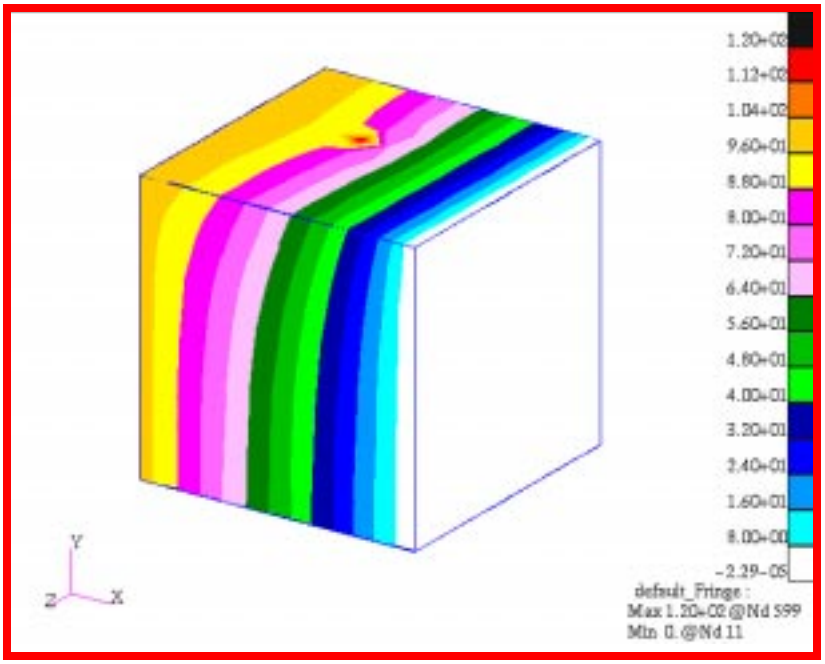
Supplementary Exercise - 3

Thermal Block



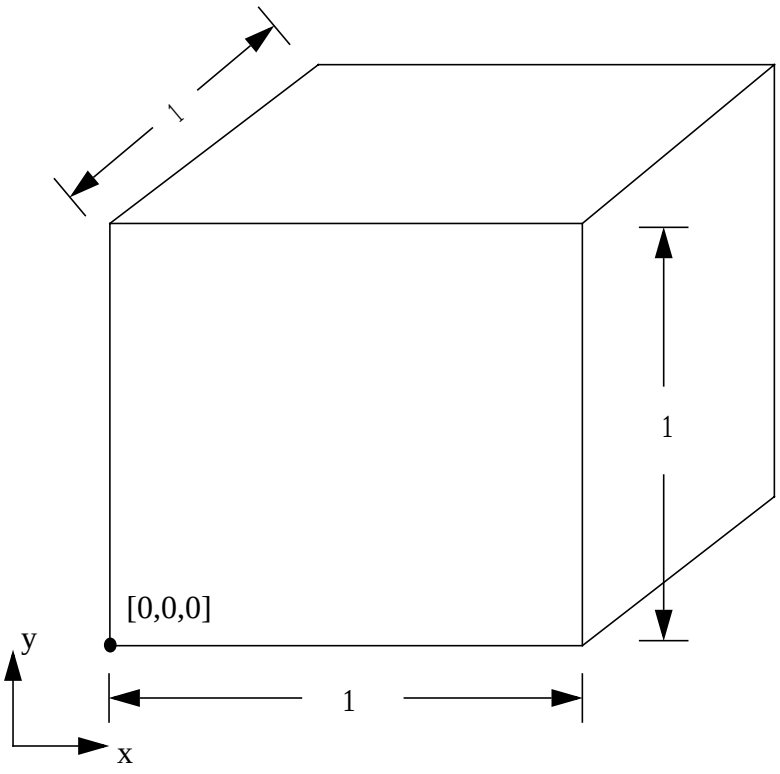
Objectives:

- Model various thermal conditions on a simple block model. This involves adding various boundary conditions:
  - Temperature
  - Heating, flux
  - Heating, nodal source
  - Radiation
  - Convection

Thermal Block

Model Description:

In this exercise you will create a simple cube conduction thermal model and apply several thermal loads and boundary conditions. For each added boundary condition, an analysis is performed.



