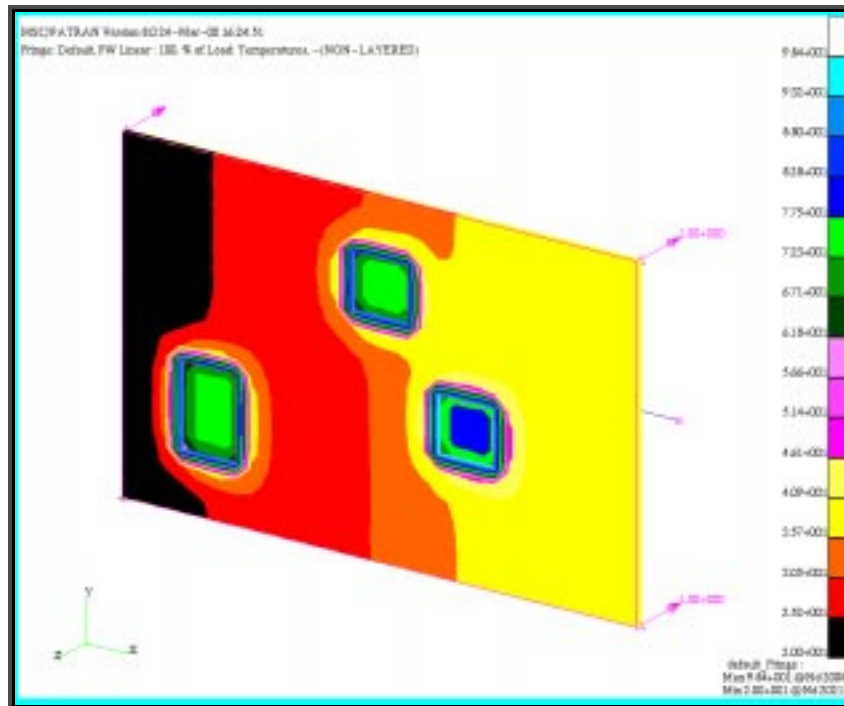


WORKSHOP 3

Forced Air Convection on Printed Circuit Board



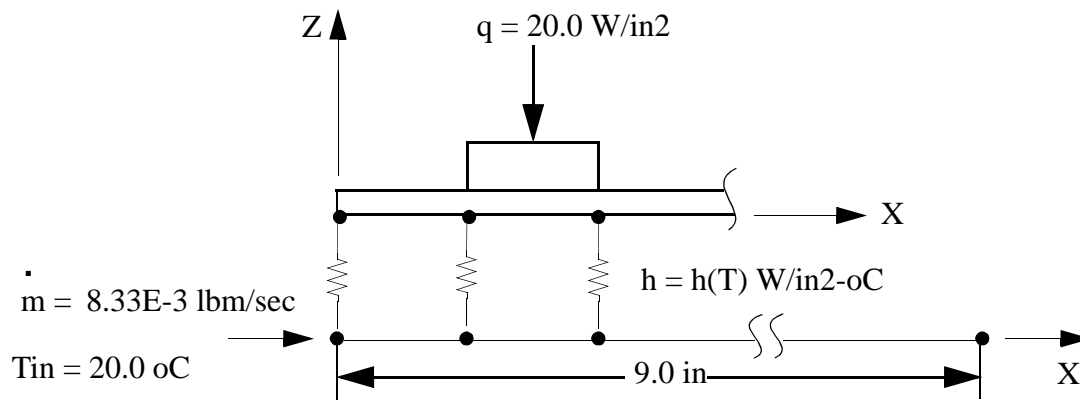
Objective:

- Create Geometry from MSC.Patran
-



Model Description:

We will model the previous PCB thermal analysis with forced air convection over the flat plate, using the Coupled Advection feature. The air temperature rises in the X direction as the fluid stream traverses the circuit board. The temperature dependency of the convection coefficient will be defined using a temperature dependent field



Air

$$K = 6.66E-4 \text{ W/in-oC}$$

$$C_p = 456.2 \text{ J/lbm-oC}$$

$$\rho = 5.01E-5 \text{ lbm/in}^3$$

$$\mu = 1.03E-6 \text{ lbm/in-sec}$$



Suggested Exercise Steps:

