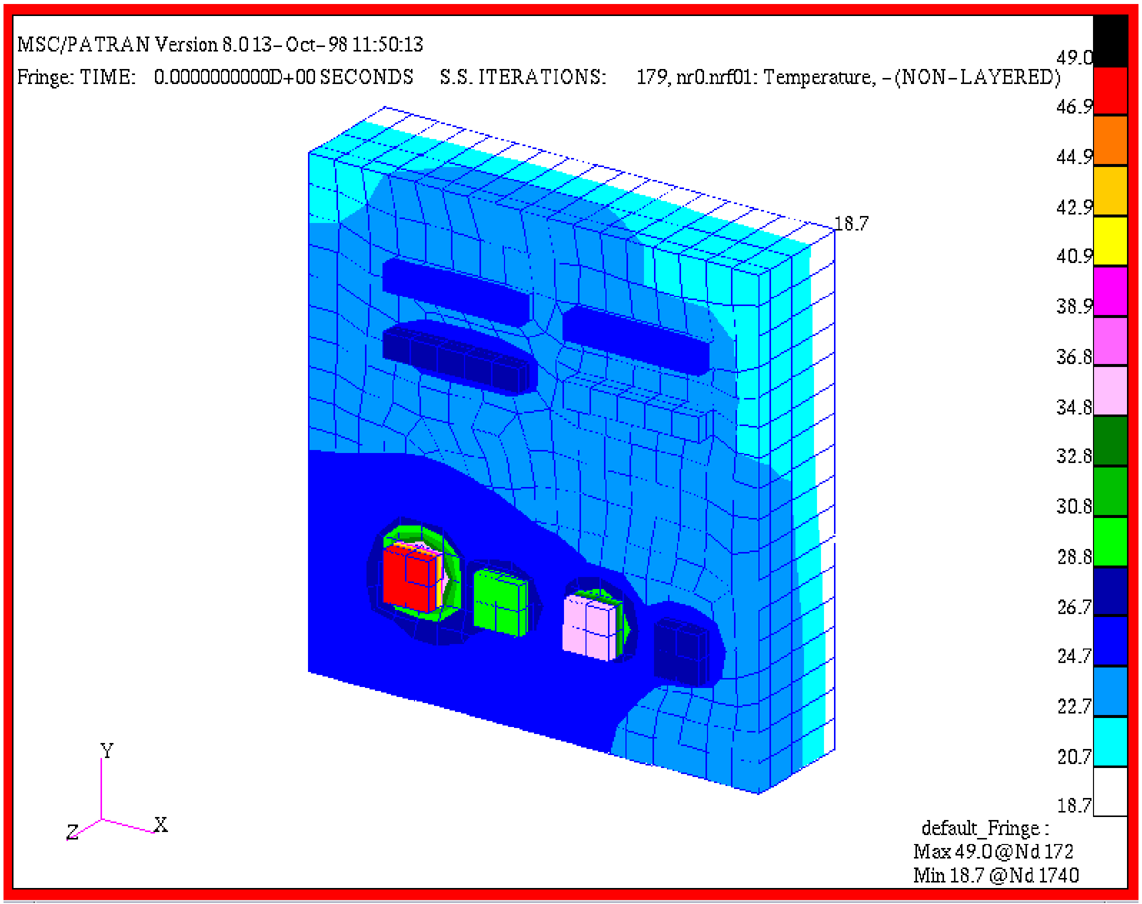
WORKSHOP 9

Thermal Analysis of the Hybrid Microcircuit



Objective:

- In this exercise you complete a steady state thermal analysis of the 3D hybrid microcircuit.

Model Description:

In this exercise complete the analysis of a hybrid microcircuit which is subjected to a bench functional test. The hybrid is clamped to a test fixture which is chilled by iced water. The microcircuit is continuously flushed by a dry nitrogen purge at **21^{o}C**.

During functional testing, which takes approximately **1 hour**, the entire hybrid dissipates **8 watts**. Each device dissipates a constant wattage, as listed. The goal of the analysis is to verify that all device temperature shall remain below **50^{o}C**.

