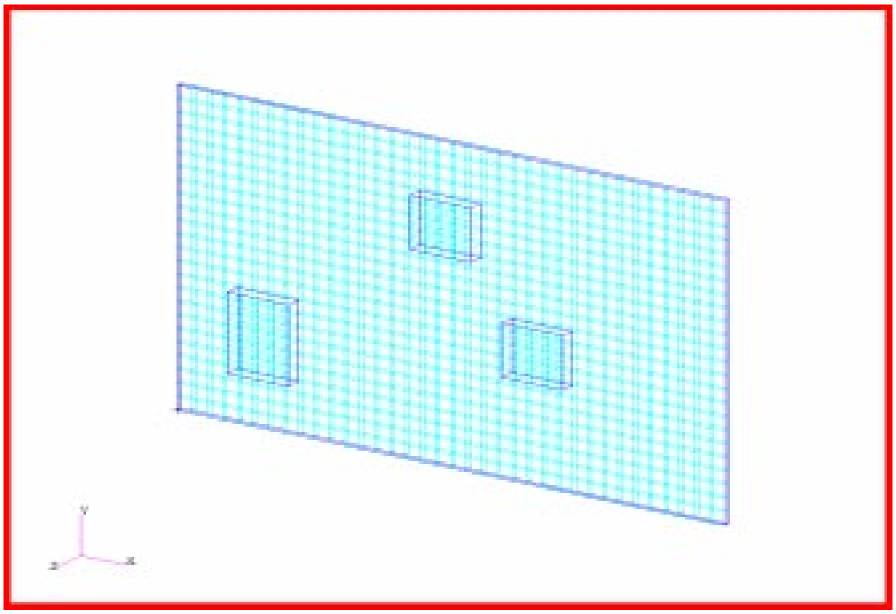
WORKSHOP 2

Free Convection on Printed Circuit Board



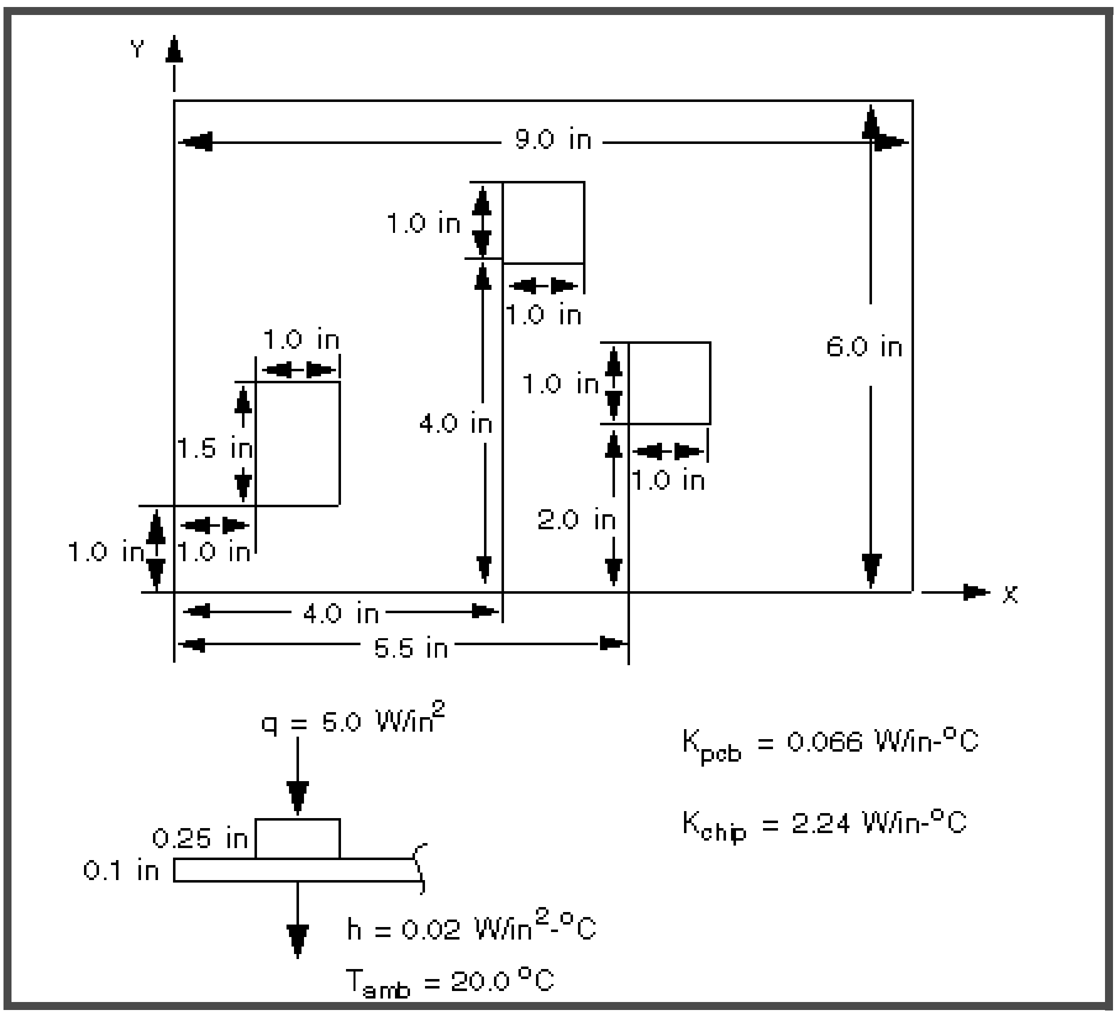
Objectives:

- Create surfaces for PCB and electronic devices.
- Apply thermal loads and boundary conditions.
- Perform a steady-state analysis.

Model Description:

Shown below depicts a printed circuit board (PCB assembly which has three significant ship devices mounted on it. Each chip is generating heat at a rate that is consistent with the application of a heat flux of 5.0 W/in2 over each device surface area. Heat is dissipated by thermal conduction within the chips and underlying board. Free convection to the ambient environment provides the ultimate heat take. The ambient temperature for convection is assumed to be 20.0 ^{o}C, and a heat transfer coefficient of 0.02 W/in2-oC is used to apply convection to the entire assembly surface. We will analyze the printed circuit board to determine the device temperature so that they can be compared to manufacturer allowables.

Shown below is a drawing of the model you will be building and suggested steps for its construction.



Suggested Exercise Steps:

- 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. Create a **New Database** and name it **free_conv_pcb.db**.

File/New...

New Database Name	**free_conv_pcb.db**

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC.Nastran** in the *New Model Preferences* form. Verify that the *Analysis Type* is **Structural**.

New Model Preference

Tolerance	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type	**Thermal**

OK

3. Create the surfaces of printed circuit board and electronic components..

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Surface ID List:	**1**
Vector Coordinates List:	**<9 6 0 >**
Origin Coordinates List:	**[0 0 0]**

Apply

For Chip 1

Surface ID List:	**2**
Vector Coordinates List:	**<1 1.5 0>**
Origin Coordinates List:	**[1 1 0]**

Apply

.For Chip 2

Surface ID List:	**3**
Vector Coordinates List:	**<1 1 0>**
Origin Coordinates List:	**[4 4 0]**

Apply

For Chip 3

Surface ID List:	**4**
Vector Coordinates List:	**<1 1 0>**
Origin Coordinates List:	**[5.5 2 0]**

Apply

4. Create the PCB solid by extruding surfaces 1 by -0.1 inch in the Z direction. Extrude surfaces.2,3 and 4 in the Z direction by 0.25 inches.

Geometry

Action:	**Create**
Object:	**Solid**
Method:	**Extrude**
Solid ID List:	**1**
Translation Vector:	**<0 0 -0.1>**

If the **Auto Execute** is *ON*, you do not need to click on **Apply**.