- Create a new database called **forced_conv_pcb.db**
- Create a solid that represents the electronic component and the printed circuit board.
- Mesh surfaces and curves with global edge length of 0.25 using Hex8 as element topology
- Merge nodes by using Equivalence method under Finite Elements.
- Use Free Edges to verify the Element Boundaries
- Input specify Material Properties for the **chip, pcb, air, and flow tube**.
- Define the solids' properties using **pcb** and **chip** for their property names.
- Apply loads and boundary conditions to the model using Coupled Advection Feature to simulate the forced air convection on the back surface of PCB.
- Apply heat flux on each device using Element uniform and define the inlet Temperature of the fluid.
- Perform, read, and display the results



Exercise Procedure:

1. Create a **New Database** and name it **forced_conv_pcb.db**.

File/New...

New Database Name

forced_conv_pcb

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 **Thermal**.

New Model Preference

Tolerance

◆ **Default**

Analysis Code:

MSC/NASTRAN

Analysis Type

thermal

OK

3. Create the surfaces representing the printed circuit board.

◆ **Geometry**

Action:

Create

Object:

Surface

Method:

XYZ

Vector Coordinates List:

<9 6 0>

Origin Coordinates List:

[0 0 0]

Apply

Vector Coordinates List:

<1 1.5 0>

Origin Coordinates List:

[1 1 0]

Apply

Vector Coordinates List:

<1 1 0>

Origin Coordinates List:

[4 4 0]

Apply

Vector Coordinates List:

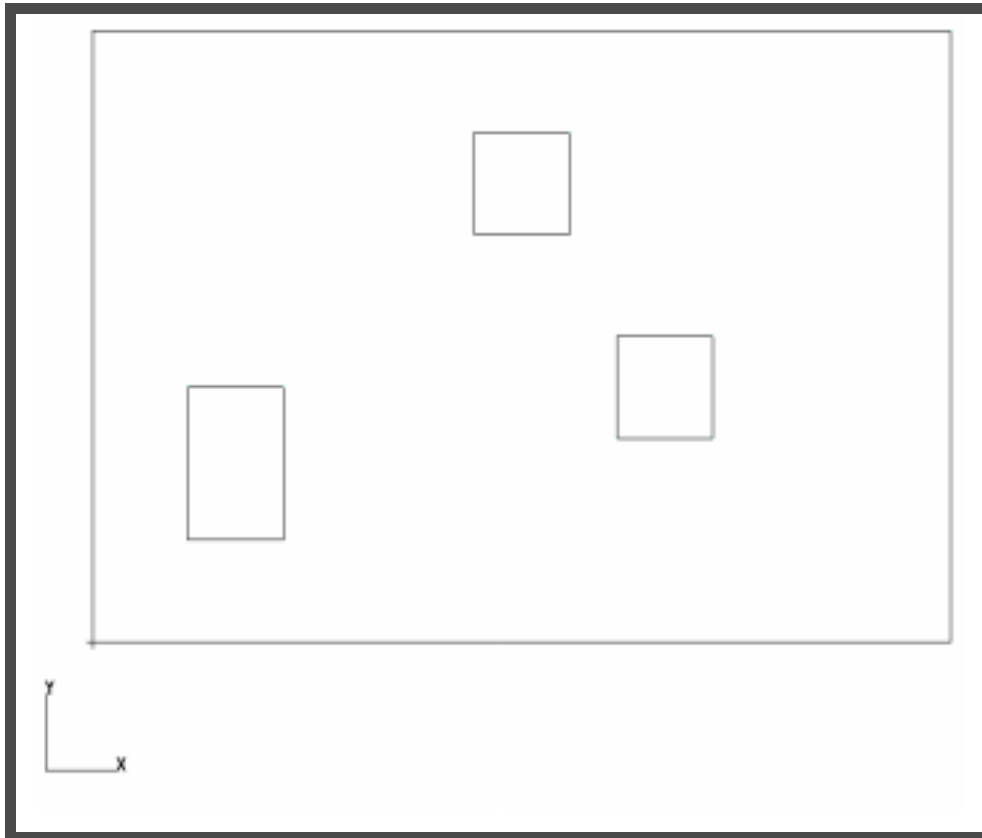
<1 1 0>

Origin Coordinates List:

<5.5 2 0>

Apply

When you are finished your model should look like the one shown in the figure below.