Analysis Code:	MSC/THERMAL
Element type:	Hex8
Element Global Edge Length:	0.1

Figure 1-1

Suggested Exercise Steps:

- Create a new database named **block.db**.
- Change the Tolerance to Default and the Analysis Code to MSC/THERMAL.
- Create the geometry and finite element mesh using the information in Figure 1-1.
- Define a material named **alum**.
- Define a Property called **block**.
- Create a fixed temperature of 100^{o}C on left face of block; use set name **fixed_left**.
- Couple the right face of the block with **Node 9999** set to 0^{o}C; use set name **coupled_face**.
- Analyze the model and read in the results. View results using quick plot method.
- Add a constant heat flux called **constant_flux** to the model and re-analyze.
- Add a couple of heat nodal sources to the model and analyze.
- Remove the coupled nodes from the **default** load case. Change temperature of **Node 9999** to -100.0^{o}C. Add radiation to the model between the right side of block and **Node 9999**. Analyze.
- Create a new node and add convection to model. Analyze.
- The purpose of this exercise is to demonstrate how boundary conditions can be created for a simple situation (model).

Thermal Block

Exercise Procedure:

1. Create a new database and name it **block.db**.

File/New...

New Database Name **block**

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC/THERMAL**.

New Model Preference

Tolerance ◆ **Default**

Analysis Code: **MSC/THERMAL**

OK

3. Create the geometry and finite element mesh.

Create a solid

◆ ***Geometry***

Action: **Create**

Object: **Solid**

Method: **XYZ**

Vector Coordinate List **<1,1,1>**

Origin Coordinate List **[0, 0, 0]**

Apply

Mesh the model

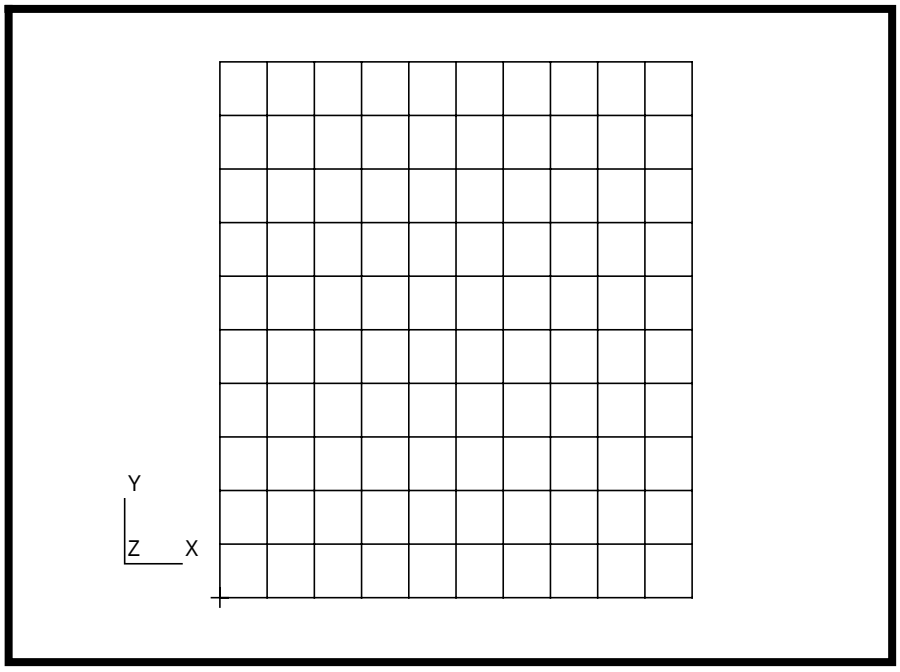
Now create the mesh for the model.

◆ ***Finite Elements***

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Global Edge Length	**0.1**
Mesher	◆ **Isomesh**
Element Topology	**Hex8**
Solid List	**Solid 1**

Apply

Your finite element model should look like the one shown in the figure below.



The following node to be created will be used as the coupling node for specifying the temperatures on the right side of the model.

