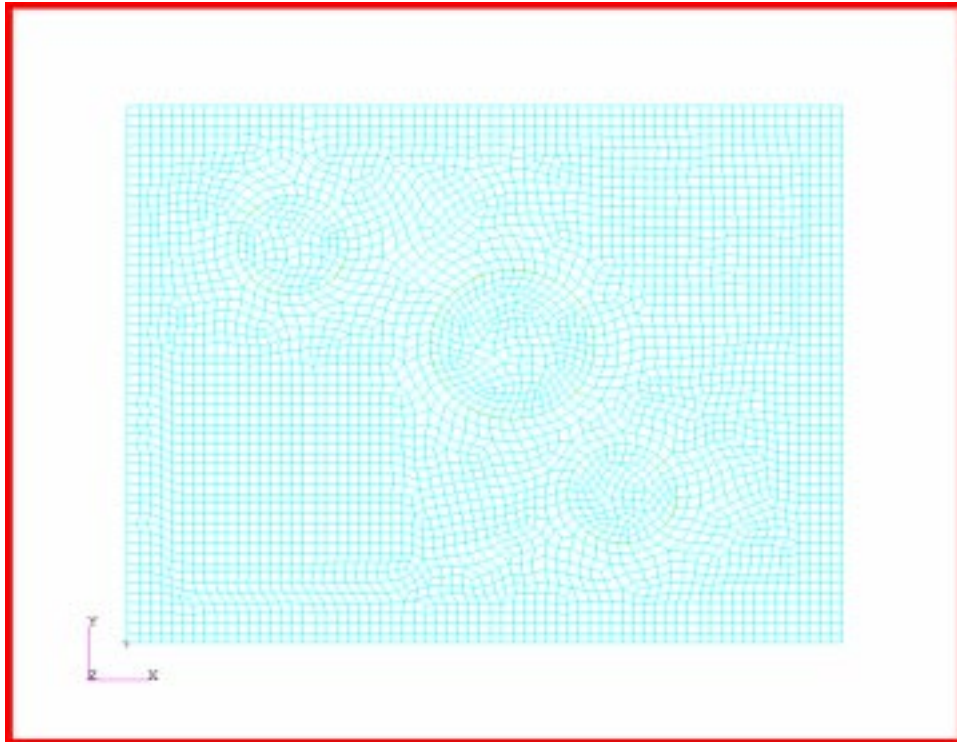


---

## APPENDIX C

---

# *Printed Circuit Board Using 2 1/2 D Paved meshing method*



### **Objective:**

- Create Geometry from MSC.Patran



## Suggested Exercise Steps:

- Create a new database called **pcb.db**



## Exercise Procedure:

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

**File/New...**

*New Database Name*

**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**

Whenever possible click  **Auto Execute** (turn off).

3. Create the surfaces representing the printed circuit board.

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**XYZ**

*Vector Coordinates List:*

**<6.5 5.4 0>**

*Origin Coordinates List:*

**[ 0 0 0 ]**

**Apply**

Repeat the previous step to create the two rectangular components.

*Vector Coordinates List:*

**<0.75 0.875 0>**

*Origin Coordinates List:*

**[5 3.5 0]**

**Apply**

---

*Vector Coordinates List:*   
*Origin Coordinates List:*

Use the **Curve** object to create the two circular components

◆ **Geometry**

*Action:*   
*Object:*   
*Method:*   
*Input Radius*   
*Construction Plane List*   
*Center Point List*

*Input Radius*   
*Construction Plane List*   
*Center Point List*

*Input Radius*   
*Construction Plane List*   
*Center Point List*

4. Create trimmed surfaces that will represent the components and the circuit boards.

In order to create trimmed surfaces, the curves that represents the edges of the surface must be chained together.

◆ **Geometry**

*Action:*   
*Object:*

*Method:*

Chain

Delete Original Elements

*Curve List*

Surface 1.4 1.3 1.2 1.1

Apply

*Curve List*

Surface 2.4 2.3 2.2 2.1

Apply

*Curve List*

Surface 3.4 3.3 3.2 3.1

Apply

Generate the trimmed surfaces using the chained curved created in the previous steps.

