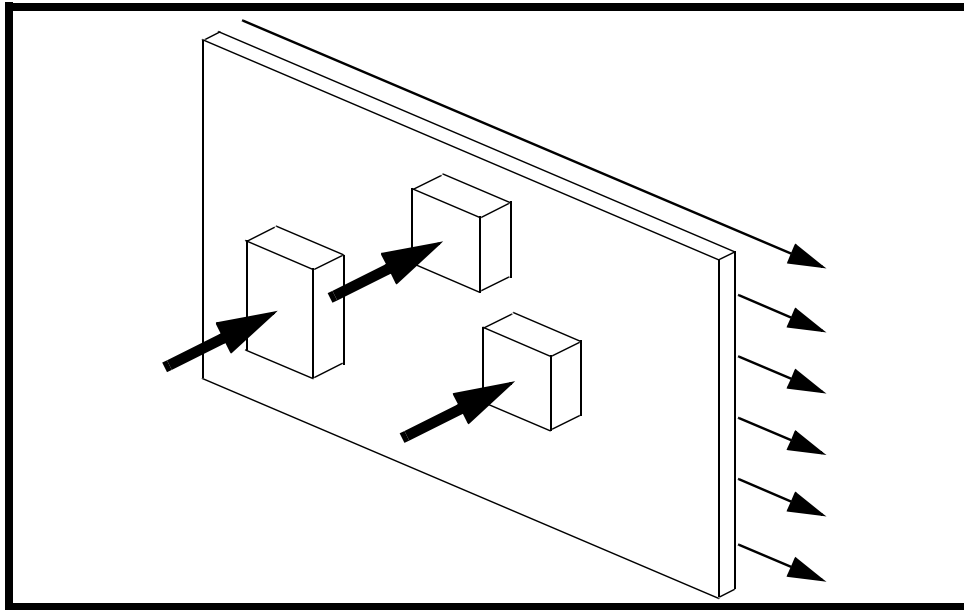

WORKSHOP 2

Forced Convection on a Printed Circuit Board

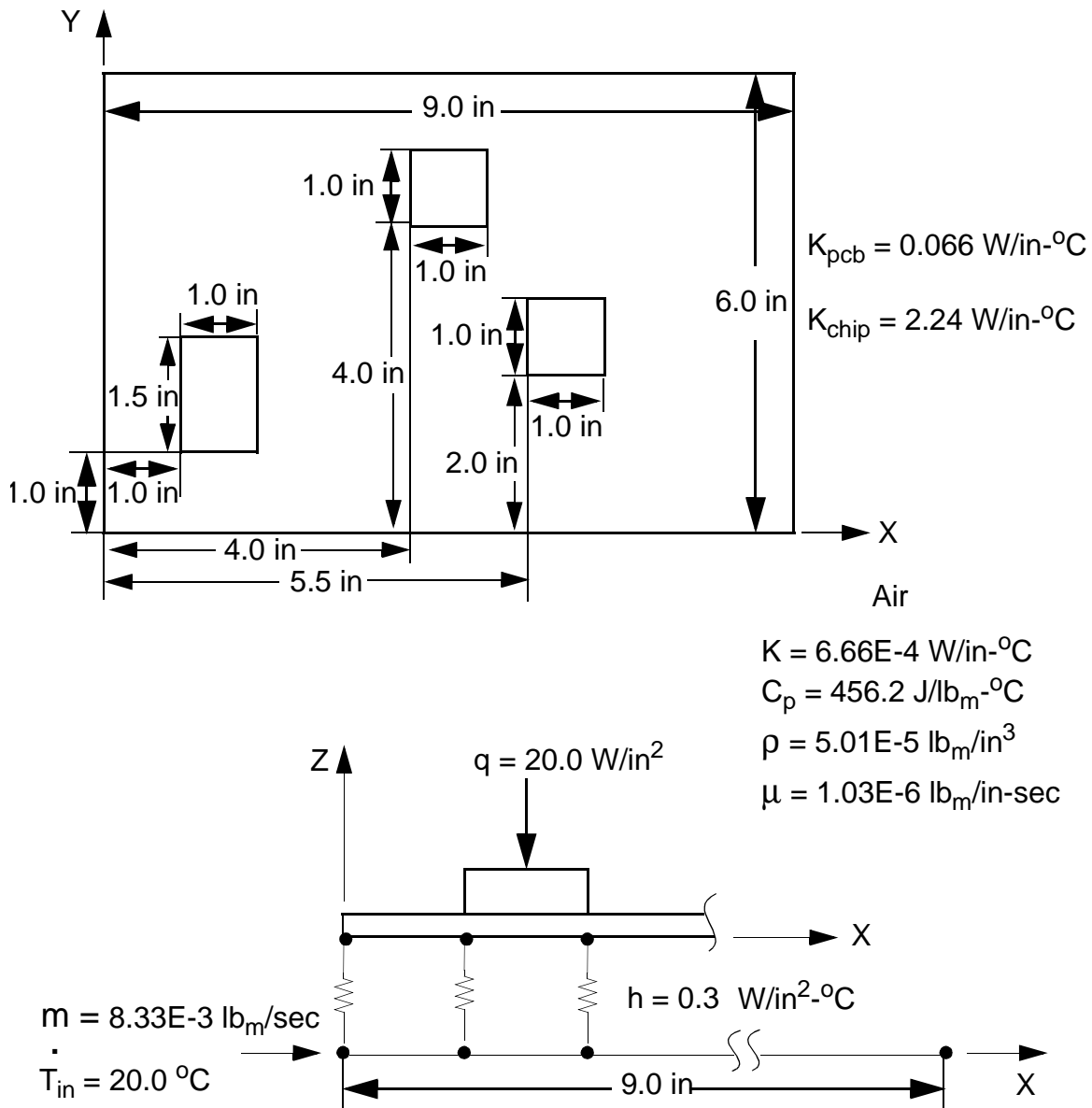


Objectives:

- Create a geometric representation of a printed circuit board.
- Apply thermal loading of forced convection and heat fluxes to the model.
- Run a steady-state heat transfer analysis of the pcb.

Model Description:

Below is shown a model for a printed circuit board, complete with dimensions, material properties, and thermal loading conditions. You will create this model and analyze it to determine the steady-state temperature distribution.



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. Turn off the Parasolid Geometry Engine.

Tools/Advanced Geometry

Standard

OK

3. Create the materials of the model.

First, create a material called **pcb**.

Model/Material...

Title:

pcb

Conductivity, k:

.066

OK

Next, create a material called **chip**.

Model/Material...

Title:

chip

Conductivity, k:

2.24

OK

Cancel

4. Create the element properties for the model.

First, create a property for the printed circuit board called **pcb**.

Model/Property...

Title:

Material:

Elem/Property Type...

Volume Elements: Solid

Next, create an element property for the chips called **chip**.

Title:

Material:

Finally, create a property for the thin membrane elements to be applied to the back of the board (**since convection in MSC.Nastran for Windows currently does not accept faces of solid elements as valid forced convection regions**).

Title:

Material:

Elem/Property Type...

Plane Elements: Membrane

Thickness:

5. Create the geometry for the model.

First, create surfaces to represent the board and the chips.

Geometry/Surface/Corners...

Create the board surface:

X:	Y:	Z:	
0	0	0	OK
9	0	0	OK
9	6	0	OK
0	6	0	OK

Create the first chip surface:

X:	Y:	Z:	
1	1	0	OK
2	1	0	OK
2	2.5	0	OK
1	2.5	0	OK

And the second chip surface:

X:	Y:	Z:	
4	4	0	OK
5	4	0	OK
5	5	0	OK
4	5	0	OK

And the third chip surface:

X:	Y:	Z:	
5.5	2	0	OK
6.5	2	0	OK
6.5	3	0	OK

X:	Y:	Z:	
5.5	3	0	OK
Cancel			

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

View/Autoscale... (or use <Ctrl>+ A)

Next, extrude the surfaces into solids.

