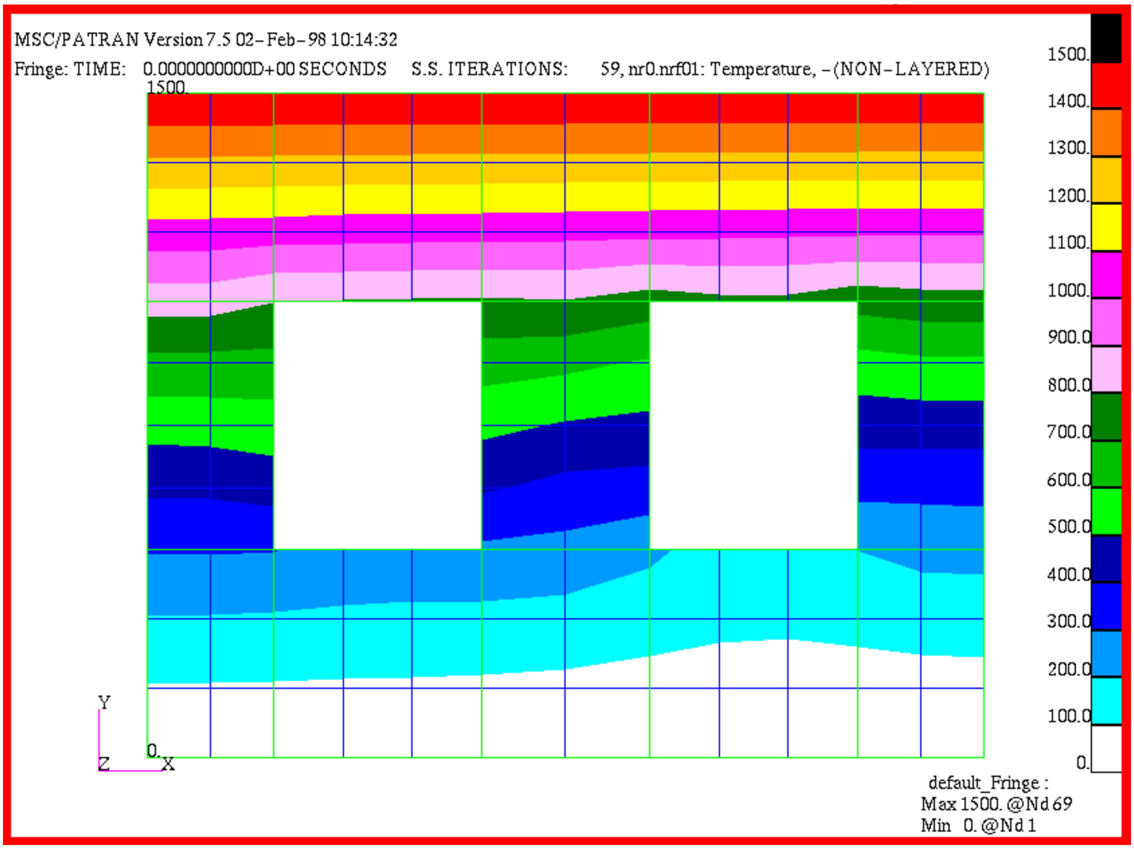
Exercise 22

Steady State Radiative Boundary Conditions

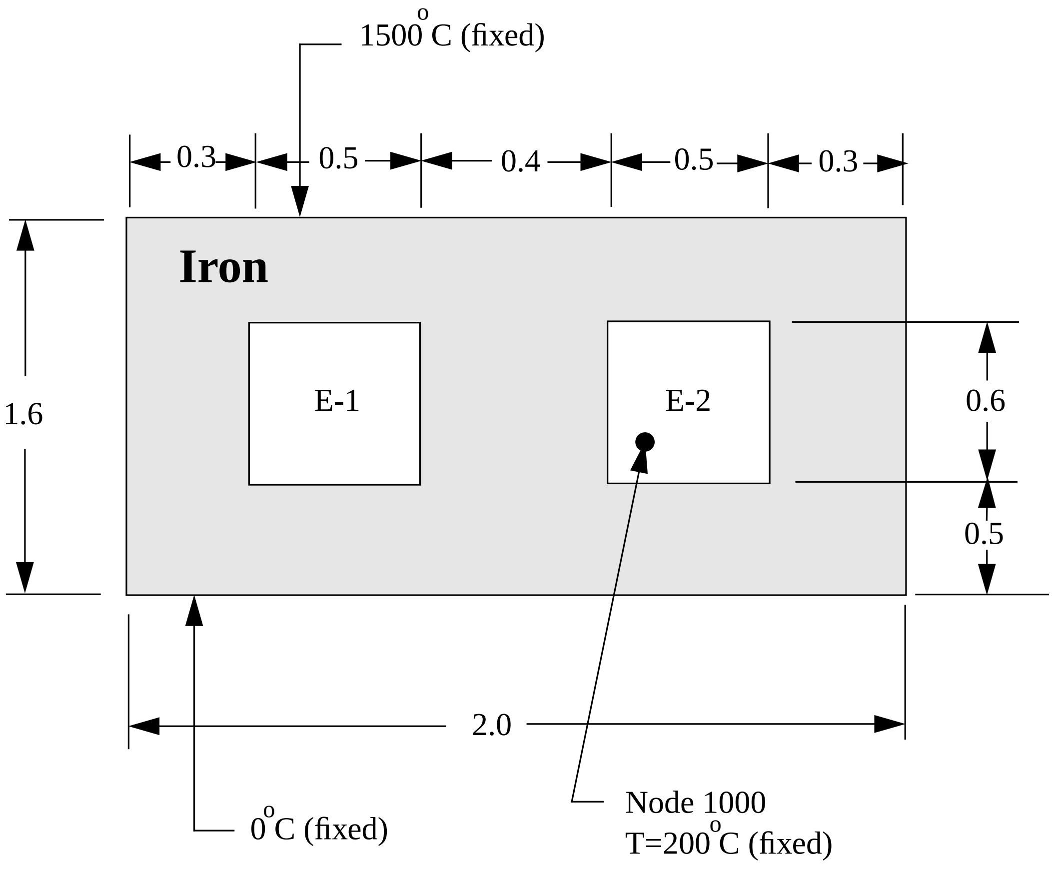


Objectives:

- Create a 2D model that incorporates two enclosures.
- Define separate radiative boundary conditions for gray body and wave length dependent radiation within the enclosures.
- Perform the Steady State thermal analysis and post process the analysis results with MSC/PATRAN's Result and Insight tools.

Model Description:

In this exercise you will construct a model with two separate radiation enclosures, one for gray body radiation and the other for wave length dependent radiation. No material (e.g. air) will be defined in the enclosure therefore only Radiation heat transfer can transfer heat energy across the enclosures. In the enclosure where it is assumed that the surfaces are gray the emissivity will be constant regardless of the surface temperatures. The other enclosure will incorporate wave length dependent radiation which is a significant extension of the gray body theory. Normal radiosity is divided into discrete frequency bands with emissivity and transmissivity assumed to be constant within these frequency bands.



Enclosure Emissivity Information:

Enclosure 1	Gray		$\varepsilon = 0.9$
Enclosure 2	For:	$0.0 \le \lambda \le 5.0$	$\varepsilon(\lambda)=0.9$ $\tau=0.4$
		$5.0 < \lambda \le \infty$	$\varepsilon(\lambda)=0.2$ $\tau=0.4$

Exercise Overview:

- Create a new database named **exercise_22.db.** Set *Tolerance* to **Default,** and the ***Analysis Code*** to **MSC/THERMAL**.
- Create a plate geometry.
- Mesh the surface with an IsoMesh of quad4 elements, global edge length of 0.16666.
- Equivalence nodes to eliminate duplicate nodes and eliminate “cracks” in the mesh.
- Create a fixed temperature boundary nodes.
- Apply Temperature boundary conditions.
- Apply View Factor boundary conditions.
- Define the Element Properties for the models Iron material.
- Prepare and submit the model for analysis**.**
- Read and plot the results**.**
- Create Temperature and Insight Contours**.**
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_22.db**.