Exercise Overview:

- Open the existing database named **microcircuit.db**.
- Use **Finite Elements/Create/Node/Edit** to create the two fixed temperature boundary nodes.
- With **Display/Finite Elements...** or the equivalent Tool Bar function increase the display size of nodes to facilitate boundary definition.
- Use **Loads/BCs/Create/Temperature/Nodal** with **Option: Fixed** to set the boundary node temperatures.
- Use **Loads/BCs/Create/Convection** with **Option: Fixed Coefficient** to apply the contact and nitrogen flow heat transfer coefficients.
- Post only the **device_and_solder** group and use the *middle mouse button* or various **Viewing** functions to expose the individual device surfaces.
- Use· **Loads/BCs/Create/Heating** with **Option: Template, Volumetric Heat** to apply the heating load to the individual devices.
- Select **Analysis** to prepare and to submit the model for analysis and to **Read Results**.
- Post **hybrid_fem**, select an **isometric_view**, select **Results**, and review results data.
- **Quit** MSC.Patran.

Hybrid microcircuit boundary conditions

Figure 1-Boundary Conditions

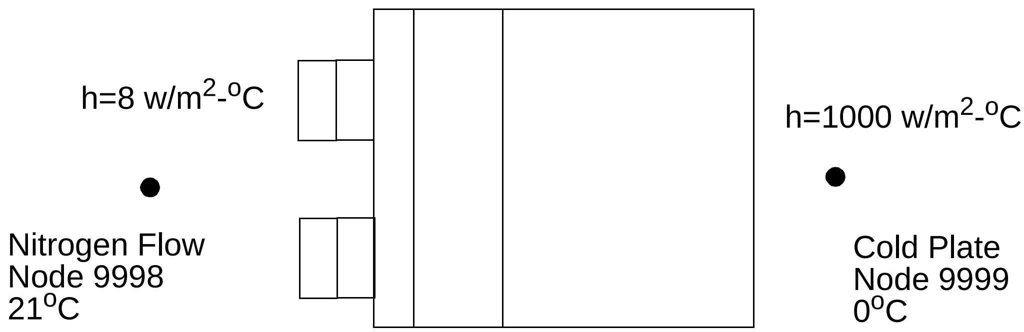
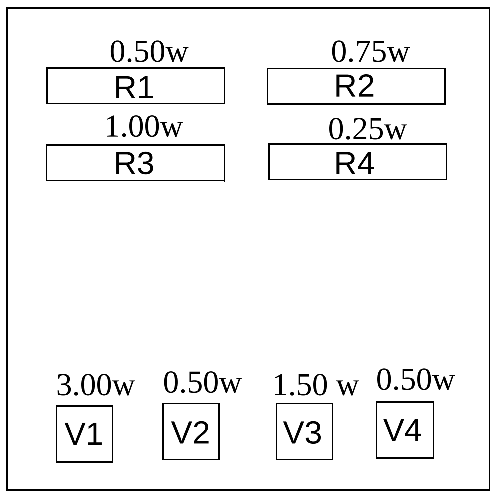


Figure 2-Device Position



Total 8.00 watts

Table 2-Device Heat Generation

Device	w/m^3
R1	0.167E+09
R2	0.250E+09
R3	0.333E+09
R4	0.083E+09
V1	1.500E+09
V2	0.250E+09
V3	0.750E+09
V4	0.250E+09

Exercise Procedure:

1. Open the existing database named **microcircuit.db**.

Open an 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 open the existing microcircuit database.

File	
Open...	
Database List	**microcircuit.db**
OK	

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

2. Create the two fixed temperature boundary nodes.

Create 2 boundary nodes

Select **Viewing** from the *Menu Bar* or use the Tool Bar *Right Side View* icon to change to a side_view of the model hybrid_fem entities.

Viewing	
Named View Options...	
Select Named View	**side_view**
Close	

Select the **Finite Elements** *Applications radio button*. Create two nodes which **are not associated with geometry**. The first node is numbered **9998**.

◆ Finite Elements	
Create/Node/Edit	
Node ID List	**9998**
☐ Associate with Geometry	
Node Location List	**[0.01 0.01 0.002]**
Apply	

The second node is numbered 9999.

