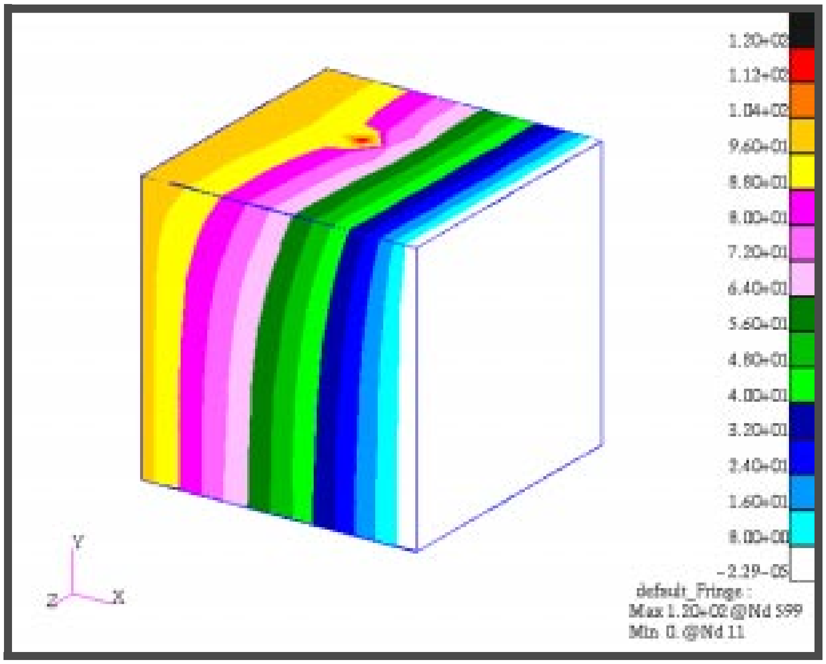
Supplementary Exercise - 7

MSC.Nastran Thermal Block

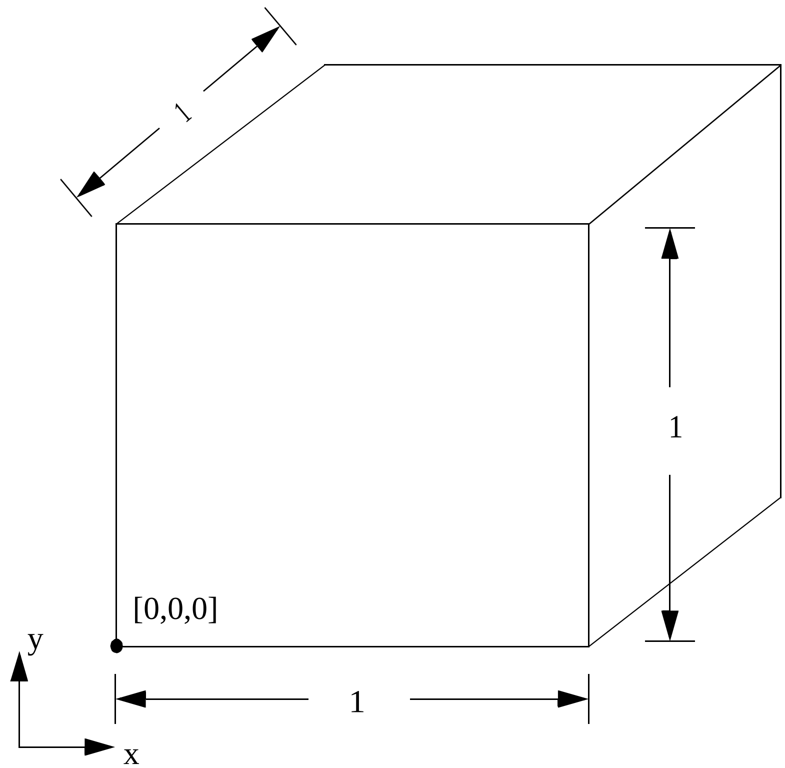


Objectives:

- Model various thermal conditions on a simple block model. This involves adding various boundary conditions and performing an analysis for each addition
  - Temperature
  - Applied heat, normal flux
  - Applied heat, nodal source
  - Radiation
  - Convection

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.Nastran 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.Nastran 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 and 0 ^{o}C on left and right face of the block model, respectively.
- Analyze the model and read the results. View results using Quick Plot.
- Add a constant heat flux called **constant_flux** to the model and re-analyze.
- Add a couple of nodal heat sources to the model and analyze.
- Create a node(Node 9999) off the right face of the block. Set the temperature there to -100 ^{o}C. Add radiation from the right face to that node. 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 and added or removed for a simple model.