◆ **Geometry**

*Action:*

Create

*Object:*

Surface

*Method:*

Trimmed

*Option:*

Planar

Use All Edge Vertices

Delete Outer Loop

*Outer Loop List*

Curve 4

Delete Inner Loops

*Inner Loop List*

Curve 2 1 6 3 5

Apply

Click NO when the message “Do you wish to delete the original curves?” appears. This also applies to the following steps.

◆ **Geometry**

---

*Action:*   
*Object:*   
*Method:*   
*Option:*

Use All Edge Vertices

Delete Outer Loop

*Outer Loop List*

Delete Inner Loops

Be sure to delete the curve lists in the **Inner Loop List** box

*Inner Loop List*

◆ **Geometry**

*Action:*   
*Object:*   
*Method:*   
*Option:*

Use All Edge Vertices

Delete Outer Loop

*Outer Loop List*

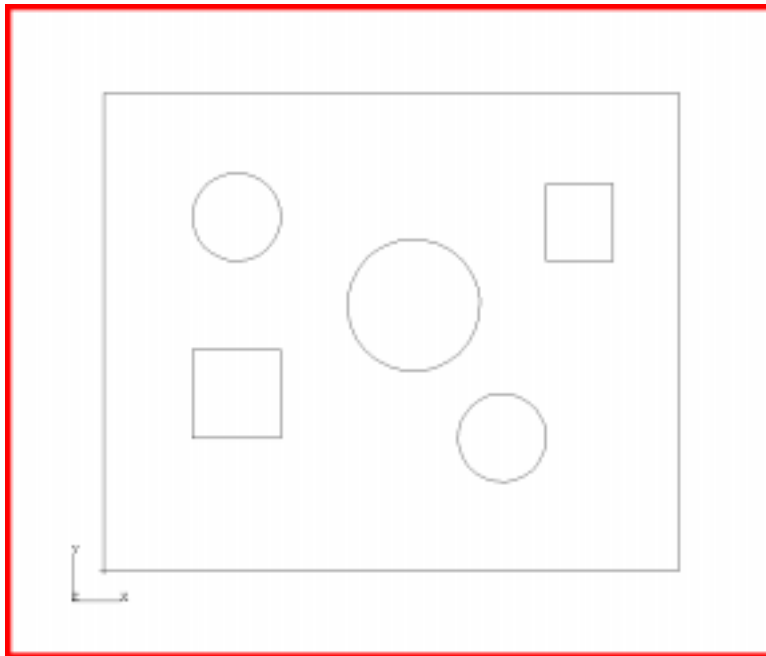
Delete Inner Loops

*Inner Loop List*

◆ **Geometry**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Surface"/>
<i>Method:</i>	<input type="text" value="Trimmed"/>
<i>Option:</i>	<input type="text" value="Planar"/>
	<input checked="" type="checkbox"/> Use All Edge Vertices
	<input checked="" type="checkbox"/> Delete Outer Loop
<i>Outer Loop List</i>	<input type="text" value="Curve 3"/>
	<input checked="" type="checkbox"/> Delete Inner Loops
<i>Inner Loop List</i>	<input type="text"/>
	<input type="text" value="Apply"/>

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



5. Create the finite element model

Because the surfaces are trimmed and non-parametric surfaces, paver meshed is need to generate the finite elements.

◆ **Finite Element**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Method:</i>	<input type="text" value="Surface"/>
<i>Global Edge Length</i>	<input type="text" value="0.1"/>
<i>Element Topology</i>	<input type="text" value="Quad4"/>
<i>Mesher</i>	◆ <b>Paver</b>
<i>Curve List</i>	<input type="text" value="Surface 4"/>
<input type="text" value="Apply"/>	

Create the finite elements for the components.