Geometry/Volume/Extrude...

(Select the largest surface, representing the board.)

OK

	X:	Y:	Z:
Base:	0	0	0
Tip:	0	0	-.1

OK

(Select the three small surfaces, representing the chips.)

OK

	X:	Y:	Z:
Base:	0	0	0
Tip:	0	0	.25

OK
Cancel

6. Change the display to get a better view at the volume created.

Change the viewing angle.

View/Rotate...

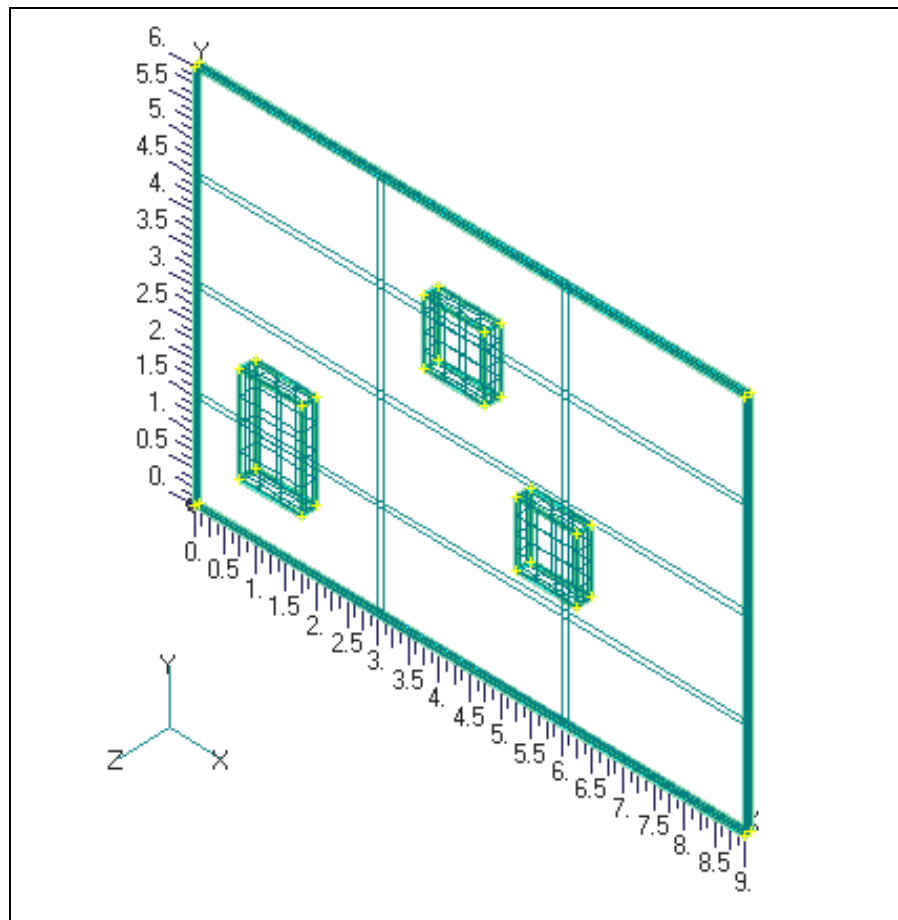
(or use <F8>)

Isometric

OK

Your model should be like the following:

Figure 2-1: The isometric view of the model.



7. Set the default size for the mesh.

Mesh/Mesh Control/Default Size...

Size:

0.25

OK

8. Create the mesh for the model.

First, turn off all labels to keep the screen from getting “messy”.

View/Options...

Quick Options...

(or use <Ctrl>+Q)

Labels Off

Done

Apply

OK

Next, create the mesh for the pcb (the circuit board).

Mesh/Geometry/Volume...

(Select the largest solid (the board).)

