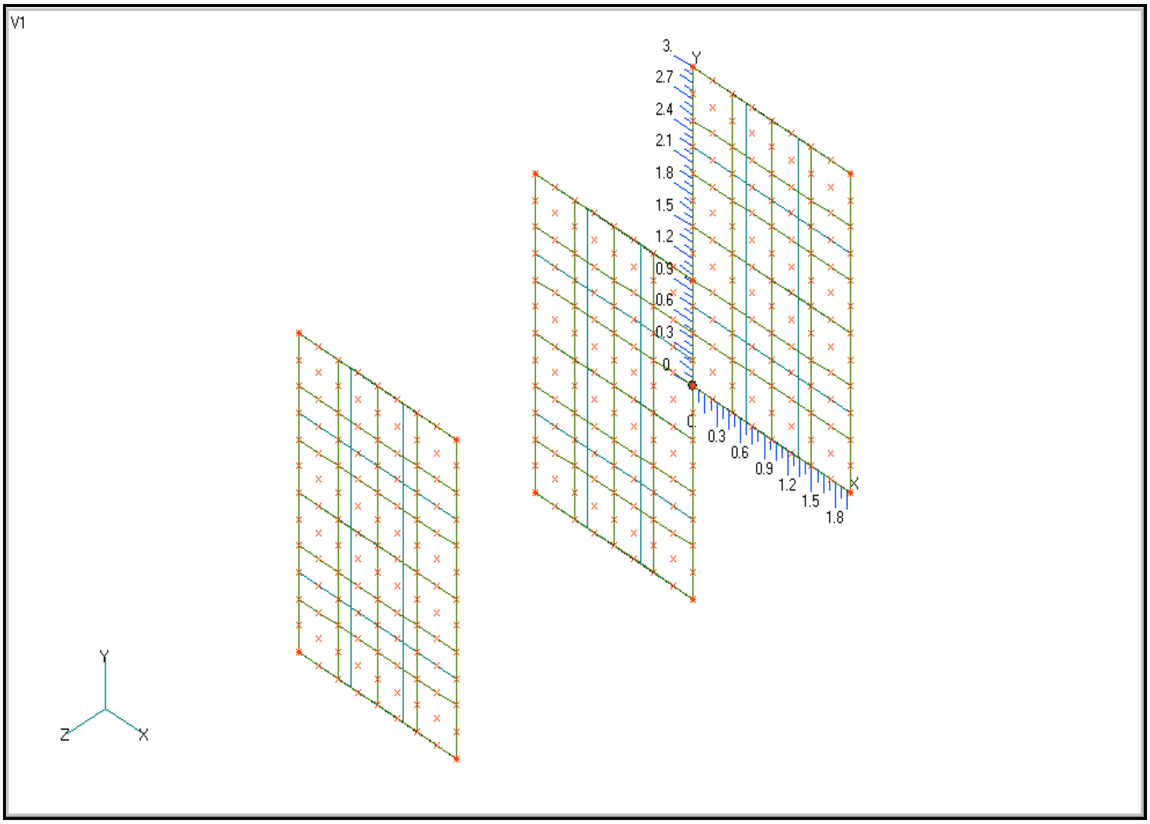
OK

Property: **1..cquad8**

OK

You model should look like the following:

Figure 3-1: The meshed model.



9. Remove the labels from the screen.

View/Options... **(or use <CTRL Q>)**

Quick Options...

Labels Off

Done

OK

10. Remove the excess nodes of the model.

In NASTRAN for Windows, our **cquad8** elements have been created with an extra grid in the middle. We will want to delete this node from all elements in order to make them true CQUAD8 elements.

Delete/Model/Node

Select All

OK

When asked if it is “OK to delete 351 selected nodes?”, respond **Yes**.

Yes

Clean up the view of the model.

View/Regenerate **(or use <CTRL G>)**

11. Modify the element normals so that the outer surface normals face the middle surface.

First, you will need to plot the element normal vectors on the screen.