◆ **Finite Element**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Method:</i>	<input type="text" value="Surface"/>
<i>Global Edge Length</i>	<input type="text" value="0.1"/>
<i>Element Topology</i>	<input type="text" value="Quad4"/>
<i>Mesher</i>	◆ <b>Paver</b>
<i>Curve List</i>	<input type="text" value="Surface 5 6 3 7 2"/>
<input type="text" value="Apply"/>	

Extrude the Quad4 elements to generate solid elements to represent the epoxy layer.

◆ **Finite Element**

<i>Action:</i>	<input type="text" value="Sweep"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Method:</i>	<input type="text" value="Extrude"/>
<i>Direction Vector</i>	<input type="text" value="&lt;0 0 0.010&gt;"/>

*Extrude Distance*

*Offset*

Delete Orginial Elements

*Base Entity List*

**Mesh Control**

*Number =*

**OK**

**Apply**

Repeat the previous step and create the solid elements on top of the epoxy elements to represent the components, and the circuit board.

◆ **Finite Element**

*Extrude Distance*

*Offset*

Delete Orginial Elements

*Base Entity List*

**Mesh Control**

*Number =*

**OK**

**Apply**

*Direction Vector*

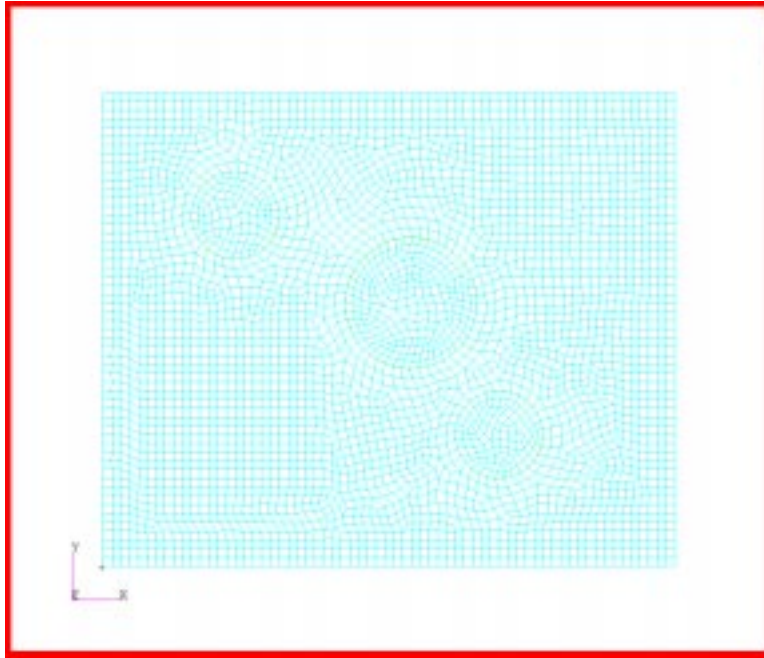
*Extrude Distance*

*Offset*

Delete Orginial Elements

*Base Entity List*

**Apply**



Equivalence the model to make sure all the intersecting nodes are connected.

◆ **Finite Element**

*Action:*

**Equivalence**

*Object:*

**All**

*Method:*

**Tolerance Cube**

*Equivalencing Tolerance*

**0.0045**

**Apply**

Use the **Verify** command to make sure that the model is meshed correctly.