Within your window environment change directories to a convenient working directory. Run MSC/PATRAN by typing **p3** in your xterm window.

Next, select **File** from the *Top Menu Bar* and select **New…** from the drop-down menu. Assign the name exercise_23.db to the new database by clicking in the *New Database Name* box and entering **exercise_22**.

Select **OK** to create the new database.

File	
New	
New Database Name	**exercise_22**
OK	

MSC/PATRAN will open a Viewport and change various *Control Panel* selections from a ghosted appearance to a bold format. When the *New Model Preferences* form appears on your screen, set the *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences form.

Tolerance	◆ **Default**
Analysis Code	**MSC/THERMAL**

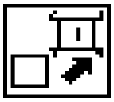
OK

Create plate geometry

2. Create a plate geometry.

Select the **Geometry** *Applications Radio Button*. Create a surface using the following *Action*, *Object*, and *Method*. Click in the appropriate list boxes to edit the default values and change them to values listed below.

First, turn on the labels using the Tool Bar *Show Label* icon.



◆ Geometry

Create/Surface/XYZ

Vector Coordinate List	**<0.3 0.5 0>**

Apply

Vector Coordinate List	**<0.5 0.5 0>**
Origin Coordinates List	**Point 4**

Apply

Vector Coordinate List	**<0.4 0.5 0>**
Origin Coordinates List	**Point 6**

Apply

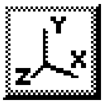
Transform/Surface/Mirror

Define Mirror Plane Normal	**Coord 0.1**
Offset Parameters	**1.0**

☐ **Auto Execute (off)**

Surface List	**Surface 1:2**
Apply	
Define Mirror Plane Normal	**Coord 0.2**
Offset Parameters	**0.8**
Surface List	**Surface 1:5**
Apply	
Create/Surface/Curve	
Starting Curve list	**Surface 1.2**
Ending Curve List	**Surface 6.3**
Apply	
Starting Curve list	**Surface 3.2**
Ending Curve List	**Surface 8.3**
Apply	
Starting Curve list	**Surface 9.2**
Ending Curve List	**Surface 4.3**
Apply	

Since this is a 2D model using radiation, check surface normal to verify that they are all in the +Z direction. Change to *Iso 1 view* using the Tool Bar *Iso 1 View* icon.