Action:	**Create**
Object:	**Node**
Method:	**Edit**

Node ID List **9999**

☐ ***Associate with Geometry***

Node Location List **[1.5 .5 .5]**

Apply

4. Create a material property set for aluminum called **alum**.

Create Material

◆ ***Materials***

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name **alum**

Input Properties...

Thermal Conductivity **204**

Density **1.0**

Specific Heat **1.0**

Apply

Cancel

5. Define solid element properties with set called **block**.

Create Property

◆ ***Properties***

Action: **Create**

Object: **3D**

Type: **Thermal 3D Solid**

Property Set Name **block**

Input Properties...

Material name **m:alum**

OK

Select Members **Solid1**

Add

Apply

Create Loads and BCs

6. Create a fixed temperature on the left face of the block model. Set the temperature there to 100^{o}C (set name **fixed_left**). Then specify the temperature on the right face using the nodal coupling option-- couple the temperature at a single node to that of a set of nodes. The coupling node is Node 9999 and is at 0^{o}C.

◆ ***Loads/BCs***

Action: **Create**

Object: **Temperature (PThermal)**

Type: **Nodal**

Option: **Fixed**

New Set Name **fixed_left**

Input Data...

Fixed Temperature **100**

OK

Select Application Region...

◆ ***Geometry***

Select Geometry Entities ***select left face of the solid (Solid 1.1)***

Add

OK

Apply

Action: **Create**

Object: **Temperature (PThermal)**

Type: **Nodal**

Thermal Block

Option: **Coupled**

New Set Name **coupled_face**

Select Application Region...

◆ *FEM*

Slave Region

Select Nodes **select nodes on right face**

Add

Master Region

Select Nodes **Node 9999**

Add

OK

Apply

Set the temperature of the master region, ie. **Node 9999**.

Action: **Create**

Object: **Temperature (PThermal)**

Type: **Nodal**

Option: **Fixed**

New Set Name **fixed_node**

Input Data...

Fixed Temperature **0.0**

OK

Select Application Region...

◆ *FEM*

Select Nodes **Node 9999**

Add

OK

Apply

7. Analyze the model

◆ ***Analysis***

Action: **Analyze**

Object: **Full Model**

Method: **Full Run**

Solution Parameters...

Calculation Temperature Scale: ◆ **Celsius**

OK

Output Requests...

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

OK

Apply

8. Read in the results.

Action: **Read Result**

Object: **Result Entities**

Select Results File...

Select under Directories the jobname subdirectory, e.g. /baja/users/patran.class/jobname (as seen in Jobname).

Selected Results Files: **nr0.nrf.01**

OK

Select Rslt Template File...

Template File: **pthermal_1_nodal.res_tmpl**

OK

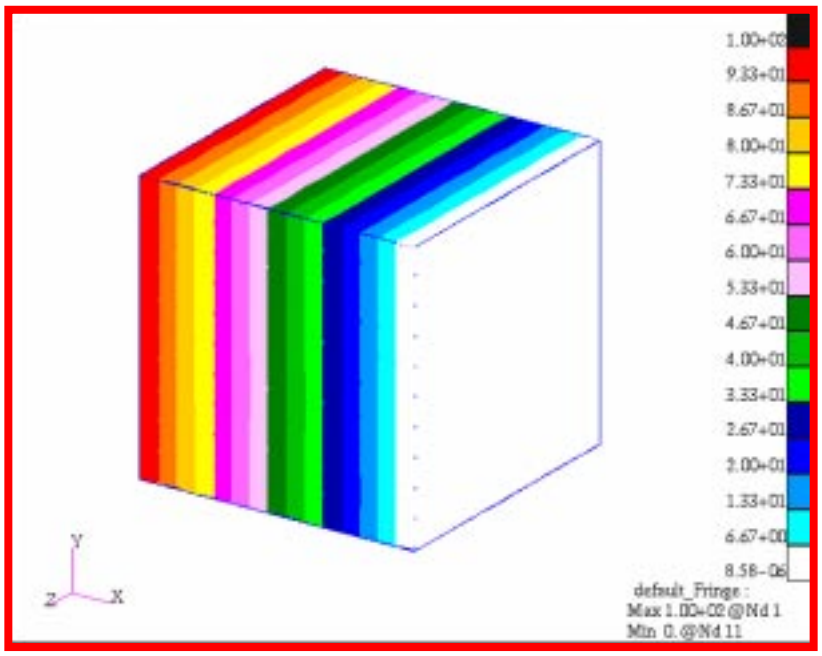
Apply

9. In Results, use quick plot and choose temperature as your fringe result.

◆ ***Results***

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	**0.000000000D +00**
Select Fringe Result:	**Temperature,**

Apply



You will need to reset the graphics by using the **Reset Graphics** Icon after viewing the results.