4. Extrude the solid

◆ **Geometry**

Action:

Create

Object:

Solid

Method:

Extrude

Translation Vector:

<0 0 -0.1>

Surface List:

Surface 1

Apply

Translation Vector:

<0 0 0.25>

Surface List:

Surface 2:4

Apply

5. Mesh the solids to create Hex8 element with global edge length 0.25.

◆ Finite Elements

Action:

Create

Object:

Mesh

Type:

Solid

Global Edge Length

0.25

Element Topology

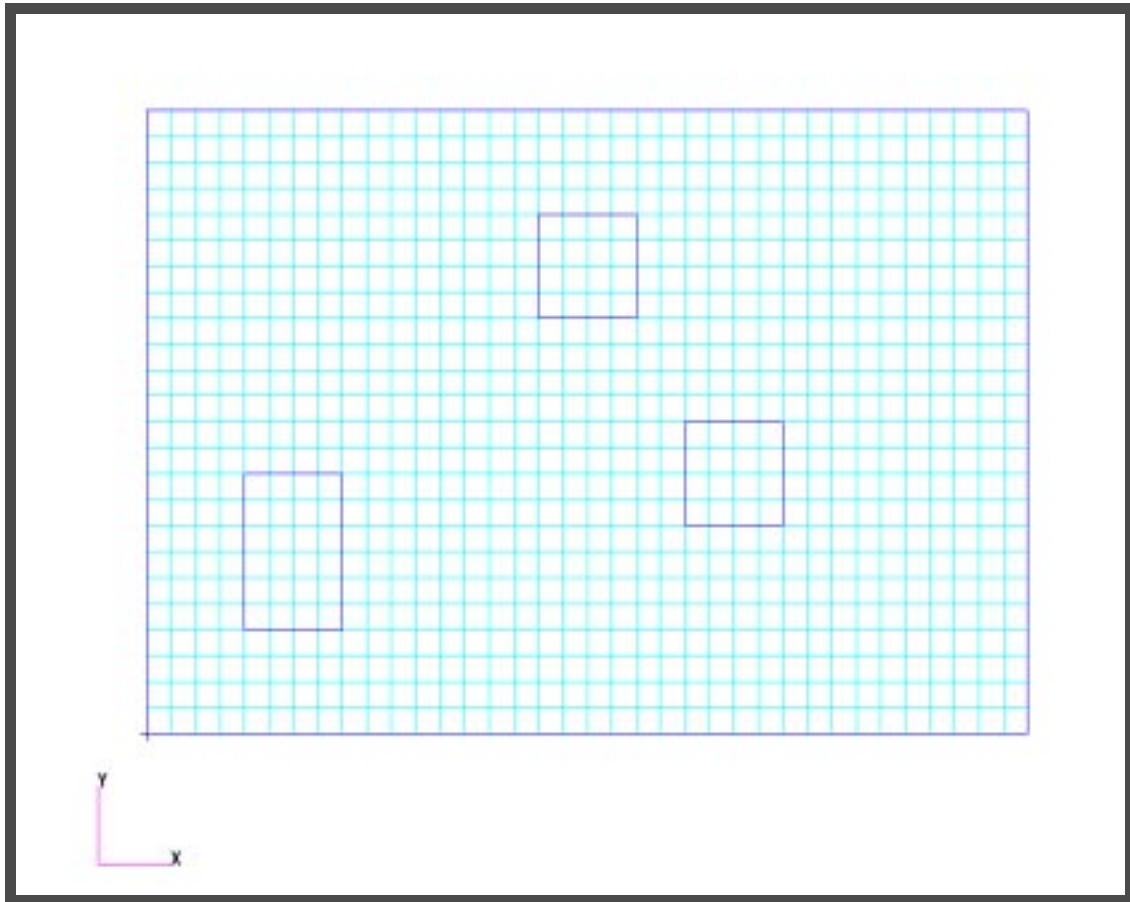
Hex8

Solid List

Solid 1:4

Apply

Your model should appear like the one shown below.