◆ **Finite Elements**

Create/Node/Edit

Node ID List **9999**

☐ **Associate with Geometry**

Node Location List **[0.01 0.01 -0.007]**

Apply

Change display and picking preferences

3. Increase the display size of nodes and picking preferences to facilitate boundary definition.

Increase the display size of nodes and modify the Picking Preferences to facilitate the application of boundary condition. Use either **Display/Finite Element/Node Size** or the associated Tool Bar icon to change the node size.

Display

Finite Elements...

Node Size (Use Slider Bar) **6**

Apply

Cancel

And, select **Preference/Picking...** to change the *Rectangle/Polygon* picking method to **Enclose Centroid**.

Preferences

Picking...

◆ **Enclose Centroid**

Close

Select **Display/ Load/BC/Element Props /Vectors...** to facilitate viewing boundary conditions.

Display

Load/BC/Elem. Props...

☐ **Show LBC/El. Prop Values** **<deselect this option>**

Vectors/Filters...

Scale Factors: **0.05**

Apply

Cancel

Apply

Cancel

4. Fix the boundary node temperatures.

Fix nodal boundary temperatures

Begin applying boundary conditions. Select the **Loads/BCs** *Applications radio button*.Create a fixed temperature boundary named **Cold_plate**.

◆ Loads/BCs

Create/Temperature/Nodal

Option: **Fixed**

New Set Name **Cold_plate**

Input Data...

In the Input Data form define the fixed temperature.

Fixed Temperature **0.0**

OK

Select Application Region...

In the Select Applications Region form pick node **9999**.

◆ FEM

Select Nodes **<select node 9999>**

Add

OK

Apply

Repeat this process for a *New Set Name* **Nitrogen** with a fixed temperature of **21.0** applied to **Node 9998**.

New Set Name	**Nitrogen**
Input Data...	
Fixed Temperature	**21.0**
OK	
Select Application Region...	
◆ **FEM**	
Select Nodes	**<select node 9998>**
Add	
OK	
Apply	

The display should highlight each node and append the fixed temperature. On some displays the symbol and value may be difficult to discern.

Apply convection boundary conditions

5. Apply contact and nitrogen flow heat transfer coefficients.

Create two convective boundary conditions with the **Use Correlations** option and the heat transfer coefficients provided in Figure 1. Name the first set **nitrogen_flow** and apply the boundary condition to all of the element free faces on the top and sides of **hybrid_fem**.

◆ **Loads/BCs**

Create/Convection/Element Uniform	
Option:	**Fixed Coefficient**
New Set Name	**nitrogen_flow**
Target Element Type	**3D**
Region 2	**Nodal**
Input Data...	

In the Input Data form provide the convection coefficient and fluid node association.

Convection Coefficient	**8.0**
OK	
Select Application Region...	

In the Select Applications Region form select all the free faces of the top and sides of the model. Exclude the bottom of the model by not enclosing it in the dragged rectangle.

◆ **FEM**

Application Region / Select 3D Element Faces — **<select all top and side free faces by dragging a rectangle around them, as shown below>**

