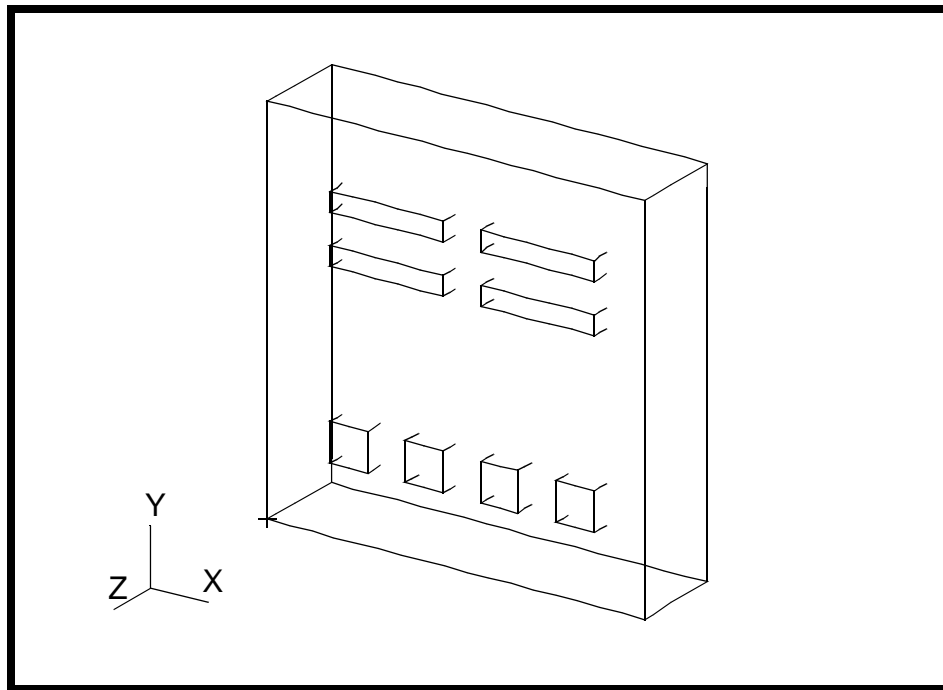


WORKSHOP 3

Equivalence and Verify the Hybrid Mesh



Objective:

- In this exercise you will equivalence the 3D Hybrid Microcircuit model mesh.
- You will sample the finite element verification functions to examine the aspect ratio, skewness, and taper of the mesh elements.



Model Description:

The hybrid microcircuit is monolithic: all material interfaces have negligible resistance to heat transfer. Hence, there are no contact resistances modeled in the structure and the entire model is materially continuous. In this exercise you will identify any incongruities, “cracks”, in the finite element mesh and equivalence to eliminate them.

You will examine the completed mesh with quantitative verification tools. You will evaluate the mesh element aspect ratio, taper, and skewness. These are generally useful in qualitatively assessing the accuracy of results and identifying problem areas for convergence to a solution.

Exercise Overview:

- Open the existing database named **microcircuit.db**.
- Use **Finite Element/Verify/Element/Boundaries** to identify any “cracks” which remain as artifacts from the geometry creation and meshing process.
- **Equivalence/All/Tolerance Cube** to eliminate duplicate nodes and eliminate “cracks” in the mesh.
- **Verify/Hex/Aspect** to identify elements with aspect ratios greater than 3.0.
- **Verify/Hex/Face Skew** to identify elements with face skew angles greater than 110 degrees.
- **Verify/Hex/Face Taper** to identify elements with highly tapered faces.
- **Quit** MSC.Patran.

Exercise Procedure:

Open an existing database

1. Open the existing database

Within your window environment change directories to the microcircuit.db working directory. Run MSC.Patran by typing **p3** in your xterm window.

