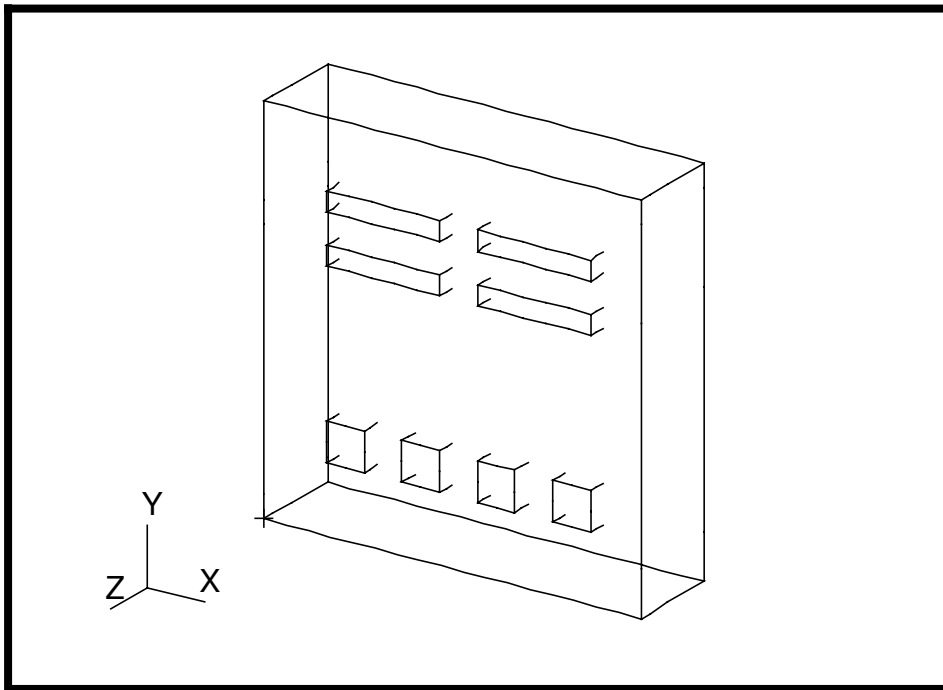

Exercise 3

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.

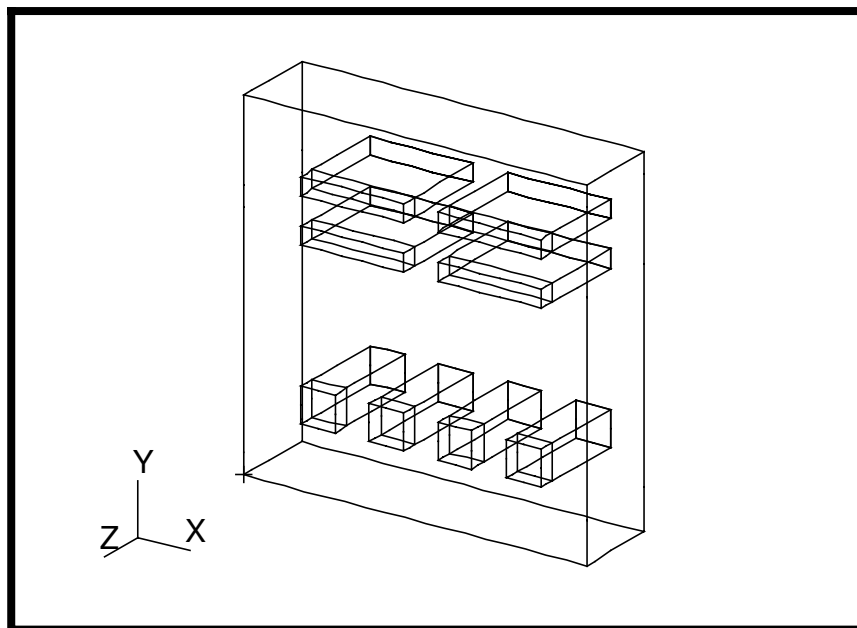
Next, select **File** from the *Menu Bar* and select **Open...** from the drop-down menu. Select the name **microcircuit.db** from the *Database List* box.

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

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.