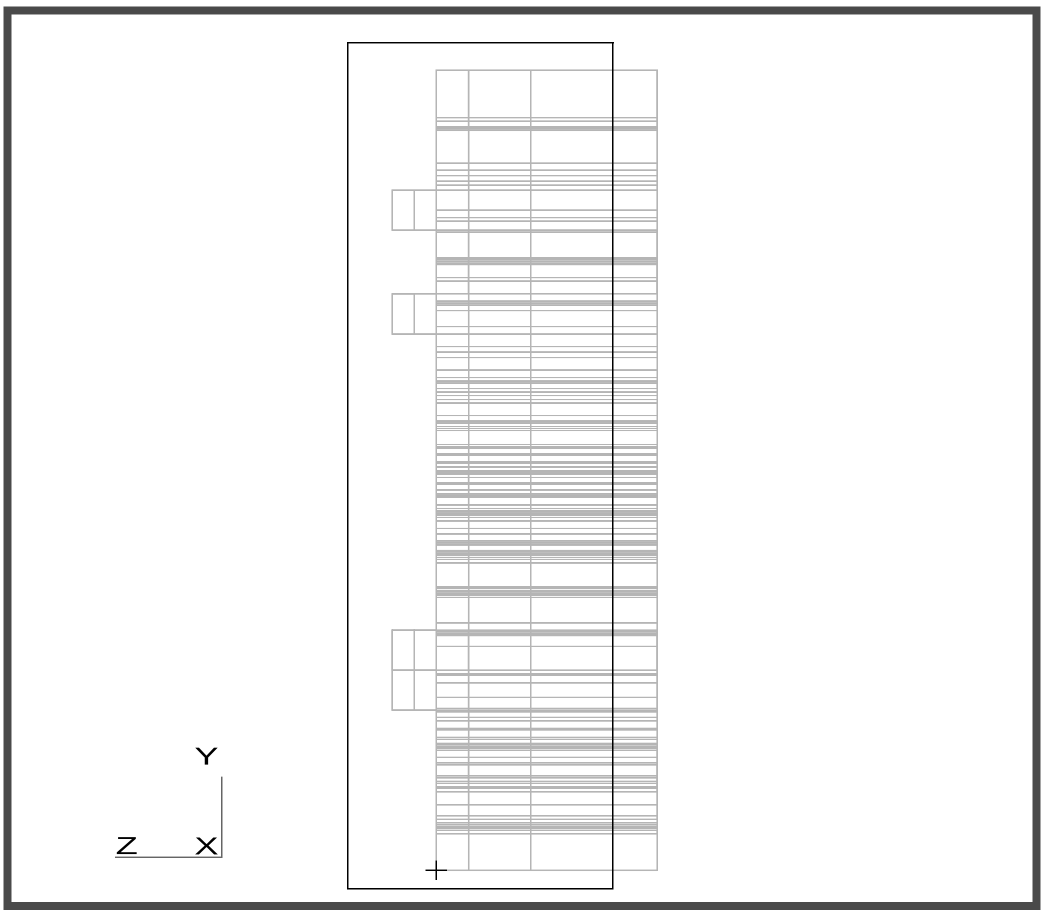
Add

Coupling Region / Select Nodes — **<select node 9998>**

Add

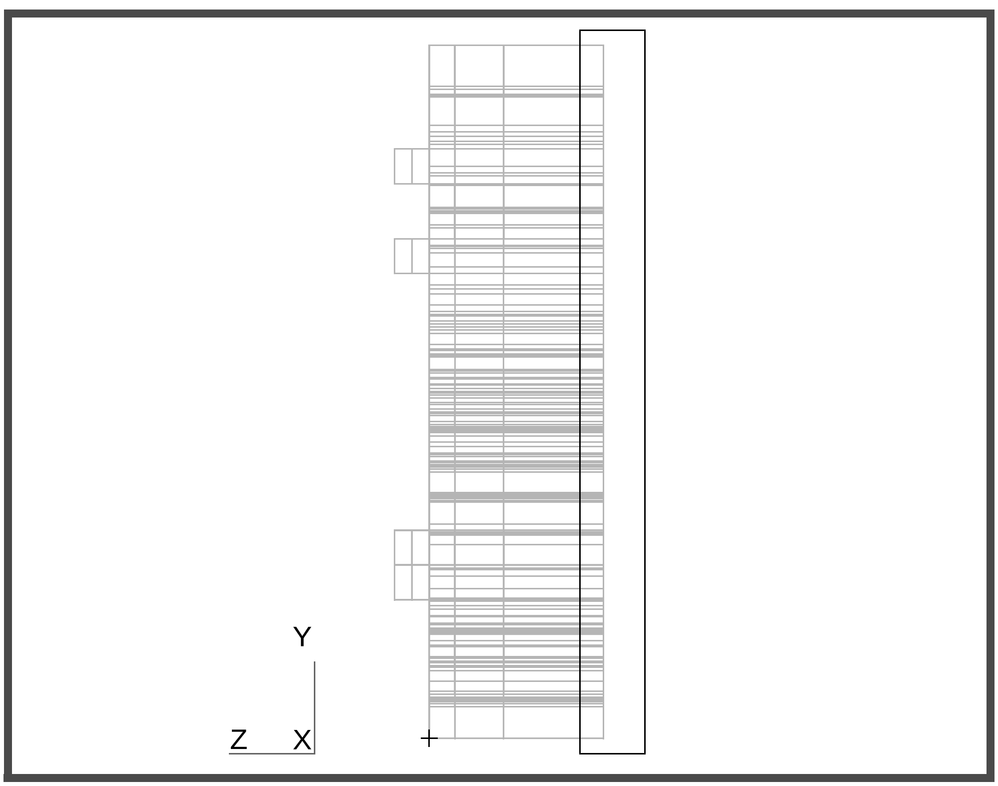
OK

Apply



Repeat this process for a *New Set Name* **heat_sink** with a convection coefficient of **1000.0** applied to the bottom surface of the hybrid_fem.

