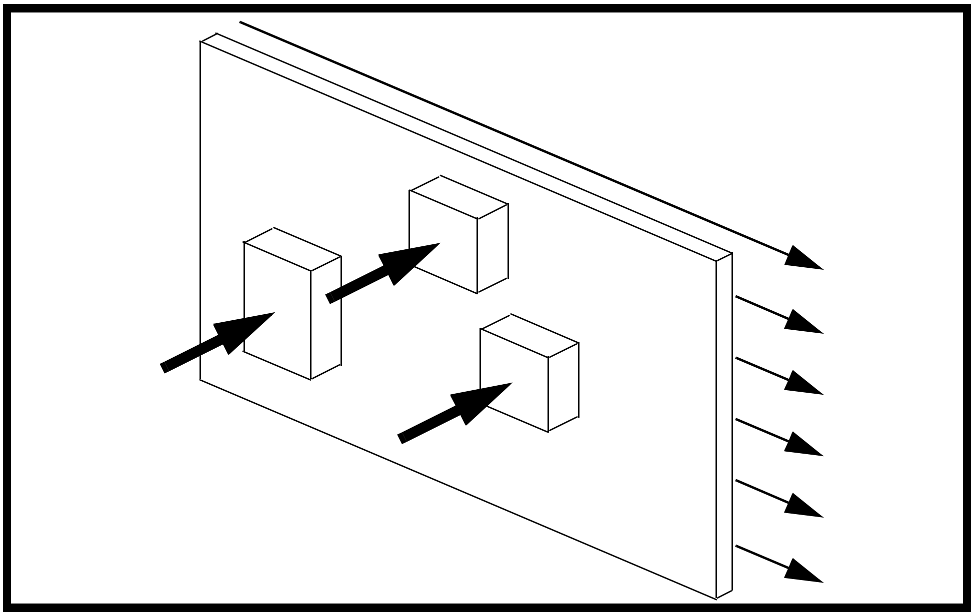
LESSON 7

Forced Convection on a Printed Circuit Board

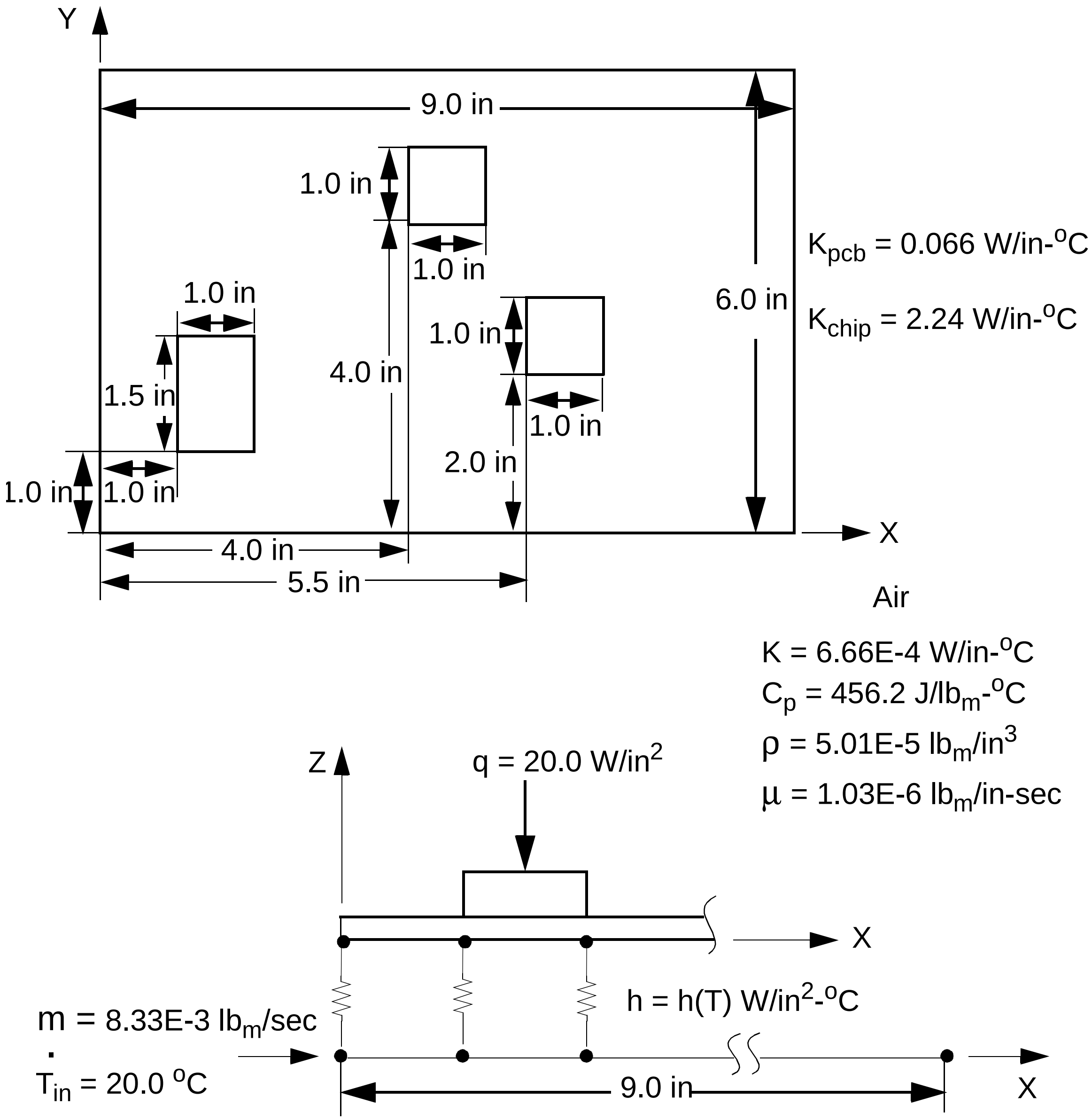


Objectives:

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

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 analyzye it to determine the steady-state temperature distribution.



Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 2.1 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V2.1.

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

Open Model File:	**New Model**

2. Turn off the Geometry Engine.

Tools/Advanced Geometry ● **Standard**

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:	**pcb**
Material:	**1..pcb**

Elem/Property Type...

Volume Elements: ● **Solid**

OK

OK

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

Title: **chip**

Material: **2..chip**

OK

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: **back**

Material: **1..pcb**

Elem/Property Type...

● **Membrane**

OK

Thickness: **.001**

OK

Cancel

5. Create the NASTRAN 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**
5.5	**3**	**0**	**OK**

Cancel

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

View/Autoscale... **<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.