6. Equivalence the Finite Elements to reduce the number of elements by eliminating duplicate nodes.

◆ **Finite Elements**

Action:

Equivalence

Object:

All

Type:

Tolerance Cube

Equivalence Tolerance:

0.005

Apply

7. Verify the Element Boundaries.

◆ **Finite Elements**

<i>Action:</i>	<input type="text" value="Verify"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Test:</i>	<input type="text" value="Boundaries"/>
<i>Display Type:</i>	◆ <input type="text" value="Free Edges"/>
<input type="text" value="Apply"/>	

8. Create the isotropic material properties.

◆ **Materials**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="chip"/>
<input type="text" value="Input Properties..."/>	
<i>Constitutive Model:</i>	<input type="text" value="Solid properties"/>
<i>Thermal Conductivity:</i>	<input type="text" value="2.24"/>
<input type="text" value="Apply"/>	

<i>Material Name:</i>	<input type="text" value="pcb"/>
<i>Constitutive Model:</i>	<input type="text" value="Solid properties"/>
<i>Thermal Conductivity:</i>	<input type="text" value="0.066"/>
<input type="text" value="Apply"/>	

9. Create the model's element properties assigning the material type to the correct region of the model.

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Dimension:</i>	<input type="text" value="3D"/>
<i>Type:</i>	<input type="text" value="Solid"/>
<i>Property Set Name:</i>	<input type="text" value="chip"/>

Input Properties...

Material Name:

m:chip

OK

Select Members:

Solid 2:4

Add

Apply

Property Set Name:

pcb

Input Properties...

Material Names:

m:pcb

OK

Select Members:

Solid 1

Add

OK

10. Define Temperature Dependent Field.

◆ **Fields**

Action:

Create

Object:

Material Property

Method:

Tabular Input

Field Name:

conv_temp

Active Independent Variables:

Temperature (T)

Input Data....

Input Scalar Data:

Hit Enter Key

T.Value

00.2

1000.3

2000.35

Add

OK

11. Select two nodes to create a curve.

◆ Geometry

<i>Action:</i>	Create
<i>Object:</i>	Curve
<i>Method:</i>	Point



select the Node Icon

<i>Starting Point List:</i>	←	Node 938
<i>Ending Point List:</i>		Node 1838

Apply

12. Define the location of the Air Stream.

◆ Geometry

<i>Action:</i>	Transform
<i>Object:</i>	Curve
<i>Method:</i>	Translate
<i>Translation Vector:</i>	<0 0 -1.0>
<i>Curve List:</i>	Curve 1

Apply

13. Mesh the Airstream. preferably, the mesh size should be the same on the air stream as on the PCB.

◆ Finite Element

<i>Action:</i>	Create
<i>Object:</i>	Mesh
<i>Type:</i>	Curve

<i>Global Edge Length:</i>	0.25
<i>Ending Point List:</i>	Bar2
<i>Curve List:</i>	Curve 2
Apply	

Note: The identical mesh size is not required, but may provide the most accurate model. The Closest Approach method will select the nearest neighboring structure and fluid nodes.

14. Specify the material properties of air.

◆ Materials

<i>Action:</i>	Create
<i>Object:</i>	Isotropic
<i>Method:</i>	Manual Input
<i>Material Name:</i>	air
Input Properties...	
<i>Constitutive Model:</i>	Fluid properties
<i>Thermal Conductivity:</i>	6.66e-4
<i>Specific Heat:</i>	456.2
<i>Density:</i>	5.01e-5
<i>Dynamic Viscosity:</i>	1.03e-3
Apply	

15. Define flow tube properties.

◆ Properties

<i>Action:</i>	Create
<i>Dimension:</i>	1D
<i>Type:</i>	Flow Tube
<i>Property Set Name:</i>	flow_tube
Input Properties...	
<i>Material Name:</i>	m:air

Diameter at Node 1:

OK

Select Members

16. We will use the Coupled Advection Feature to simulate the forced air convection on the back surface of PCB.

◆ **Load/BCs**

Action:

Object:

Type:

Options:

There are two application regions:

- The Structure Region (Application Region 1) can be 1D, 2D, or 3D. In this case we have a 3D structure, and the appropriate Target Element Type is 3D.
- The Second Application Region must be 1D, which represents the air-flow over the flat plate. In this case, select the curve along the X direction. MSC.PATRAN will then couple the fluid to the structure locally by the Closest Approach method.