◆ **Finite Element**

*Action:*

**Verify**

*Object:*

**Element**

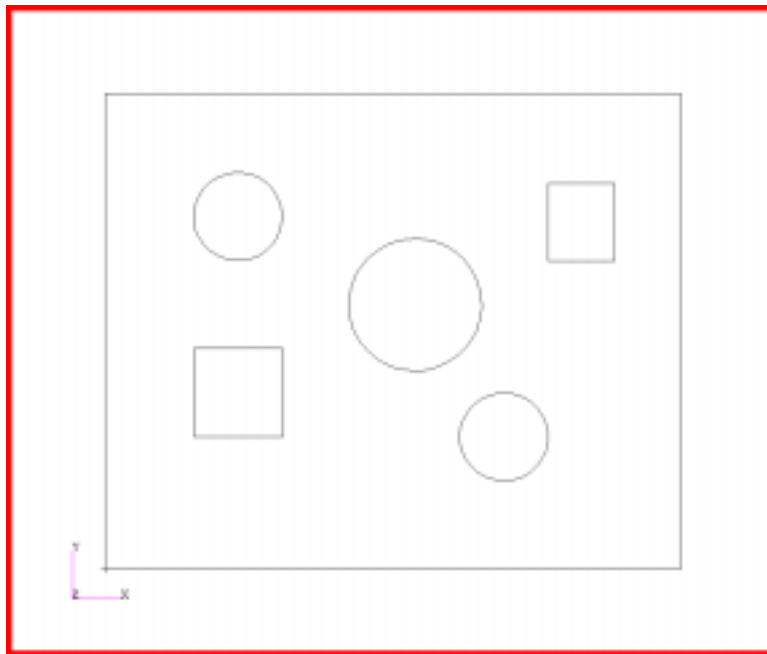
*Method:*

**Boundaries**

*Display Type*

◆ **Free Edges**

**Apply**



6. Create groups that represent the base component, chip, and epoxy finite elements.

**Group/Create...**

*Action:*

Create

*New Group Name*

base\_component

Make Current

*Group Contents:*

Add Entity Selection

*Entity Selection*

Elm 3044:3657:1

Apply

**Group/Create...**

*Action:*

Create

*New Group Name*

chip

Make Current

---

*Group Contents:*

**Add Entity Selection**

*Entity Selection*

**Elm 4886:6113:1**

**Apply**

**Group/Create...**

*Action:*

**Create**

*New Group Name*

**epoxy**

**Make Current**

*Group Contents:*

**Add Entity Selection**

*Entity Selection*

**Elm 3658:4885:1**

**Apply**

7. Post the **Chip**, **default\_group**, and **epoxy** groups to show the finite elements in the group.

**Group/Post...**

*Action:*

**Post**

*Select Group to Post*

**Chip  
default\_group  
epoxy**

**Apply**

**Group/Post...**

*Action:*

**Post**

*Select Group to Post*

**default\_group**

**Apply**

8. Define the three different materials using the specified thermal conductivity.

◆ **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="component"/>
<b>Input Properties</b>	
<i>Thermal Conductivity</i>	<input type="text" value="0.89"/>
<input type="text" value="OK"/>	
<input type="text" value="Apply"/>	

◆ **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="glue"/>
<b>Input Properties</b>	
<i>Thermal Conductivity</i>	<input type="text" value="0.2"/>
<input type="text" value="OK"/>	
<input type="text" value="Apply"/>	

◆ **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="circuit_board"/>
<b>Input Properties</b>	
<i>Thermal Conductivity</i>	<input type="text" value="0.3"/>

---

**OK**

**Apply**

9. Apply the material properties to the finite elements.

◆ **Properties**

*Action:*

**Create**

*Object:*

**3D**

*Type:*

**Solid**

*Property Set Name*

**pcb**

**Input Properties**

*Material Name*

**m:circuit\_board**

**OK**

*Select Members*

**Elm 6114:13427:1**

**Add**

**Apply**

◆ **Properties**

*Action:*

**Create**

*Object:*

**3D**

*Type:*

**Solid**

*Property Set Name*

**chip**

**Input Properties**

*Material Name*

**m:component**

**OK**

*Select Members*

**Elm 4886:6113:1**

**Add**

**Apply**

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="3D"/>
<i>Type:</i>	<input type="text" value="Solid"/>
<i>Property Set Name</i>	<input type="text" value="epoxy"/>
<b>Input Properties</b>	
<i>Material Name</i>	<input type="text" value="m:glue"/>
<input type="text" value="OK"/>	
<i>Select Members</i>	<input type="text" value="Elm 3658:4885:1"/>
<input type="text" value="Add"/>	
<input type="text" value="Apply"/>	

10. Apply the heat flux to the top surfaces of the chips

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Applied Heat"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>New Set Name</i>	<input type="text" value="heatflux"/>
<i>Target Element Type:</i>	<input type="text" value="3D"/>
<b>Input Data</b>	
<i>Form Type:</i>	<input type="text" value="Basic"/>
<i>Heat Flux</i>	<input type="text" value="3"/>
<input type="text" value="OK"/>	
<b>Select Application Region</b>	

*Geometry Filter*

◆ **FEM**

Before you select the element faces, change the rectangular picking preference to **Enclose Entire Entity**.