New Set Name	**heat_sink**
Input Data...	
Convection Coefficient	**1000.0**
OK	
Select Application Region...	
◆ **FEM**	
Application Region / Select 3D Element Faces	**<select all bottom free faces by dragging a rectangle around them>**
Add	
Coupling Region / Select Nodes	**<select node 9999>**
Add	
OK	
Apply	



Post only device_and_ solder

6. Post only the **device_and_solder** group and rotate to a view which shows the top device elements

Select **Group/Post...** and **Reset Graphics** to facilitate applying volumetric heat loads.

Group

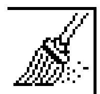
Post...

Select Groups to Post	**device_and_solder**

Apply

Cancel

Reset Graphics



Select **Viewing** from the *Menu Bar* or use the Tool Bar *Iso 1 View* icon to change to a isometric_view of the device_fem entities.

Viewing

Named View Options...

Select Named View	**isometric_view**

Close

Apply device volumetric heat loads

7. Apply device volumetric heat loads.

Based on the data in **Table 2** apply volumetric heat loads to **R1** through **V4**, the surface mounted components. The heat load should be placed only on the top layer of elements, the silicon devices.

◆ **Loads/BCs**

Create/Heating/Element Uniform

Option:	**Template, Volumetric Heat**
New Set Name	**R1**
Target Element Type	**3D**

Input Data...

Vol Heat Generation | **0.167E+09**

OK

Select Application Region...