◆ **Geometry**

Show/Surface/Normal	
Surface List	**<select all surfaces>**
Apply	

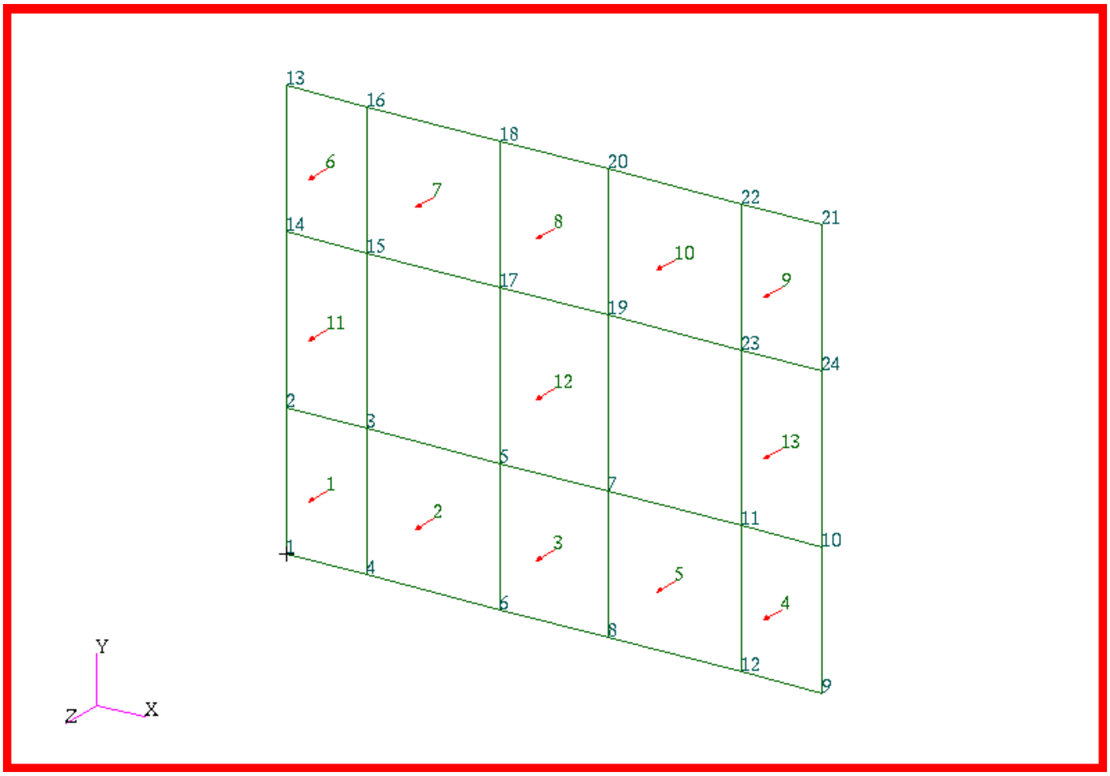
If there are any surface that is pointing the -Z direction, change them with the following steps.

Edit/Surface/Reverse

Surface List **<select any surface(s) that needs to be reversed>**

Apply

The resulting model is shown below.



IsoMesh the surfaces

3. Mesh the surface with an IsoMesh of quad4 elements, global edge length of 0.16666.

Select the **Finite Elements** *Applications Radio Button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to 0.16666 and select Surface 1 for inclusion in the *Surface List*.