Surface List:	**Surface 1**

Apply

For Chips 1, 2, 3:

Solid ID List:	**2**
Translation Vector:	**<0 0 0.25>**

Surface List:	**Surface 2:4**

Apply

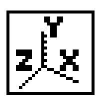
5. You will now create the model's finite elements.

◆ **Finite Elements**

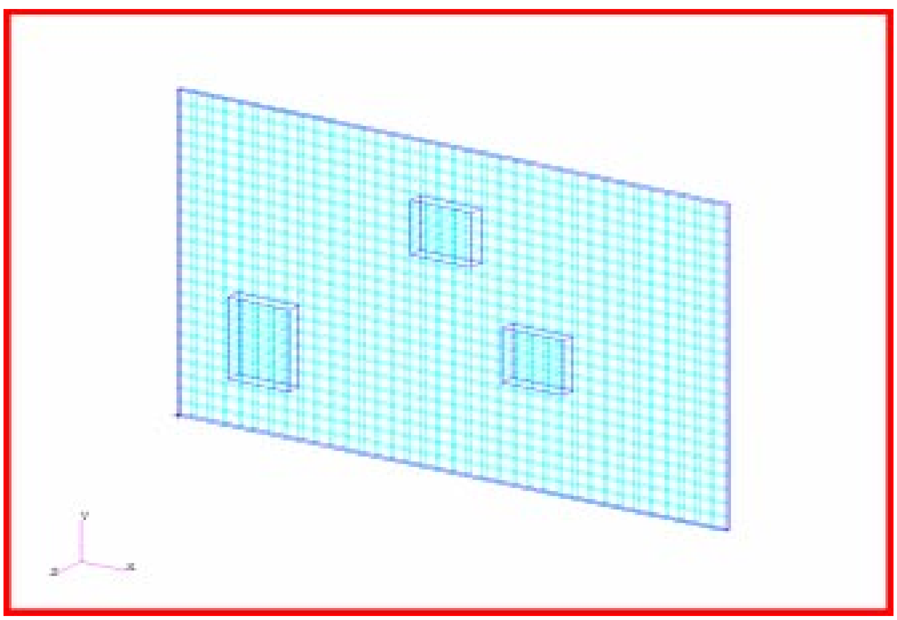
Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length:	**0.25**
Element Topology:	**Hex8**
Solid List:	**Solid 1:4**

Apply

To obtain a clearer view, select the isometric view by clicking on the *Iso View icon.*



The mesh should look like this.



6. For this model we will assume that the PCB and chips are manufactured from the isotropic materials having constant conductivities.

Kpcb = 0.066 W/in-oCKchip = 2.24 W/in-oC

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**pcb**
Input Properties....	
Thermal Conductivitiy	**0.066**
Apply	

For chips 1, 2, 3:

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**chip**
Thermal Conductivity:	**2.24**
Apply	

7. For a solid model element properties are used to assign the materials to the various parts of the model. .

◆ **Properties**

Action:	**Create**
Dimension:	**3D**
Type:	**Solid**
Property Set Name:	**pcb**
Input Properties...	