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

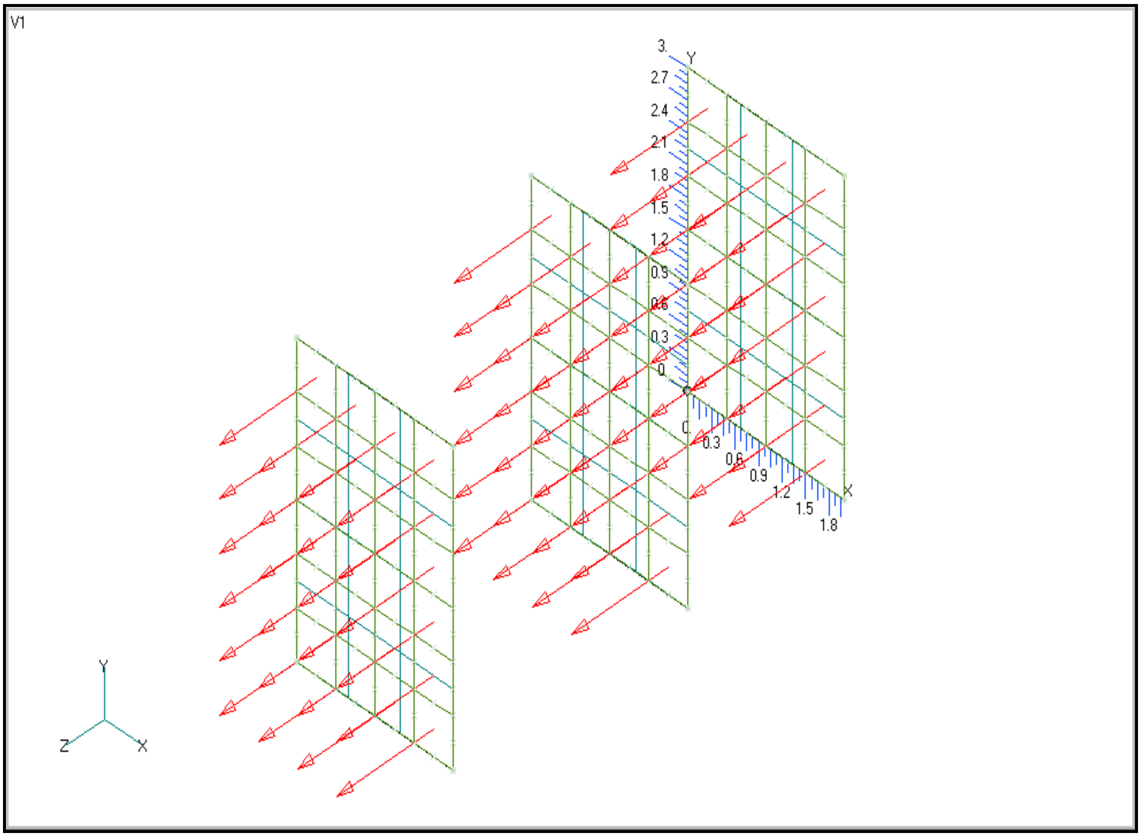
Options:	**Element-Directions**
Normal Style:	**1..Normal Vectors**
	Show Direction ☒

Apply

OK

In Figure 3-2, notice that the element normals for the left surface point in the wrong direction. You will need to reverse the normal direction for those elements.

Figure 3-2: The model with element direction.



Modify/Update Element/Reverse...

Hold down the shift key and drag a box around the elements on the left surface.