10. Add a heat flux to the top face of the solid cube.

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Heating (PThermal)**
Type:	**Element Uniform**
Option:	**Flux, Fixed**
New Set Name	**constant_flux**

Input Data...

Fixed Heat Flux	**10000**

OK

Select Application Region...

◆ ***Geometry***

Select Solid Faces	***select top face of solid (Solid 1.4)***

Add

OK

Apply

11. Check the load case named **Default**, that is always used for an MSC/Thermal analysis, to see that all loads and boundary conditions are being used.

◆ ***Load Cases***

Action:	**Modify**
Select Load case to Modify	**Default**

See the Loads/BCs that will be used for the analysis under Assigned Loads/BCs.

Cancel

12. Analyze the model and view the results (newest result case). After viewing the results reset graphics.

Change the job name for this analysis.

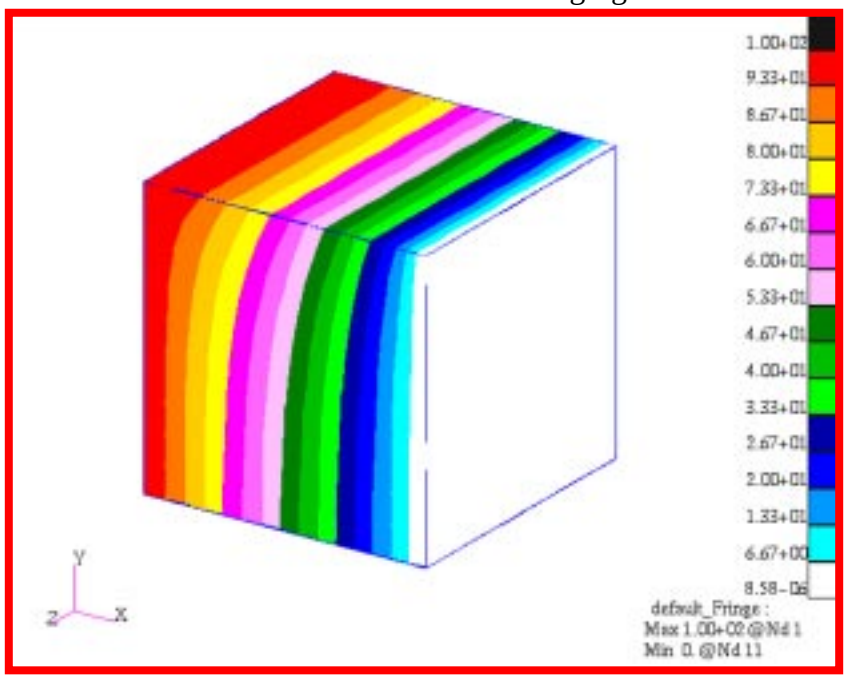
◆ ***Analysis***

Action:	**Analyze**
Object:	**Full Model**
Method:	**Full Run**
Job Name	**block_flux**

Apply

Read the results file as before, but this time select the results file, **nr0.nrf.01**, under the new job name subdirectory, **block_flux**. It is not necessary to select the results template file a second time.

The result should look like the following figure.



13. Create heating for nodal sources in the model.

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Heating (PThermal)**
Type:	**Element Uniform**
Option:	**Template, Nodal Source**
New Set Name	**node_sources**

Input Data...

Nodal Source	**1000**

OK

Select Application Region...