**Preferences/Picking...**

---

*Rectangle/Polygon Picking*

◆ **Enclose entire entity**

**OK**

Also, in the **FEM Select Menu**, pick the Face of element icon.



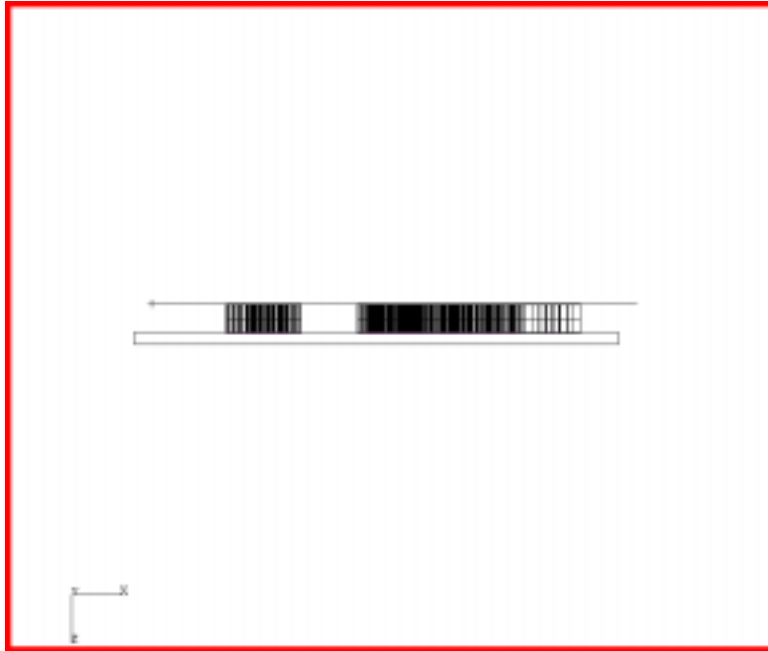
**Face of element**

Next, click on the **Top View** button on the main menu.



**Top View**

Now drag a rectangular box across the end of the chip elements as shown in the following figure.



*Select 3D Element Faces*

**Elm 4760:5372:1**

**Add**

**OK**

**Apply**

Set the boundary condition on the circuit board so it is permanently kept at 40 degree.

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Temp (Thermal)"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name</i>	<input type="text" value="temp_edge"/>

<i>Boundary Temperature</i>	<input type="text" value="40"/>
-----------------------------	---------------------------------

*Geometry Filter*                      ◆ **FEM**

Select the right and left edges of the circuit board.

<i>Select Nodes</i>	<input type="text" value="Node 2 67:120"/>
---------------------	--

- Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here:

◆ **Analysis**

<i>Action:</i>	<input type="text" value="Analyze"/>
<i>Object:</i>	<input type="text" value="Entire Model"/>
<i>Type:</i>	<input type="text" value="Analysis Deck"/>

<i>Data Output:</i>	<input type="text" value="XDB and Print"/>
---------------------	--

---

*Solution Type*

◆ **STATIC STATE ANALYSIS**

**Solution Parameters**

*Default Init Temperature*

**70**

**OK**

**OK**

**Apply**

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

12. Submit the input file to MSC.Nastran for analysis.
  - 12a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran pcb.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
  - 12b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran pcb scr=yes**. Monitor the run using the UNIX **ps** command.
13. When the run is completed, edit the **pcb.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.

---

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

15. Proceed with the Reverse Translation process, that is, attaching the **pcb.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*

**pcb.xdb**

**OK**

**Apply**

16. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

Choose the **Default** result case, and plot the result by selecting **Temperature** in the Select Fringe Result.

◆ **Results**

*Action:*

**Create**

*Object:*

**Quick Plot**

*Select Result Cases*

**Default**

*Select Fringe Result*

**Temperature**

**Apply**