OK

● **Reverse Normal Direction**

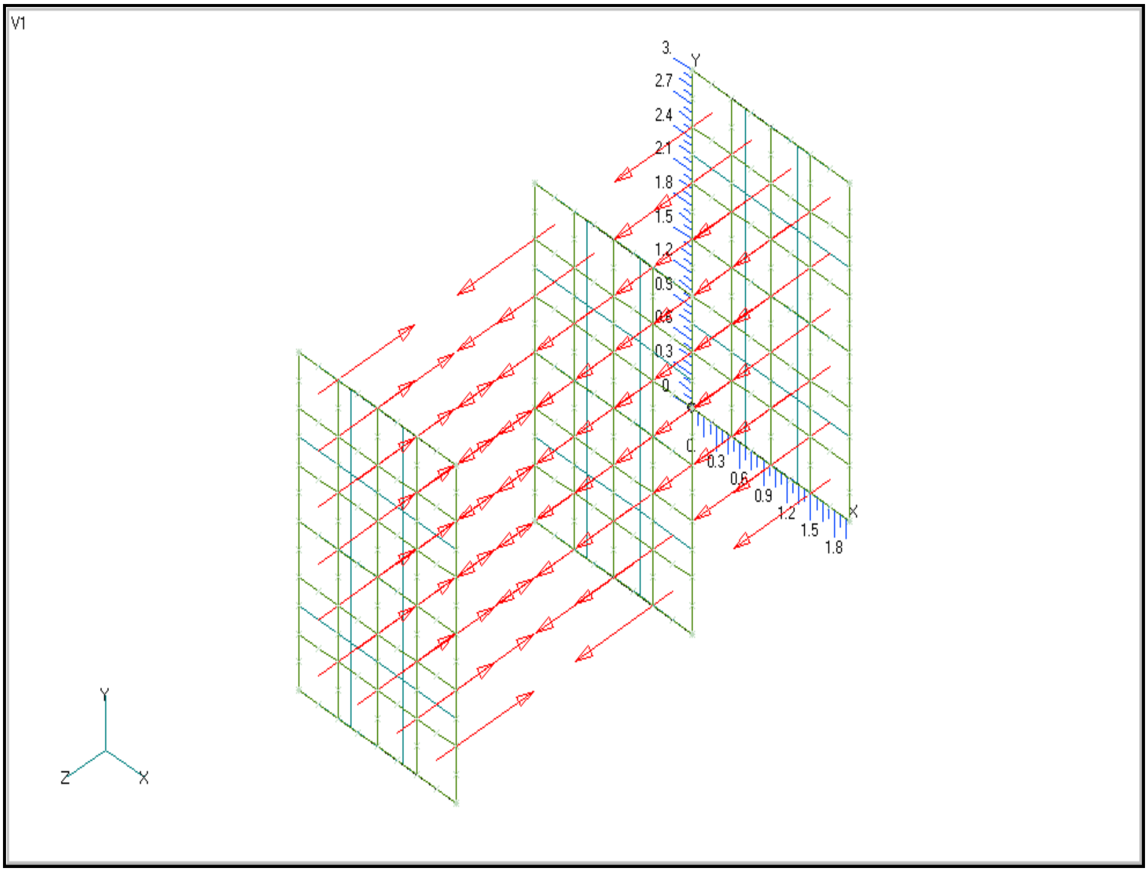
OK

Clean up the view of the model.

View/Regenerate

You model should appear as Figure 3-3.

Figure 3-3: The model with correct element directions.



12. Input a body load defining the Stefan-Boltzmann constant for radiative heat transfer in the model.

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

Model/Load/Set...

Title: **load_1**

OK

Next, set the default temperature.

Model/Load/Body...

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

Default Temperature: **500**

OK

Lastly, set the Stefan-Boltzmann coefficient.

Model/Load/Heat Transfer

Stefan-Boltzmann: **5.6696e-8**

OK

13. Create the heat flux loading for the center plate.

Model/Load/Elemental...

Select the four elements in the center of the middle surface.

OK

Type: ● **Heat Flux**

Flux: **2000**

OK

Face: **1**

OK

By specifying *Face* **1**, the user has declared that the heat flux is to be applied to the 'top' of the elements. However, since this flux is applied to 2D elements and is not directional, it does not matter whether the flux is applied to the top or to the bottom of these elements (i.e. this flux is equivalent to specifying heat generation within the 2D elements).

14. Create the emissivity of the absorbing surfaces.

Method: **On Surface**

Select the left and right surfaces.

OK

Type: ● **Radiation**

☒ **Enclosure Radiation**

☒ **Can Be Shaded**

Emissivity: **1**

OK

Face: **1**

OK

Here, given the orientation of the radiating element surfaces on the absorbing surfaces, it is crucial that the radiation condition be applied to the elements’ top faces.

15. Create the emissivity for the top and bottom radiating surfaces

Method: **On Surface**

Select the middle surface.

OK

Type: ● **Radiation**

☒ **Enclosure Radiation**

☒ **Can Shade**

Emissivity: **1**

OK

Face: **1**

OK

Method: **On Surface**

Again, select the middle surface.