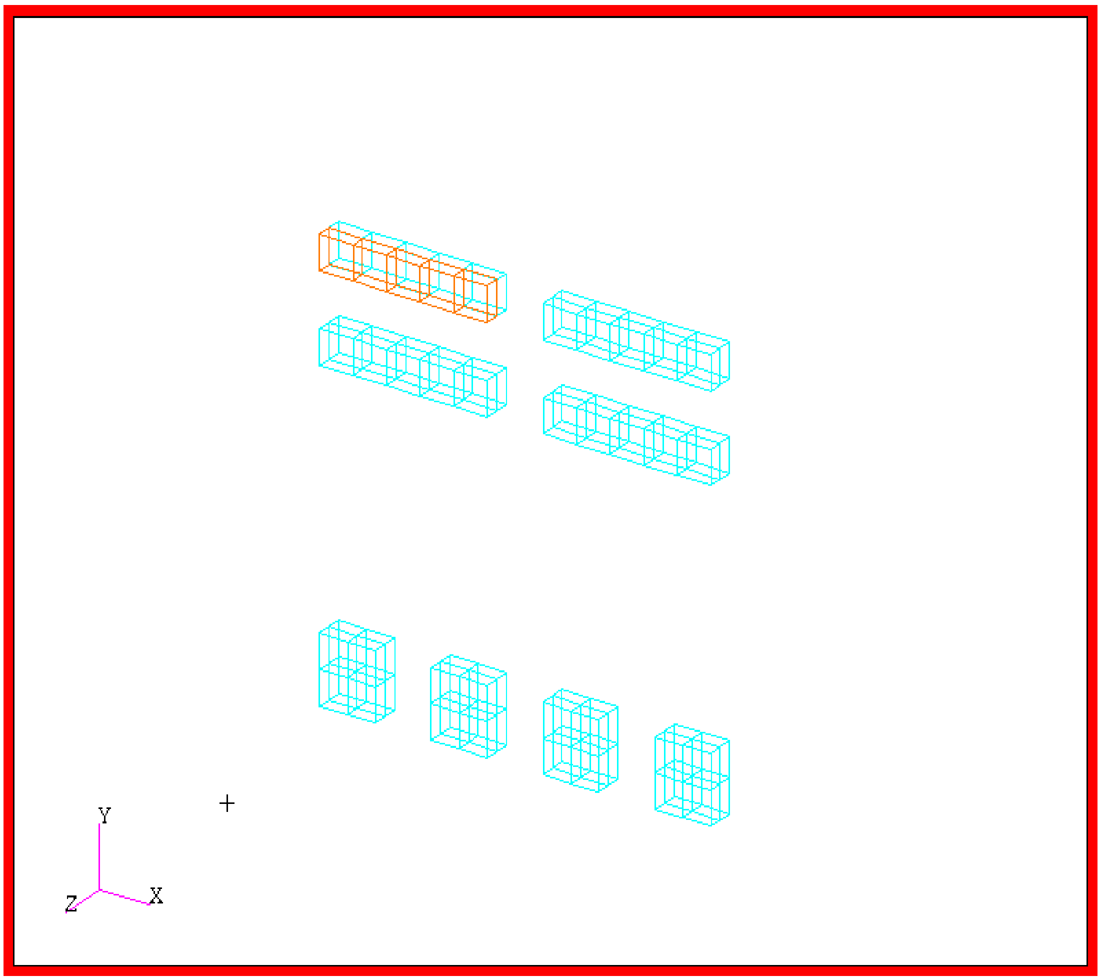
◆ **FEM**

Select 3D Elements — **<select the top elements of R1 using shift- left mouse button>**

Add

OK

Apply



Repeat the application for *New Set Names* R2 through V4. Use Figure 2 on page 9-4 to correlate heat load to device locations.

The continuous display of LBC markers, vectors and their values should have provided positive indication of the correct application of the LBC's. If you would like to further verify that the two fixed temperature, two heat transfer coefficient, and eight volumetric heating rate LBC's are correctly applied use the **Show Tabular**, **Plot Contours**, and **Plot Markers** *Action:* selections in the Load/Boundary Conditions form. You may also wish to **Group/Set Current...** different groups to facilitate this LBC's check.

After completing LBC's verification **Group/Set Current... hybrid_fem**.

Group

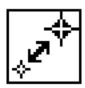
Post...

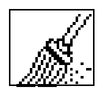
Set Groups to Post **hybrid_fem**

Apply

Cancel

Reduce the node size using the *Node Size* icon and reset graphics defaults using the *Broom* icon.





8. Prepare and submit the model for analysis.

Prepare and run analysis

Select the **Analysis** *Applications radio button* to prepare the analysis. Move through each of the five parameter forms reviewing and changing the settings or selections, if necessary, as shown below. The analysis will be submitted by selecting **Apply** in the Analysis form.

◆ Analysis

Analyze/Full Model/Full Run

Solution Parameters...

Calculation Temperature Scale **◆ Celsius**

Solver Option **1, Weakly Nonlinear Solution**

OK

Output Requests...

Units Scale for Output Temperatures | ◆ **Celsius**

OK

Apply

Read and plot results

9. Read results file and plot results.

From within MSC.Patran the only indication that the analysis has successfully finished is the existence of an **nrX.nrf.01** results file in a subdirectory one level below your working directory.

Recall that p3 was initiated from a working directory which contained the **microcircuit.db** database file. The analysis, initiated from within MSC.Patran, created a new subdirectory with the same name as the *Job Name*; it should be named **microcircuit**/. By using **Read Result** in the Analysis form and **Selecting Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of the results file.

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories | **<path>/microcircuit**

Filter

Available Files | **nr0.nrf.01**

OK

Select Rslt Template File...

Files | **pthermal_1_nodal.res_tmpl**

OK

Apply

To plot the results to posted FEM use the **Results** *Application radio button*.

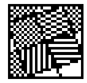
◆ Results

Create/Quick Plot

Select Result Cases | **TIME: 0.0000000000D+00 S...**

Select Fringe Result | **Temperature,**

Select the *Fringe Attributes* icon.



Display:	**Element Edges**
Label Style...	
Label Format:	**Fixed**
Significant figures	**3 <use slider bar>**
OK	
Apply	

The model should now appear as shown on the front panel of this exercise.

What is the maximum reported temperature? Is it at or below the required maximum of **50^{o}C**?

10. **Quit** MSC.Patran

Quit MSC.Patran

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