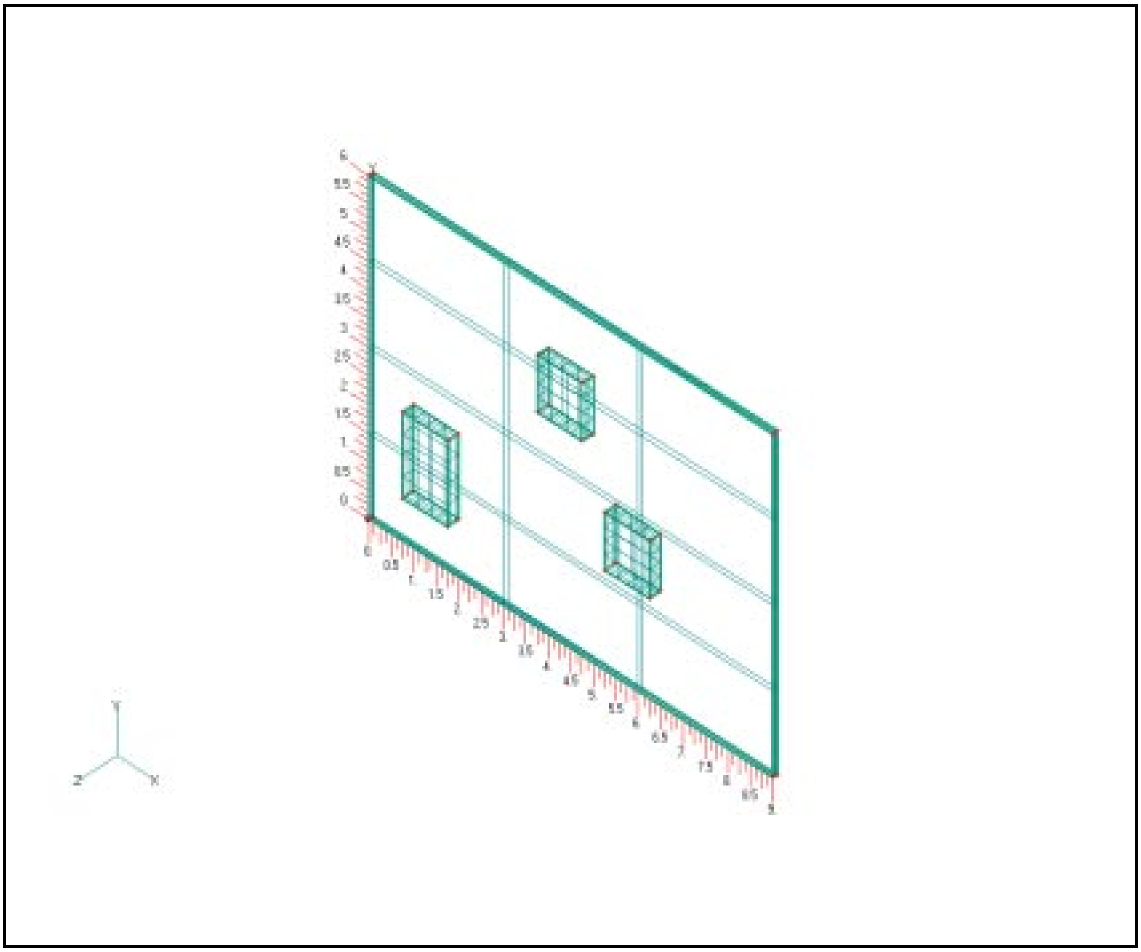
Change the viewing angle.

View/Rotate... **<F8>**

Isometric
OK

Your model should be like the following:

Figure 7-1: 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...

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:	**25**

OK

X:	*Y:*	*Z:*	
0	**0**	**-.1**	**OK**
9	**0**	**-.1**	**OK**
9	**6**	**-.1**	**OK**
0	**6**	**-.1**	**OK**

Create tube elements to help model the fluid flow.

Mesh/Between...

New Prop...

Title:	**flow_tube**
Material:	**1..pcb**

Elem/Property Type...

● **Tube**

OK

OK

Mesh Size / #Nodes / Dir 1:	**37**

OK

X:	*Y:*	*Z:*	
0	**3**	**-1**	**OK**
9	**3**	**-1**	**OK**

Finally, remove coincident nodes of the model.

Tools/Check/Coincident Nodes...

Select All

OK

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

No

☒ **Merge Coincident Entities**

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...