OK

Property:

1..pcb

OK

Next, create the mesh for the chips.

Mesh/Geometry/Volume...

(Select the three smaller solids (the chips).)

OK

Property:

2..chip

OK

Create the membrane elements on the back of the board.

Mesh/Between...

Property:

3..back

Mesh Size / #Nodes / Dir 1:

37

Mesh Size / #Nodes / Dir 2:

X:	Y:	Z:	
<input type="text" value="0"/>	<input type="text" value="0"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>
<input type="text" value="9"/>	<input type="text" value="0"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>
<input type="text" value="9"/>	<input type="text" value="6"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>
<input type="text" value="0"/>	<input type="text" value="6"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>

Create tube elements to help model the fluid flow.

Mesh/Between...

Title:

Material:

Line Elements: Tube

Mesh Size / #Nodes / Dir 1:

X:	Y:	Z:	
<input type="text" value="0"/>	<input type="text" value="3"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>
<input type="text" value="9"/>	<input type="text" value="3"/>	<input type="text" value="-1"/>	<input type="button" value="OK"/>

Finally, remove coincident nodes of the model.

Tools/Check/Coincident Nodes...

When asked, "OK to Specify Additional Range of Nodes to Merge?" respond with **No**.

No

Merge Coincident Entities

OK

9. Check element normal direction when directional load is involved.

For plate elements, it is important to check to make sure the surface normal is pointing in the right direction. To facilitate and ensure that only the membrane elements, put them in a group.

Group/Set...

Title:

membrane

OK

Group/Element/Property...

ID:

3

OK

Post the membrane elements only.

View/Select...

(or use <F5>)

Model Data...

Group:

Select **1.. membrane**

OK

OK

Check the surface normal direction.

View/Options...

(or use <F6>)

Options:

Element - Directions

Show Direction

Normal Style:

1.. Normal Vectors

OK

The last thing you will need to do is reverse the normal direction of the membrane elements, so that the normal points towards the tube elements.

Modify/Update Elements/Reverse...

Group:

1.. membrane

More

OK

Reverse Normal Direction

OK

Turn off the normal vectors.

View/Options...

Show Direction

OK

Return to the default group.

View/Select...

(or use <F5>)

Model Data...

Group:

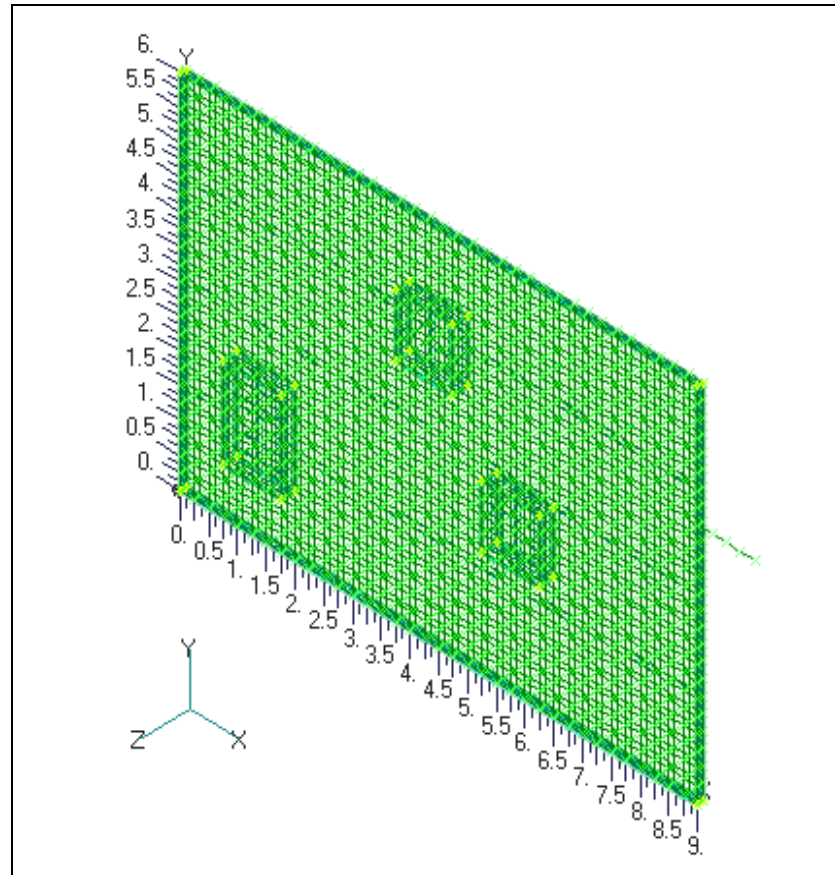
None

OK

OK

Your model should look like the following:

Figure 2-2: Meshed model.



10. Create the thermal 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 and define the parameters for the fluid flow.

Model/Load/Body...

(next to Thermal options)

 Active

Default Temperature:

OK

Model/Load/Heat Transfer

Constant Coefficient:

Reynolds Exponent:

Prandtl Exponent (into fluid):

Prandtl Exponent (out of fluid):

Fluid Conductivity:

Fluid Specific Heat:

Fluid Viscosity:

Fluid Density:

OK

Change the view to make applying the loads easier.

View/Rotate... <F8>

Back

OK

Apply a heat flux to the chips in the model.

Model/Load/Elemental...

Hold shift and drag a box around the bottom edges of all chip elements.

OK

Type:

Flux:

OK

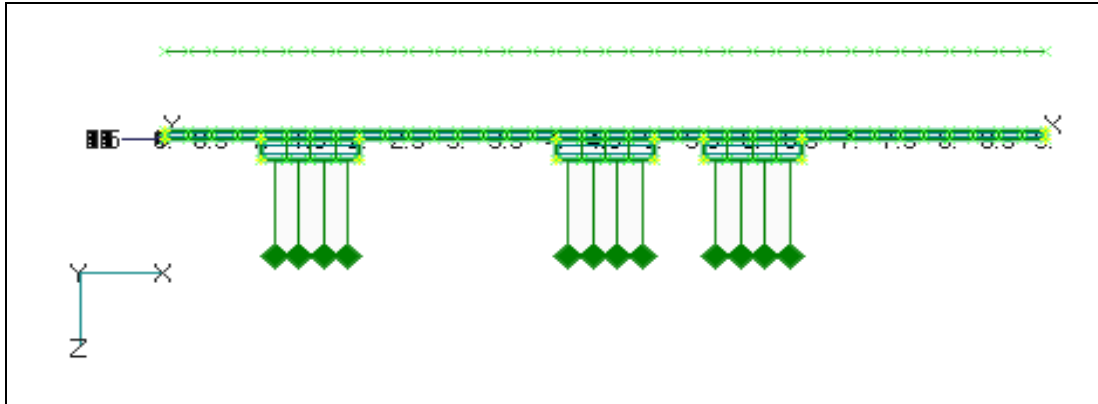
Face:

OK

Cancel

Your model should be like the following:

Figure 2-3: Model with thermal load and heat flux.



11. Create the inlet temperature for the fluid flow.

Model/Load/Nodal...

(Select the node on the left side of the curve.)

OK

Type:

Temperature

Temperature:

20

OK

Cancel

Apply forced convection to the back of the board in the model.

Model/Load/Elemental...

Group:

membrane

More

OK

Type:

Convection

Forced Convection

Disable Advection

Flow Rate:

Diameter:

Area Factor:

(Hold shift and drag a box around the tube elements.)

Type:

Forced Convection

Disable Convection

Flow Rate:

Diameter:

Temperature:

12. Change the display to get a better view at the loaded model.

Change the viewing angle.