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.Nastran**.

New Model Preference

Tolerance	◆ **Default**
Analysis Code:	**MSC.Nastran**
Analysis Type	**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

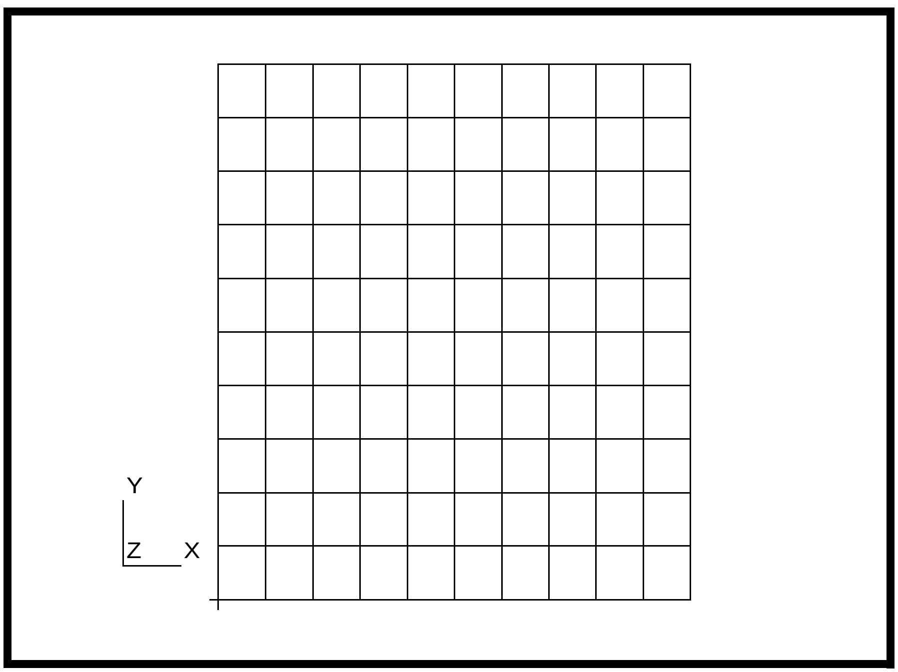
Now create the mesh for the model.

◆ ***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.



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	
Specific Heat	
OK	
Apply	

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

Create Property

◆ ***Properties***

Action:	**Create**
Object:	**3D**
Type:	**Solid**
Property Set Name	**block**
Input Properties...	
Material name	**m:alum**
OK	
Select Members	**Solid 1**
Add	
Apply	

6. Create a fixed temperature on the left and right face of the block model. Set the temperatures there to 100^{o}C and 0 ^{o}C, respectively.

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Temp (Thermal)**
Type:	**Nodal**
New Set Name	**fixed_left**
Input Data...	
Boundary Temperature	**100**
OK	
Select Application Region...	
◆ ***Geometry***	
Select Geometry Entities	**Select left face of solid (Solid 1.1)**
Add	
OK	
Apply	

Action:	**Create**
Object:	**Temp (Thermal)**
Type:	**Nodal**
New Set Name	**fixed_right**
Input Data...	
Boundary Temperature	**0.0**
OK	
Select Application Region...	
◆ ***Geometry***	
Select Geometry Entities	**Select right face of solid (Solid 1.2)**

Add

OK

Apply

7. Analyze the model

◆ ***Analysis***

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Translation Parameters...	
Data Output:	**OP2 and Print**
OUTPUT2 Format:	**Text**
OK	
Solution Type...	
Solution Type:	**Steady State Analysis**
OK	
Subcase Create...	
Available Subcases:	**Default**
Subcase Options:	**Output Requests...**
Select Result Type:	**Heat Fluxes**
OK	
Apply	
Cancel	
Subcase Select...	
Subcases Selected	**Default**
OK	
Apply	