OK

Type: ● **Radiation**

☒ **Enclosure Radiation**

☒ **Can Shade**

Emissivity: **1**

OK

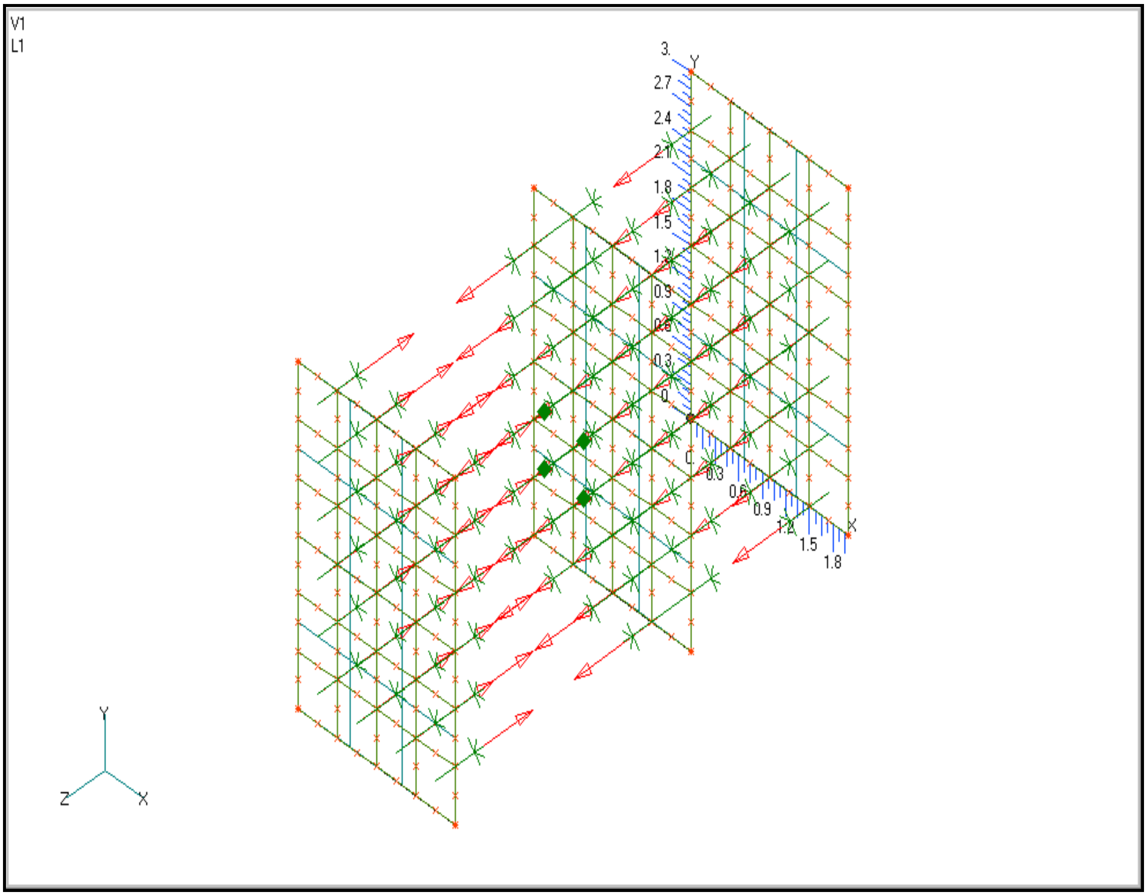
Face: **2**

OK

Cancel

With all the loads applied to your model, it should appear as Figure 3-4

Figure 3-4: The model loaded with heat flux and radiation.



16. 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: **radiate**

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

17. Remove the thermal loading markers, normal vectors, and geometry from the screen..

View/Options...

Quick Options...

Geometry Off

☐ **Load - Heat Flux**

☐ **Load - Radiation**

Done

Apply

Options: **Element-Directions**

Show Direction ☐

Apply

OK

18. Create a final temperature distribution contour plot.

View/Select...

Contour Style: ● **Contour**

Deformed and Contour Data...

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

Contour: **31..Temperature**

OK

OK

In Figure 3-5, notice how the middle surface temperature profile radiates from the center, showing the effects of the central heat flux and the radiative heat transfer.

Also notice that the plate which is three meters away from the center plate has a lower temperature that the plate which is two meters away.

When done, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Figure 3-5: Radiation enclosures