Equivalence mesh nodes

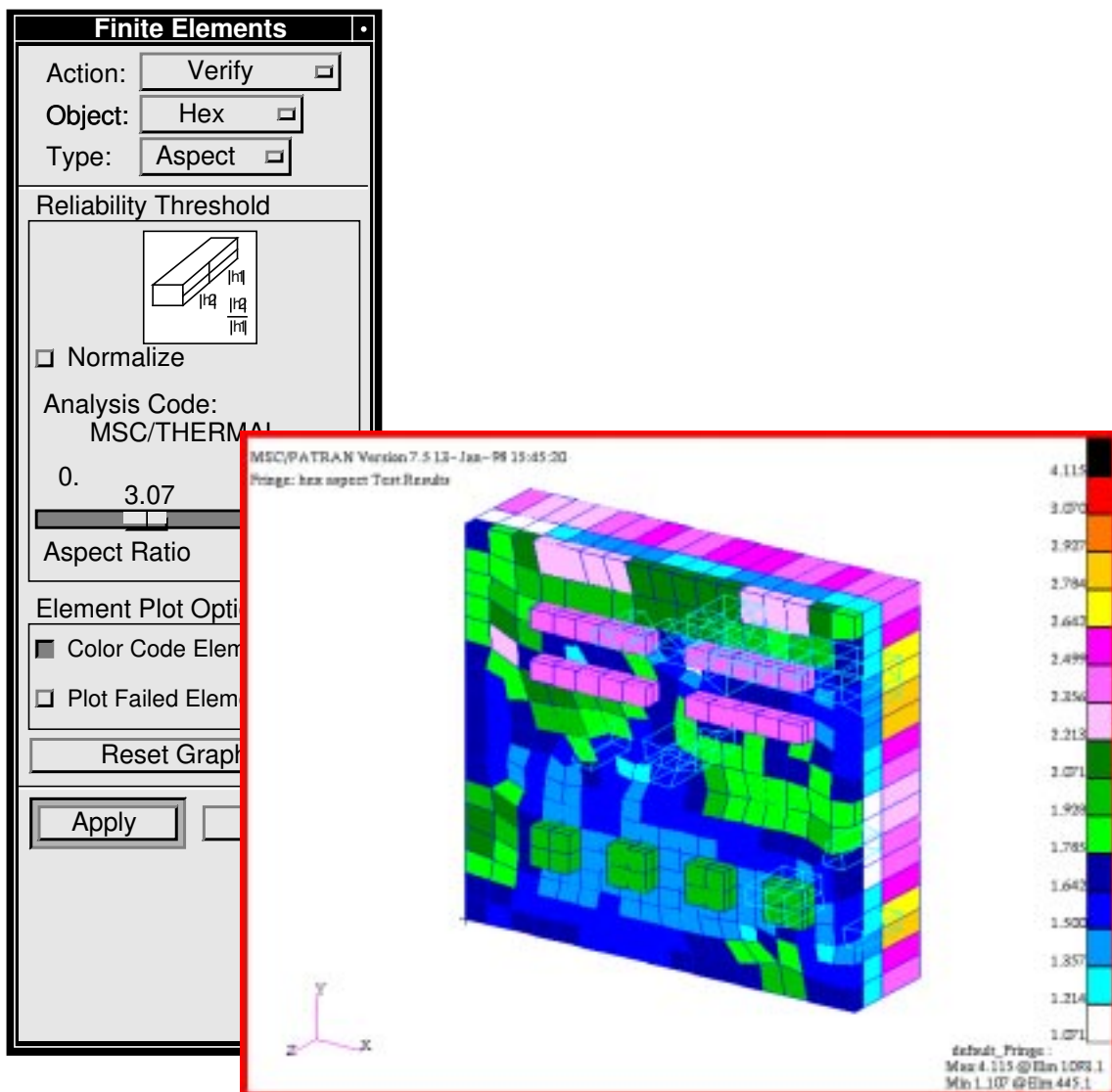
The nodes bounding the interior cracks will be circled in the display and the Command Line will indicate that a number of nodes are deleted.

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**. The completed form and resultant display are shown below. Since the paver mesher was used your results may vary from those shown below.

**Evaluate
element
aspect ratios**



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.