ID: **921**

to: **1784**

More

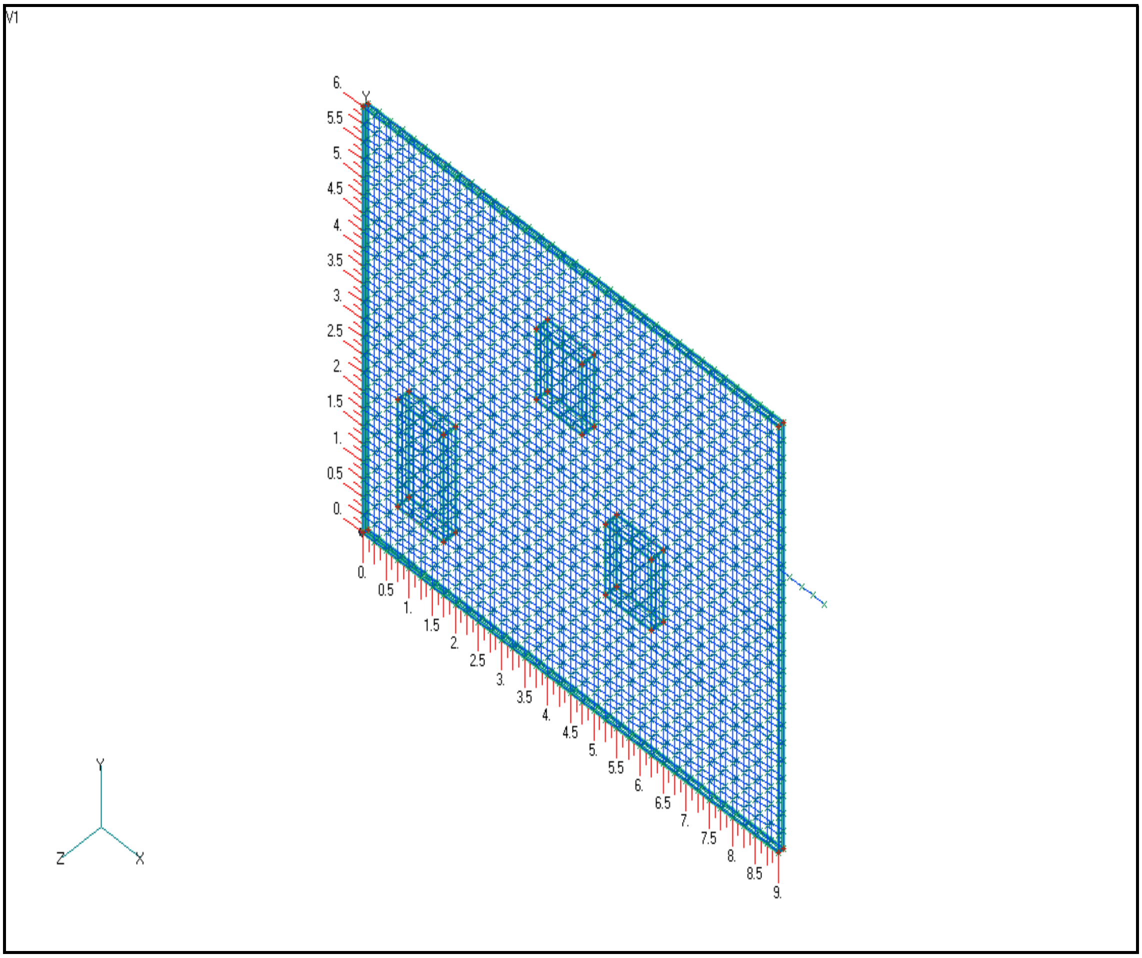
OK

● **Reverse Normals**

OK

Your model should be like the following:

Figure 7-2: Meshed model



9. 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:	**thermal**

OK

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:	**20**

Model/Load/Heat Transfer

Constant Coefficient:	**0.3**
Reynolds Exponent:	**0.0**
Prandtl Exponent (into fluid):	**0.0**
Prandtl Exponent (out of fluid):	**0.0**
Fluid Conductivity:	**6.66e-4**
Fluid Specific Heat:	**456.2**
Fluid Viscosity:	**1.03e-6**
Fluid Density:	**5.01e-5**