Material Name:	**m:pcb**
OK	
Select Members:	**Solid 1**
ADD	
Apply	

Chips 1, 2, 3

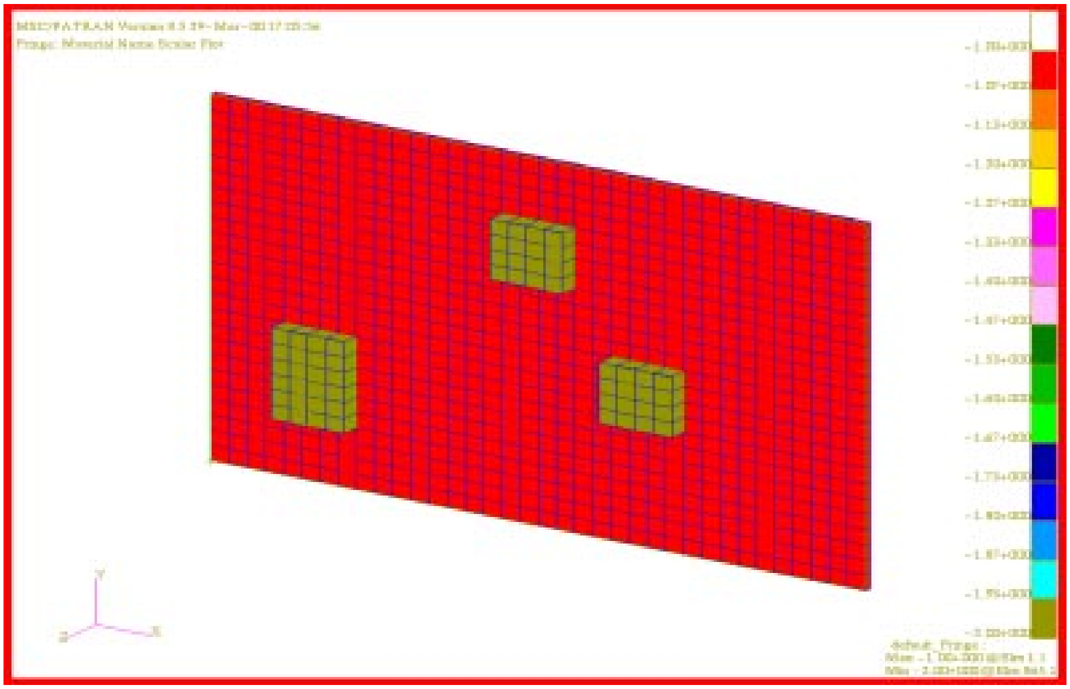
Input Properties...	
Material Name:	**m:chip**
OK	
Select Members:	**Solid 2:4**
ADD	
Apply	

8. To verify that the correct material properties have been defined and assigned to the correct model locations, change the **Action** option to **Show** and create a scalar plot of the model's materials.

◆ **Properties**

Action:	**Show**
Select Property:	**Material Name**
Display Method	**Scalar Plot**
Select Groups:	◆**Current Viewport**
	Default Group
Apply	

The scalar plot resembles the following.



9. The duplicate nodes located at the PCB and chip interfaces must be merges.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**
Equivalence Tolerance:	**0.005**

Apply

10. To check the equivalence process you should verify the element boundaries. If the model has been equivalenced properly you should see a wireframe rendering of your model where only the free edges are components of the wireframe image. Display the view to ensure that the model has no cracks between elements.

◆ **Finite Elements**

Action:	**Verify**
Object:	**Element**
Test:	**Boundaries**

Display Type: ◆**Free Edges**

Apply

11. A heat flux will now be applied to the exposed plan from face of the chips.

◆ **Load/BCs**

Action: **Create**

Object: **Applied Heat**

Type: **Element Uniform**

Option: **Normal Fluxes**

New Set Name: **flux**

Target Element Type: **3D**

Input Data...

Heat Flux: **5**

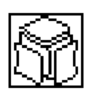
OK

Select Application Region...

Geometry Filter: ◆**Geometry**

Select Solid Faces: **Solid 2.6 3.6 4.6**

Use the **Free Face Select** icon to help you pick the exposed chip faces.



Add

OK

Apply

12. The convection boundary condition will now be applied to the back side of the PCB (side opposite the chips)..

◆ **Load/BCs**

Action: **Create**

Object: **Convection**

Type: **Element Uniform**

Option: **To Ambient**

New Set Name: **conv**

Target Element Type: **3D**

Input Data...

Convection Coefficient: **0.02**

Ambient Temperature: **20**

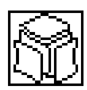
OK

Select Application Region

Geometry Filter: ◆**Geometry**

Select Solid Faces: **Solid 1.6**

Use the **Free Face Select** icon to help you pick the back face of the PCB.