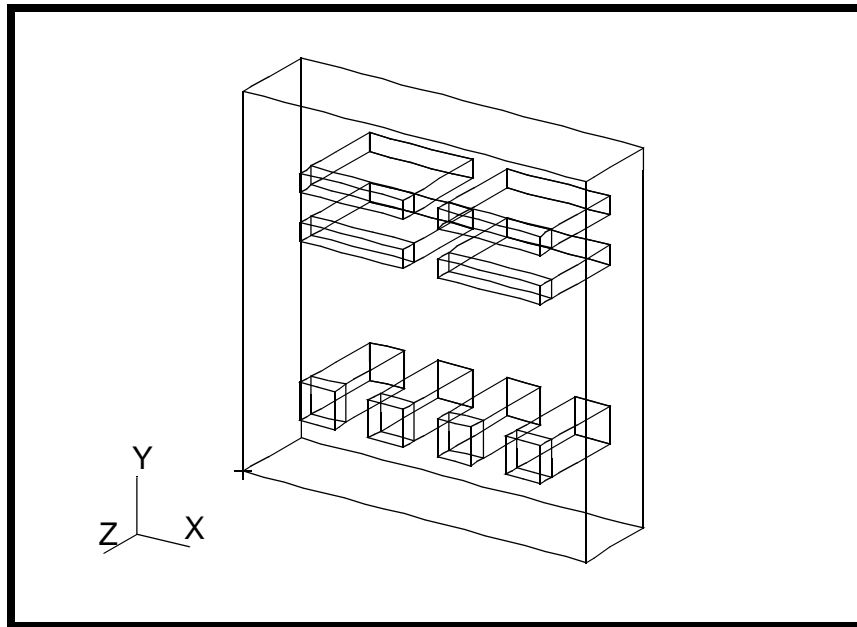
Next, select **File** from the *Menu Bar* and select **Open...** from the drop-down menu. Select the name **microcircuit.db** from the *Database List* box.

Select **OK** to open the database.

MSC.Patran will open a Viewport and change various *Control Panel* selections from a ghosted appearance to a bold format.

2. Identify any “cracks” which remain as artifacts from the geometry creation and meshing process.

To identify mesh boundaries select the **Finite Elements Applications** radio button. Set the *Action*, *Object*, and *Test* to **Verify/Element/Boundaries**. Select **Apply**. The display should appear as shown below.



Identify cracks in mesh

Equivalence mesh nodes

3. Equivalence nodes to eliminate duplicate nodes and eliminate “cracks” in the mesh.

Set the *Action*, *Object*, and *Method* to **Equivalence/All/Tolerance Cube**. Select **Apply** to complete the function.

The nodes bounding the interior cracks will be circled in the display and the Command Line will indicate that a number of nodes are deleted.

Reexamine the mesh boundaries after equivalencing with **Verify/Element/Boundaries**. Your model should appear as shown on the front panel of this exercise.

- Identify elements with aspect ratios greater than 3.0.

In the Finite Elements form set the *Action*, *Object*, and *Test* to **Verify/Hex/Aspect**. Change the *Aspect Ratio* to approximately **3.0**. Select **Apply**. Since the paver mesher was used your results may vary from those shown below. Now select **Plot Failed Elements Only**. The completed form and resultant display are shown below.

Evaluate element aspect ratios

The image shows the 'Finite Elements' dialog box in MSC.Patran. The 'Action' is set to 'Verify', 'Object' to 'Hex', and 'Type' to 'Aspect'. The 'Reliability Threshold' section includes a 'Normalize' checkbox and an 'Aspect Ratio' slider set to 3.07. The 'Element Plot Options' section has 'Color Code Elements' and 'Plot Failed Elements' checked. The 'Apply' button is highlighted. In the background, a 3D mesh plot of a mechanical part is shown with a color scale on the right ranging from 1.00+00 to 4.56+00. The plot title is 'MSC/PATRAN Version 8.013- Oct-98 14:15:45' and 'Fringe: Verify Hex Element Aspect Ratio'. The bottom right of the plot area shows 'default Fringe: Min: 4.56+00@Elem 1298.1 Max: 3.00+00@Elem 1111.1'.

MSC.Thermal is reliable in converging to a solution even with elements of relatively high aspect ratio. However it is left to the analyst to decide whether too large a gradient is resolved across the long dimension of a high aspect ratio element. If this is so then resolving the mesh to a lower aspect ratio in that area will yield more accuracy.

- Identify elements with face skew angles greater than 110 degrees.

In the **Finite Elements** form set the *Action*, *Object*, and *Test* to **Verify/Hex/Face Skew**. Change the *Face Skew Angle* to approximately **20.0**. Select **Apply**. Since the paver mesher was used your results may vary from those shown below. Now select **Plot Failed Elements Only**. The completed form and resultant display are shown below.

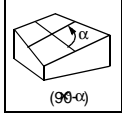
Finite Elements

Action:

Object:

Type:

Reliability Threshold



(90°)

Normalize

Analysis Code:
MSC/THERMAL

0.