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: ● **Heat Flux**

Value: **20**

OK

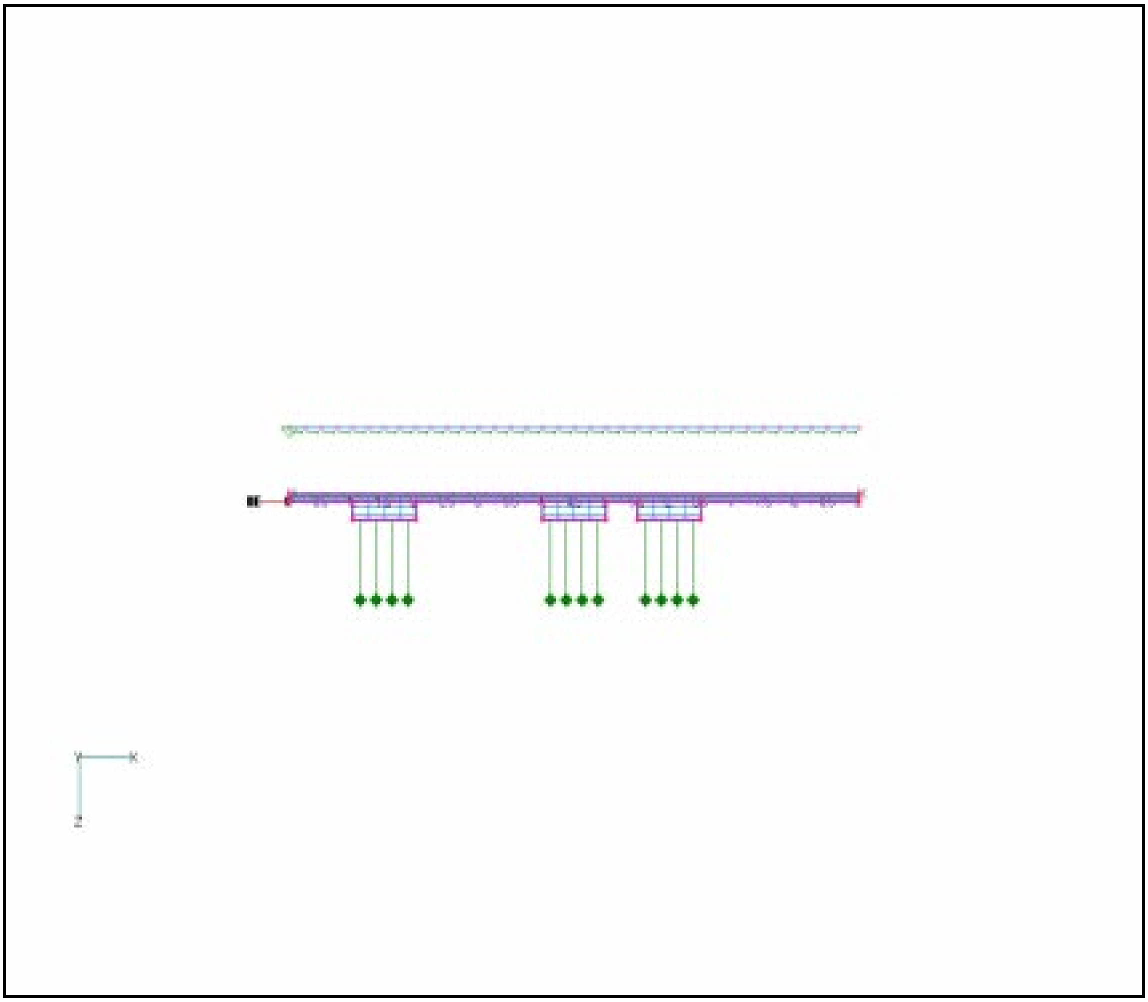
Face: **2**

OK

Cancel

Your model should be like the following:

Figure 7-3: Model with thermal load and heat flux



10. Create the inlet temperature for the fluid flow.

Model/Load/Nodal...

Select the node on the left 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...

ID: **921**

to: **1784**

More

OK

Type: ● **Convection**

☒ **Forced Convection**

☒ **Disable Advection**

Flow Rate: **.00833**

Diameter: **1.0**

Area Factor: **3**

OK

Hold shift and drag a box around the tube elements.