8. Read the results.

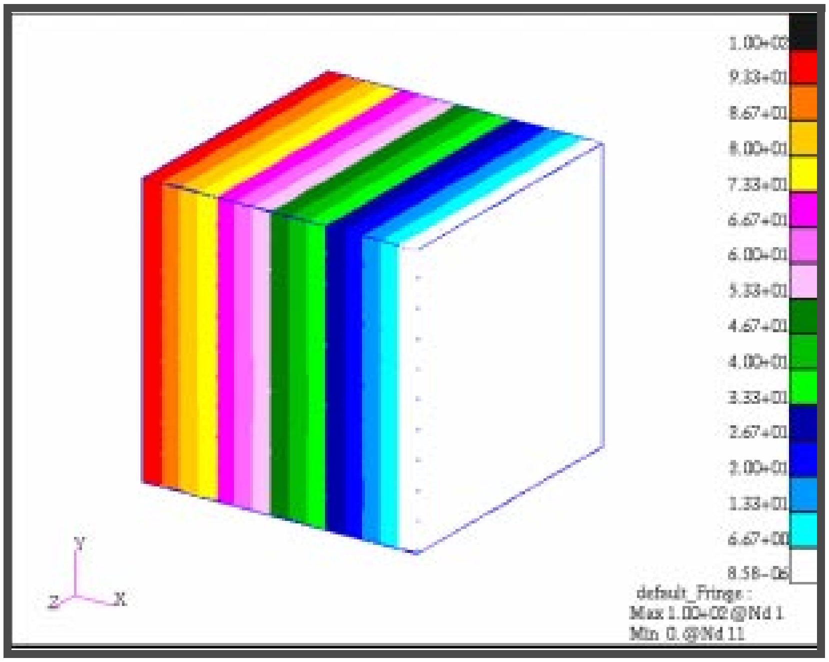
Action:	**Read Output2**
Object:	**Result Entities**
Select Results File...	
Select Results File:	**block.op2**
OK	
Apply	

9. In Results, use Quick Plot and choose temperature as your fringe result type.

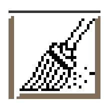
◆ ***Results***

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases:	**Default, PW Linear:**
Select Fringe Result:	**Temperature**
Apply	

The figure obtained should look like the following one. This thermal fringe is for a one dimensional heat flow that goes from 100 ^{o}C. to 0 ^{o}C.



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:	**Applied Heat**
Type:	**Element Uniform**
Option:	**Normal Fluxes**
New Set Name	**constant_flux**
Target Element Type:	**3D**
Input Data...	
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 boundary conditions that are being used in the load case named **Default**..

◆ ***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 the graphics.

Change the job name for this analysis.

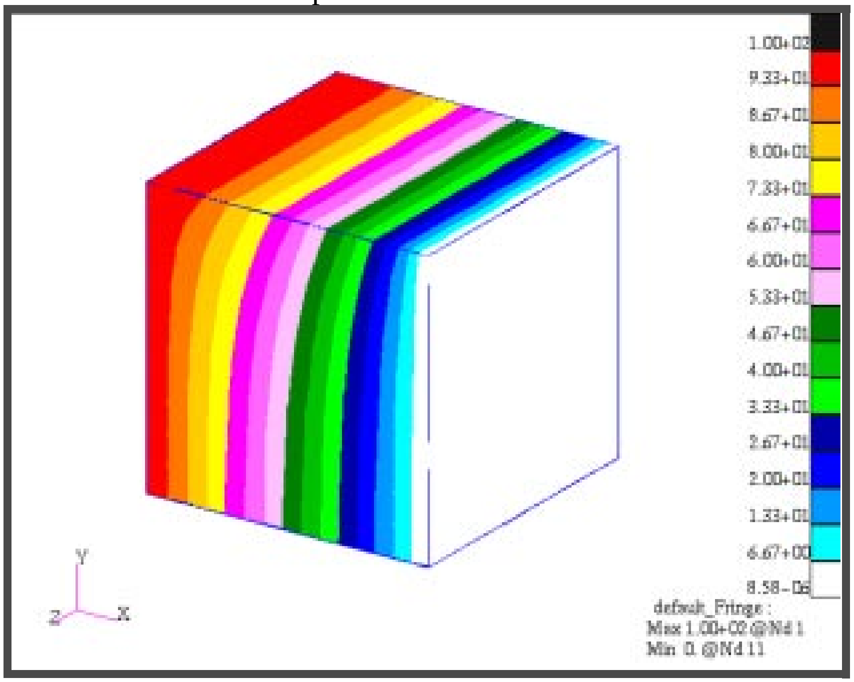
◆ ***Analysis***

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

Apply

Read the results file as before, except read the file named block_flux.op2.