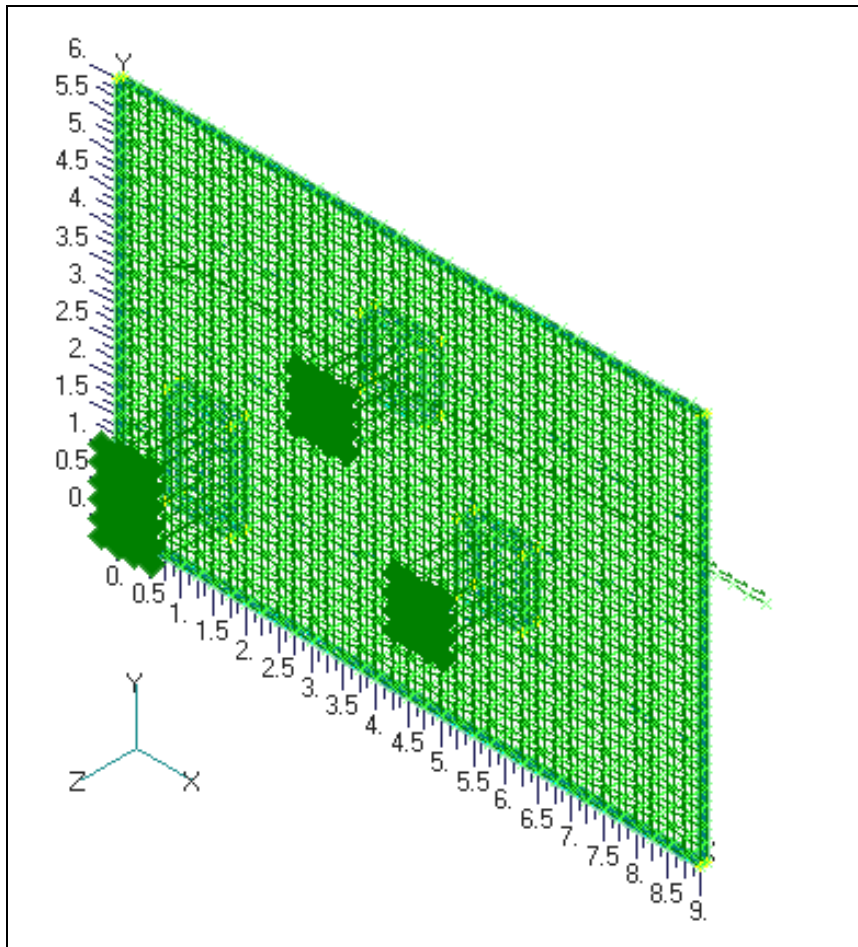
View/Rotate... <F8>

Isometric

OK

Your model should be like the following:

Figure 2-4: The loaded model.



-
13. The direction of the convection load within the membrane property is in the wrong direction. In order to fix it, do the following:

Modify/Update Elements/Reverse...

Group:

membrane

More

OK

Align First Edge to Vector

OK

Method^

(select Nodes)

Base Node ID:

37 <lower left corner node>

Tip Node ID:

925 <upper left corner node>

OK

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:

pcb2

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

15. Remove all geometry and thermal loading markers from the screen

View/Options...

Quick Options...

(or use <Ctrl>+Q)

Geometry Off

Load/Constraint Off

Done

OK

Turn off the workplane.

Tools/Workplane...

(or use <F2>)

Draw Workplane

Done

Create a final temperature distribution contour plot.

View/Select... <F5>

Model Style:

Quick Hidden Line

Render

Contour Style:

Contour

Deformed and Contour Data...

Contour:

31..Temperature

OK

OK

In **Figure 2-5**, notice the temperature gradients around the chips, where all the heat is produced. Also notice how the temperature of the board rises along the positive x-direction, downstream from the flow. Finally, notice that the maximum temperature is lower than the last exercise, and located in the downstream chip.

When done, exit MSC.Nastran for Windows.

File/Save

File/Exit

This concludes this exercise.

Figure 2-5; Thermal analysis of a circuit board with forced convection.