OK

Type: ● **Convection**

☒ **Forced Convection**

☒ **Disable Convection**

Flow Rate:	**.008333**
Diameter:	**1.0**
Temperature:	**20**

OK

Cancel

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

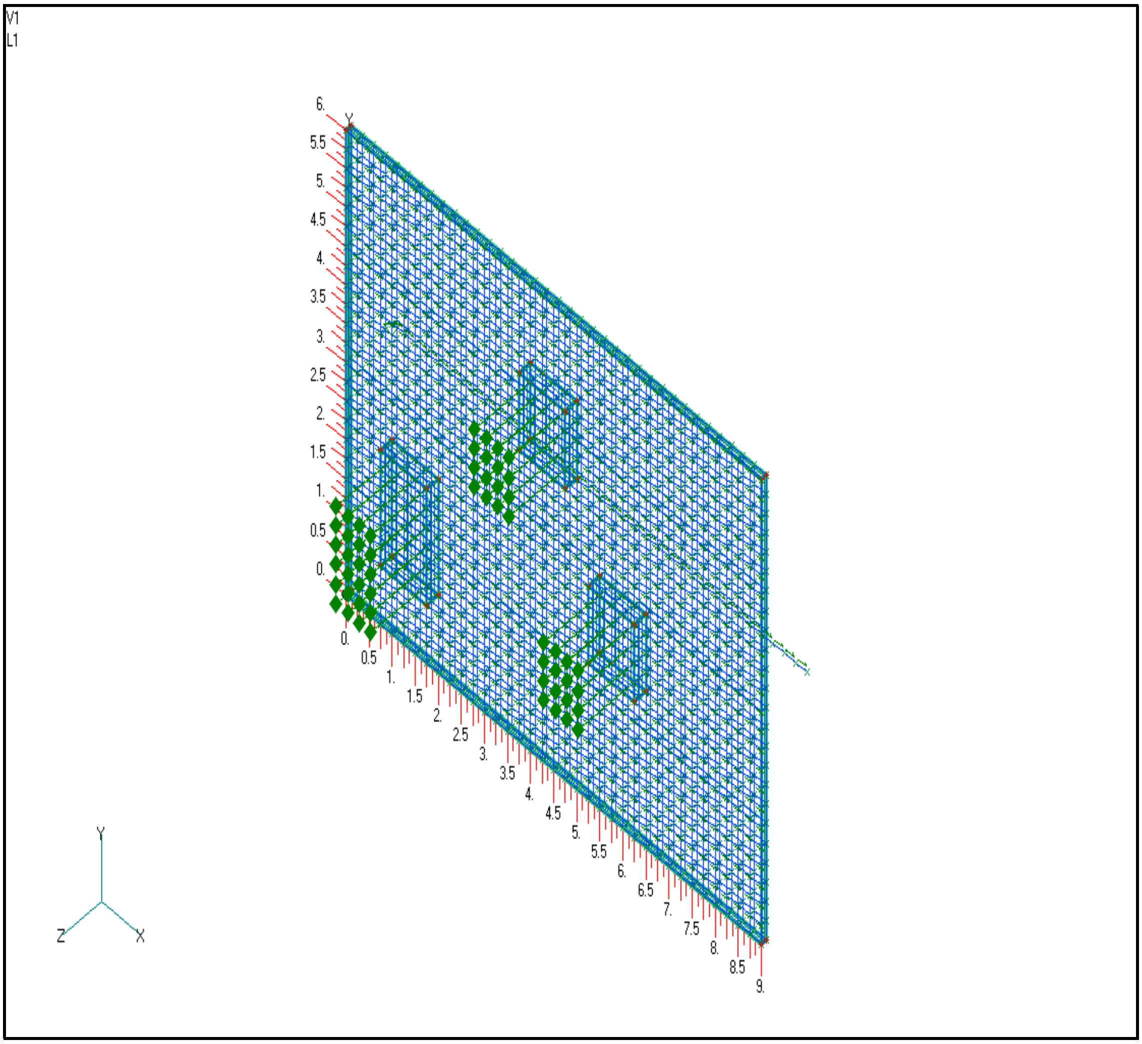
Change the viewing angle.

View/Rotate... **<F8>**

Isometric
OK

Your model should be like the following:

Figure 7-4: The loaded model.



12. 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...

ID:	**921**
to:	**1784**
More	
OK	
	● **Align First Edge to Vector**
OK	
Method^	**(**select **Nodes)**
Base Node ID:	**37**
Tip Node ID:	**925**
OK	

13. 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 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

When asked if it is “OK to Begin Reading File C:\pcb2.xdb”, respond **Yes**.

Yes

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

View/Options...

Quick Options...

Geometry Off

☐ **Load - Thermal**

☐ **Load - Heat Flux**

☐ **Load - Convection**

Done

OK

15. Create a final temperature distribution contour plot.

View/Select... **<F5>**

Model Style: ● **Quick Hidden Line**

Contour Style: ● **Contour**

Deformed and Contour Data...

Deformation: **31..Temperature**

Contour: **31..Temperature**

OK

OK

In **Figure 7-5**, notice the temperature gradients around the chips, where all the heat is produced.

When done, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Figure 7-5: Thermal analysis of a printed circuit board with forced convection.