Face Skew Angle

Element Plot Opt

Color Code El

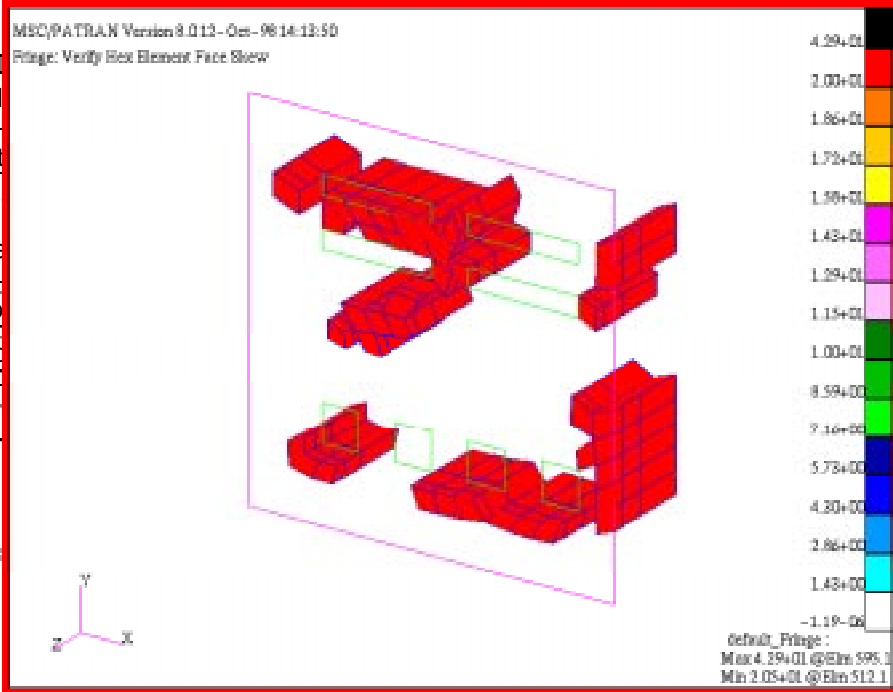
Plot Failed Ele

Fringe Attrib

Reset Graph

MSC/PATRAN Version 8.0.13 - Oct - 98 14:13:50

Pringe: Verify Hex Element Face Skew



4.29E-01
 2.00E-01
 1.86E-01
 1.72E-01
 1.58E-01
 1.43E-01
 1.29E-01
 1.15E-01
 1.00E-01
 8.59E-02
 7.16E-02
 5.73E-02
 4.30E-02
 2.86E-02
 1.43E-02
 -1.18E-06

default_Pringe :
 Max 4.39E-01 @ Elms 595.1
 Min 2.05E-01 @ Elms 512.1

The finite element formulation of an R-C network in MSC.Thermal has been developed to provide a higher order accuracy to the resultant temperature distribution than is available with the traditional lumped mass/element centroid technique. Hence, even meshes with skewed elements yield results which do not contain artifacts of the mesh geometry.

6. Identify elements with highly tapered faces.

In the Finite Elements form set the *Action*, *Object*, and *Test* to **Verify/Hex/Face Taper**. Change the *Face Taper* to approximately **0.90**. Select **Apply**. Since the paver mesher was used your results may vary from those shown below. Now select **Plot Failed Elements Only**. The completed form and resultant display are shown below.

**Evaluate
element face
taper**

The image shows the MSC.Patran software interface. On the left is the 'Finite Elements' dialog box with the following settings:

- Action: Verify
- Object: Hex
- Type: Face Taper
- Reliability Threshold: 0.90 (indicated by a slider)
- Analysis Code: MSC/THERMAL
- Element Plot Options: Color Code Element, Plot Failed Elements
- Buttons: Apply, Reset

On the right is a 3D model of a mechanical part with a color-coded temperature distribution. A legend on the right side of the model shows a color scale from 5.82 to 1.00. The model is titled 'MSC/PATRAN Version 8.0.12-Oct-98 14:36:41' and 'Fringe: Verify Hex Element Face Taper'. A coordinate system (X, Y, Z) is visible at the bottom left of the model.

The verification functions in MSC.Patran provide a tool for quantifying the geometric quality of a finite element mesh. The criteria which determine the performance of the mesh in a numerical analysis remain the province of the analyst and his or her experience with the particular analysis.

Click **Reset Graphics** in the Finite Elements Verify form.

Quit MSC.Patran

7. Quit MSC.Patran.

To stop MSC.Patran select **File** on the *Menu Bar* and select **Quit** from the drop-down menu.