Add

OK

Apply

13. Perform the Analysis.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name: **ex2**

Solution Type...

Solution Type: ◆ **STEADY STATE ANALYSIS**

OK

Apply

An MSC.Nastran input file called **ex2.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:

14. Submit the input file to MSC.Nastran for analysis.

    14a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex2.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    14b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex2 scr=yes**. Monitor the run using the UNIX **ps** command.

15. When the run is completed, edit the **ex2.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.

16. MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.

17. Proceed with the Reverse Translation process, that is, attaching the **fin.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:.

◆ **Analysis**

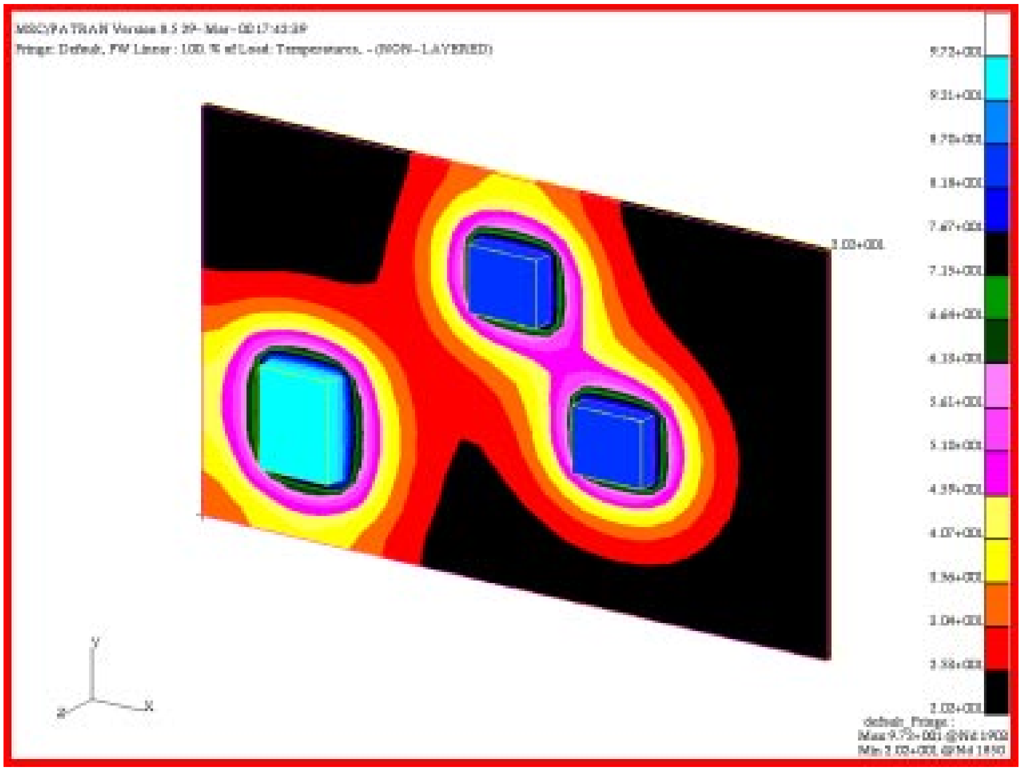
Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Result Files....	
Select Results File	**ex2.xdb**
OK	
Apply	

18. Display the results.

◆ **Results**

Object:	**Quick Plot**
Select Results Cases:	**Default, PW Linear: 100%..**
Select Fringe Result:	**Temperature**
Apply	

The result should resemble the following.



The heat generated by the electronic devices is conducted to the printed circuit board, and then spread on the epoxy glass PCB. The cooling mechanism is provided by a free convection heat exchange between the backside of the PCB and the ambient fluid that is maintained at 20 oC. As a result, the largest electronic device has the highest temperature. Because of their indentical size, the other two electronic chips possess nearly the same temperature distribution.