The result should look like the following figure. The thermal fringe now corresponds to a two dimensional heat flow. Notice that the hotest area on the top of the model has extended in the +X direction.



13. Create heating for nodal sources in the model.

◆ ***Loads/BCs***

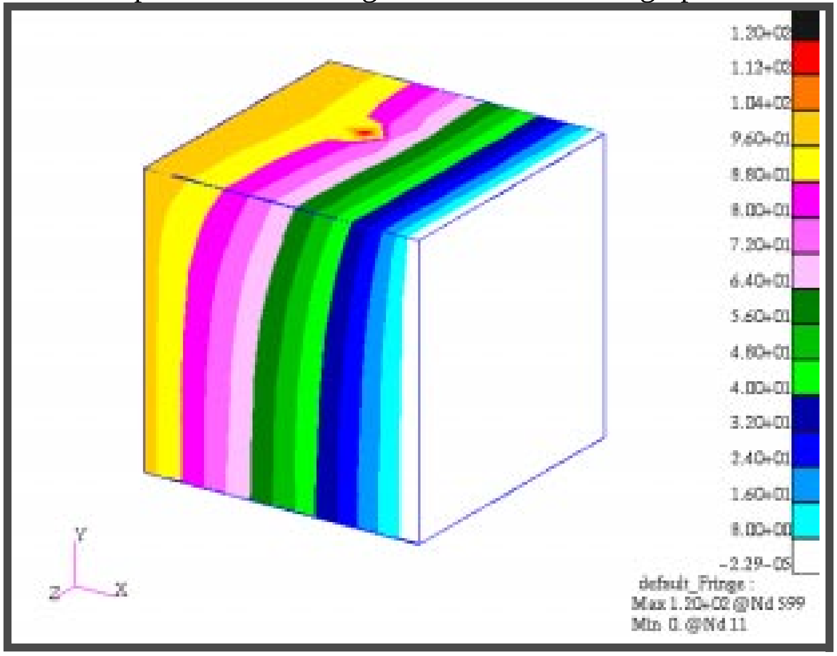
Action:	**Create**
Object:	**Applied Heat**
Type:	**Nodal**
Option:	**Nodal Source**
New Set Name	**node_sources**
Input Data...	
Nodal Source	**1000**
OK	
Select Application Region...	
◆ ***FEM***	
Select nodes	**Select 2 nodes from interior**

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 the results reset the graphics.



15. Add Node 9999 just off the right side(face) of the model. This will be radiated to by creating a radiation Loads/BCs from the right side to it. The temperature at the node will be set to -100 ^{o}C.

◆ ***Finite Elements***

Action:	**Create**
Object:	**Node**
Type:	**Edit**
Node ID List	**9999**

☐ ***Associate with Geometry***

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

Set the temperature at Node 9999 to -100^{o}C.

◆ ***Loads/BCs***

Action:	**Create**
Object:	**Temp (Thermal)**
Type:	**Nodal**
New Set Name:	**fixed_node**
Input Data...	
Boundary Temperature	**-100**
OK	
Select Application Region...	
◆ **FEM**	
Select Nodes	**Node 9999**
Add	
OK	
Apply	