Evaluate element face skewness

- 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**. The completed form and resultant display are shown below. Since the paver mesher was used your results may vary from those shown below.

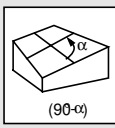
Finite Elements

Action:

Object:

Type:

Reliability Threshold



(90- α)

Normalize

Analysis Code:
MSC/THERMAL

0.

Face Skew Angle

Element Plot Options

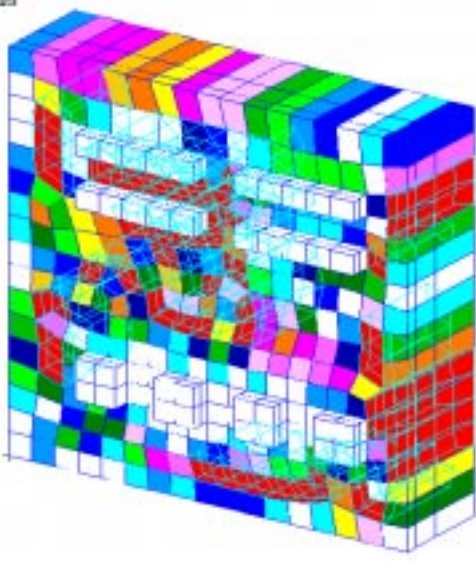
Color Code Elements

Plot Failed Elements

Reset Graphics

MSC/PATRAN Version 7.5.13- Jan-98 15:48:22

Fringe: hex face skew Test Results



42.28
30.05
18.62
17.19
15.75
14.32
13.89
11.46
10.03
8.593
7.161
5.729
4.296
2.864
1.432
-0.0000

default Fringe:
Max:42.28 @ Elm 623.1
Min: 0. @ Elm 27.1

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.

- 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**. The completed form and resultant display are shown below. Since the paver mesher was used your results may vary from those shown below.

**Evaluate
element face
taper**

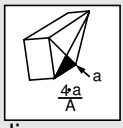
Finite Elements

Action:

Object:

Type:

Reliability Threshold



Normalize

Analysis Code:
MSC/THERMAL

0.

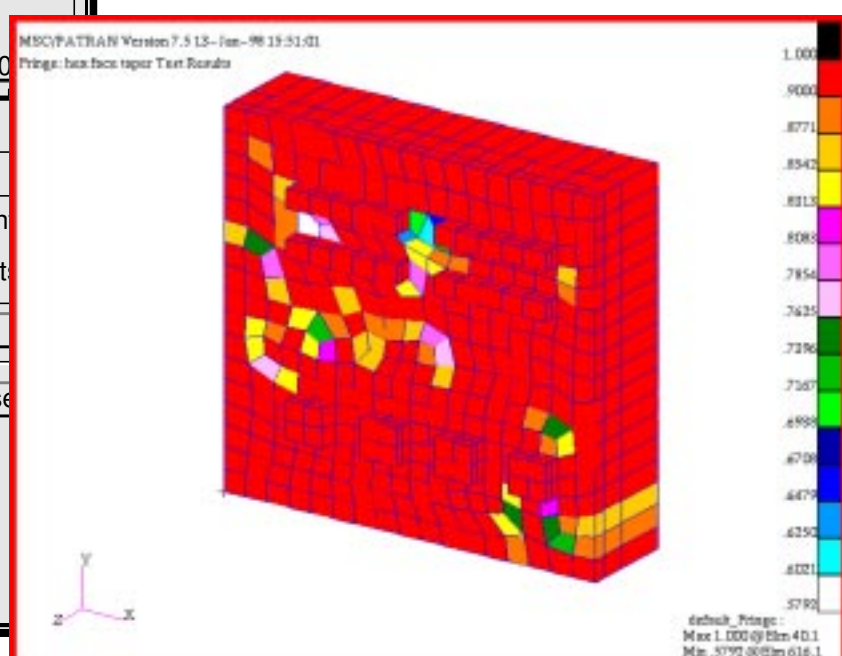
Face Taper

Element Plot Options

Color Code Elements

Plot Failed Elements

Reset Graphics



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.

7. Quit MSC/PATRAN.

Before exiting, reset graphics using the *Reset Graphics* icon.



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