◆ ***FEM***

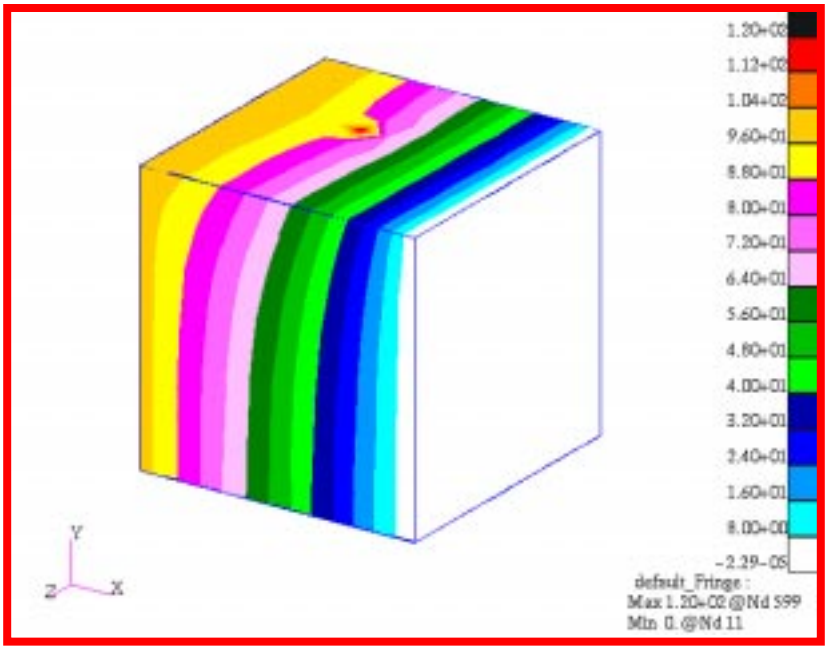
Select nodes	**select 2 nodes from interior**

Add

OK

Apply

14. Analyze the model and view the results (newest result case). Remember to change the job name. The result will depend on which nodes are chosen for the location of the nodal sources, but will have a relationship to the following plot. After viewing results reset graphic.



15. Add radiation to the right side of model and modify temperature of **Node 9999** to be -100.0^{o}C. The coupled nodes should be removed from the default load case.

◆ ***Load Cases***

Action:	**Modify**
Select Load Case To Modify:	**Default**
Assigned Loads/BCs:	**coupled_face**

Remove Selected Rows

OK

Apply

◆ *Loads/BCs*

Action:	**Create**
Object:	**Radiation (PThermal)**
Type:	**Element Uniform**
Option:	**Gap Radiation**
New Set Name	**radiation**
Target Element Type:	**3D**
Region2:	**Nodal**

Input Data...

Application Surf Emissivity	**0.8**
Coupling Surf Emissivity	**1.0**
Form Factor	**1.0**

OK

Select Application Region...

◆ *FEM*

Coupling Method	**Closest Approach**
Application Region	
Select 3D Element Faces	**select right side of block**

Add

Companion Region

Select Nodes	**Node 9999**

Add

OK

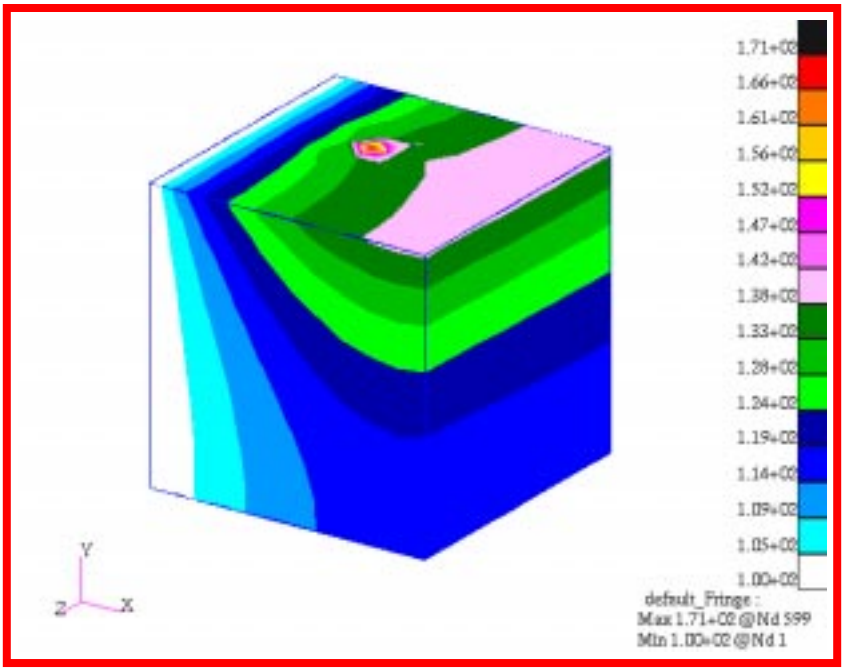
Apply

Thermal Block

Modify the temperature of the **fixed_node** to be -100^{o}C.