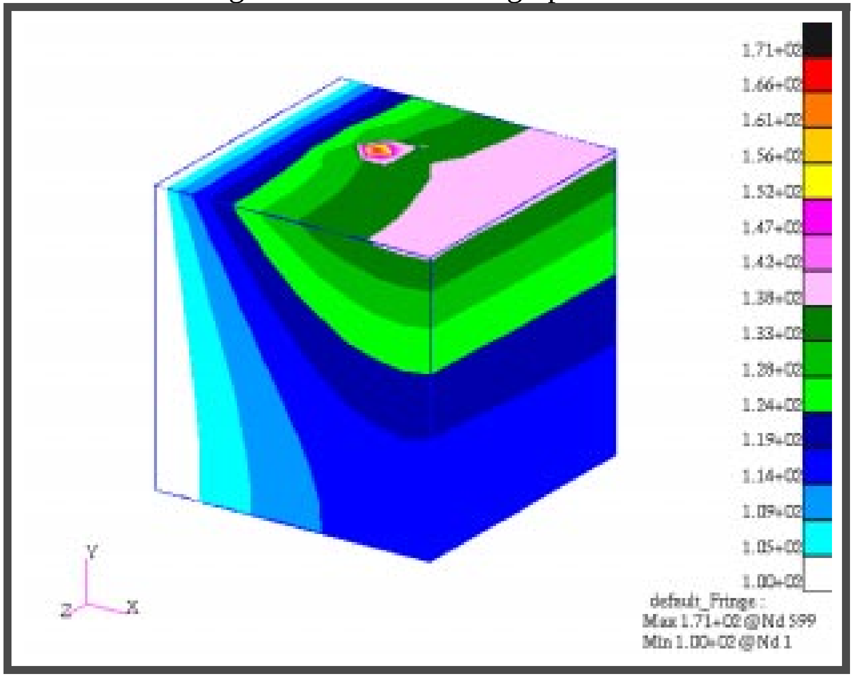
Action:	**Create**
Object:	**Radiation**
Type:	**Element Uniform**
Option:	**Ambient Nodes**
New Set Name	**radiation**
Target Element Type:	**3D**
Region2:	**Nodal**

Input Data...	
Emissivity	**0.8**
Absorbtivity	**1.0**
View Factor	**1.0**
OK	
Select Application Region...	
◆ **FEM**	
Order	**Closest Approach**
Application Region	
Select 3D Element Faces	**Select right side of block**
Add	
Active List(bottom one)	
Select Nodes	**Node 9999**
Add	
OK	
Apply	

◆ ***Load Cases***

Action:	**Modify**
Select Load Case To Modify:	**Default**
Assigned Loads/BCs:	**fixed_right**
Remove Selected Rows	
OK	
Apply	

16. Analyze the model and view the results using the latest results file. Remember to change the job name. After viewing the results reset the graphics.



17. Add convection to the bottom of the model.

Action:	**Create**
Object:	**Convection**
Type:	**Element Uniform**
Option:	**To Ambient**
New Set Name	**convection**
Target Element Type:	**3D**
Input Data...	
Convection Coefficient	**1000**
Ambient Temperature	**0.0**
OK	
Select Application Region...	

◆ ***FEM***

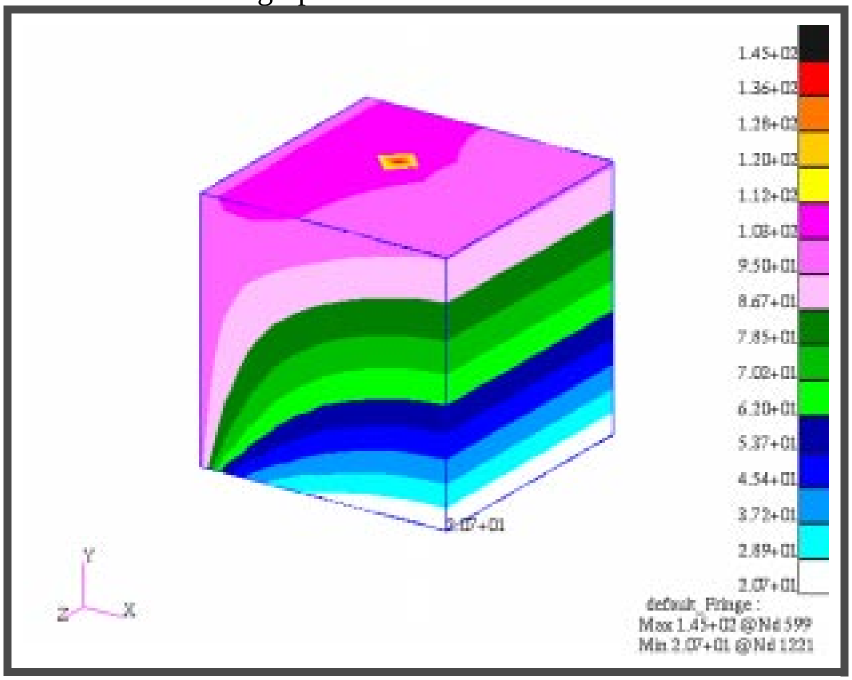
Select 3D Element Faces **Select bottom of block**

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. Complete the exercise by closing the database.

File/Quit...