New Set Name:

Target Element Type:

Region 2:

Temperature Function

Mass Flow Rate:

Geometry Filter:

Select Solid Surfaces:

■ Active List

Add

Select Curves:

Add

OK

Apply

◆ Geometry

Solid 1.6

for companion region

Curve 2

17. Apply a Heat Flux on Each Device.

◆ Load/BCs

Action:

Create

Object:

Applied Heat

Type:

Element Uniform

Options:

Normal Flux

New Set Name:

heat_flux

Target Element Type:

3D

Input Data...

Heat Flux:

20

OK

Select Application Region

Geometry Filter:

◆ Geometry

Select Solid Surfaces

Solid 2.6 3.6 4.6

Add

OK

Apply

18. Define the inlet Temperature of the fluid.

◆ **Load/BCs**

Action:

Object:

Type:

New Set Name:

Boundary Temperature:

Geometry Filter:

◆ **Geometry**

Select Geometry Entities:

19. Define the Default Initial Temperature and Perform the analysis.

◆ **Analysis**

Action:

Object:

Method:

Job Name:

Data Deck Echo:

Default Init Temperature:

Apply

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

20. Submit the input file to MSC.Nastran for analysis.
 - 20a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex3.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 20b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex3 scr=yes**. Monitor the run using the UNIX **ps** command.
21. When the run is completed, edit the **ex3.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.

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

23. Proceed with the Reverse Translation process, that is, attaching the ex3.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

ex3.xdb

OK

Apply

24. Display the Results.

◆ **Results**

Object:

Quick Plot

Select Results Cases:

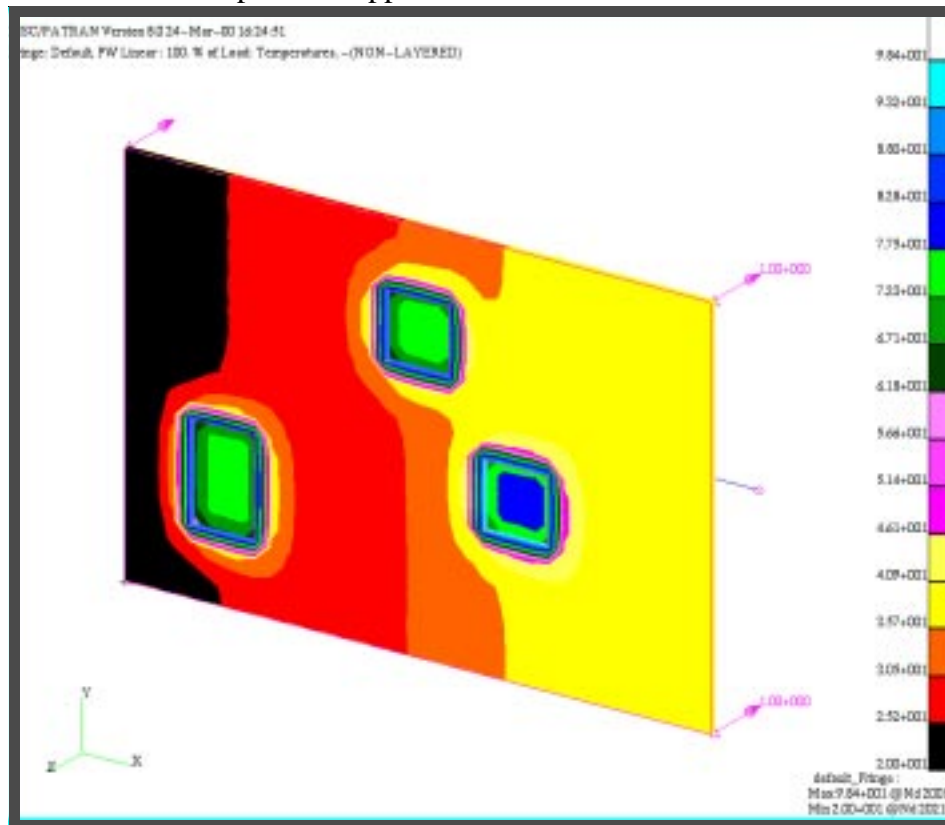
**Default, PW Linear:
100. % of Load**

Select Fringe Result:

Temperatures

Apply

Your Viewport will appear as follows.



The viewport may now be reset by clicking on the broom icon in the main window.



File/Quit...