Action:	**Modify**
Object:	**Temperature (PThermal)**
Type:	**Nodal**
Option:	**Fixed**
Select Set To Modify:	**fixed_node**
Modify Data...	
Fixed Temperature	**-100**
OK	
Apply	

16. Analyze the model and view the results (newest result case). Remember to change the job name. After viewing results reset graphic.



17. Add convection to the bottom of the model. Create a node below the bottom of model and give it a temperature of $0^{o}C$.

◆ ***Finite Elements***

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List	**20000**

☐ ***Associate with Geometry***

Node Location List	**[0.5 -0.5 0.5]**

Apply

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Temperature (PThermal)**
Type:	**Nodal**
Option:	**Fixed**
New Set Name	**conv_temp**

Input Data...

Fixed Temperature	**0.0**

OK

Select Application Region...

◆ ***FEM***

Select Nodes	**Node 20000**

Add

OK

Apply

Thermal Block

Set the convection condition between the **conv_temp** and the bottom of the solid.

Action:	**Create**
Object:	**Convection (PThermal)**
Type:	**Element Uniform**
Option:	**Fixed Coefficient**
New Set Name	**convection**
Target Element Type:	**3D**
Region2:	**Nodal**

Input Data...

Convection Coefficient	**1000**

OK

Select Application Region...

◆***FEM***

Coupling Method	**Closest Approach**
Application Region	
Select 3D Element Faces	**select bottom side of block**

Add

Coupling Region

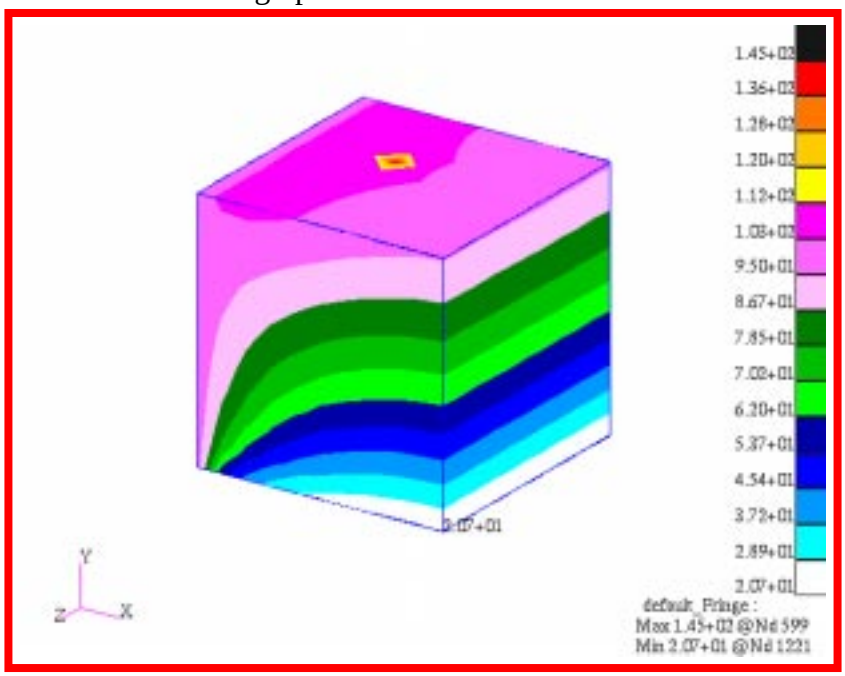
Select Nodes	**Node 20000**

Add

OK

Apply

18. Analyze the model and view the results (newest result case). Change the job name. After viewing the results reset the graphics.



19. To complete the exercise, close the database.

File/Quit...