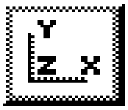
◆ Finite Elements

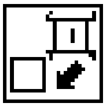
Create/Mesh/Surface

Global Edge Length	**0.16666**
Surface List	**<Surface 1:13>**

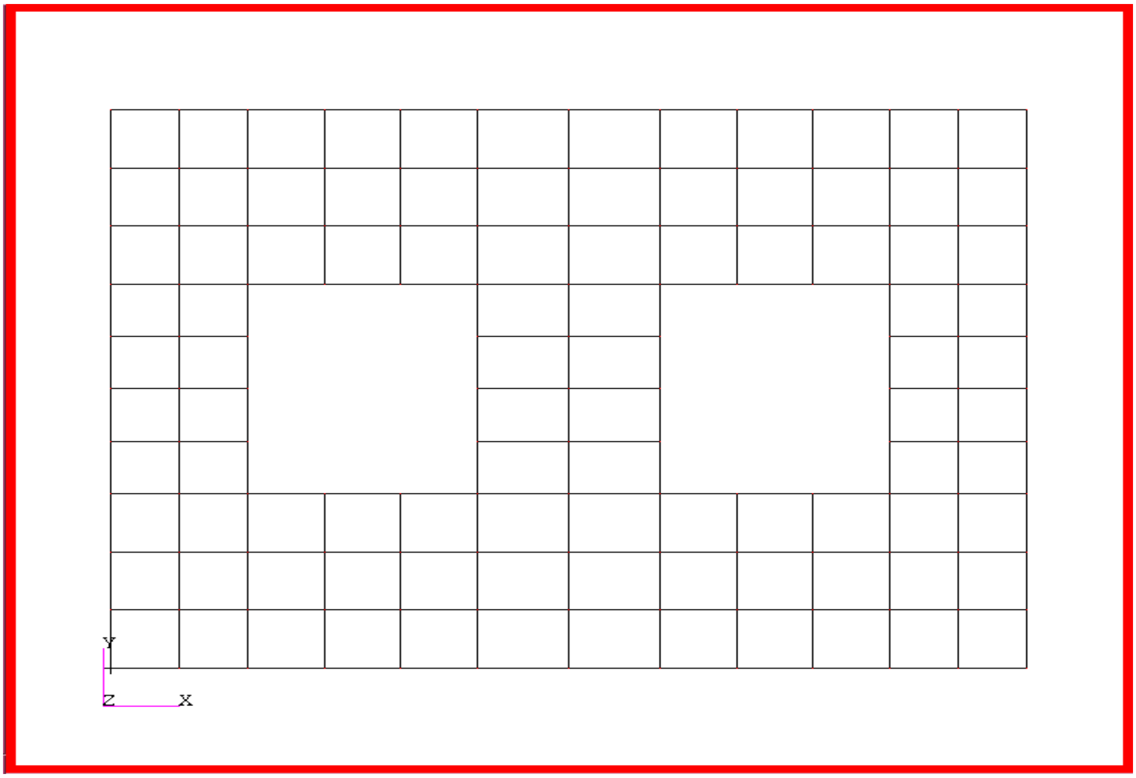
Apply

Return to the Front View using the Tool Bar *Front View* icon and turn off the labels with the *Hide Labels* icon.





The display should now appear as shown below.



Equivalence mesh nodes

4. 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 equivalencies with **Verify/Element/Boundaries** to verify the free edges.

Create a boundary nodes

5. Create a fixed temperature boundary nodes.

Select the **Finite Elements** *Applications radio button*. Create a node which is not associated with geometry. The node is numbered **1000**.

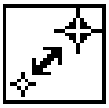
◆ **Finite Elements**

Create/Node/Edit

Node ID List	**1000**
☐ **Associate with Geometry**	
Node Location List	**[1.365 0.836 0.00]**

Apply

Increase the node size by using the Tool Bar *Node Size* icon.



Apply temperature boundary conditions

6. Apply Temperature boundary conditions.

First, create a node that will represent the Participating Medium temperature.

◆ **Load/BCs**

Create/Temperature/Nodal

Option:	**Fixed**
New Set Name	**Temp_Part_Med**

Input Data...

Fixed Temperature | **200**

OK

Select Application Region...

◆ **FEM**

Select Node | **Node 1000**

Add

OK

Apply

Next, assign fixed temperatures of **1500˚C** and **0˚C** respectively to the top and bottom geometry edges of the model. Use **T_top** and **T_bottom** for their respective *New Set Names*.

New Set Name | **T_top**

Input Data...

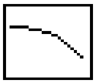
Fixed Temperature | **1500**

OK

Select Application Region...

◆ **Geometry**

Select Geometry Entities/*Select Menu* | **<select *Curve or Edge* icon>**



Select Geometry Entities | **<drag a box around the top edge of the Entity>**

Add

OK

Apply

New Set Name | **T_bottom**

Input Data...

Fixed Temperature | **0**

OK

Select Application Region...

Select Geometry Entities — **<drag a box around the bottom edge of the Entity>**

Add

OK

Apply

Apply View Factor boundary conditions

7. Apply View Factor boundary conditions.

To create the view factor boundary conditions for the two enclosures you will first supply geometric information in the P3/PATRAN Load/BCs form and then enter data concerning the Emissivity and Transmissivity values in the **template.dat.apnd** file.

In the *Load/Boundary Conditions* form, change the *Action*, *Object*, and *Type* option menus respectively to **Create**/**Radiation/Element Uniform**. Change the *Target Element Type* to **2D**.

◆ **Load/BCs**

Create/Radiation/Element Uniform

Option:	**View Factors**
New Set Name	**Encl_101**
Target Element Type:	**2D**

Input Data...

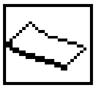
Vfac Template ID	**100**
Enclosure ID	**1**

OK

Select Application Region...

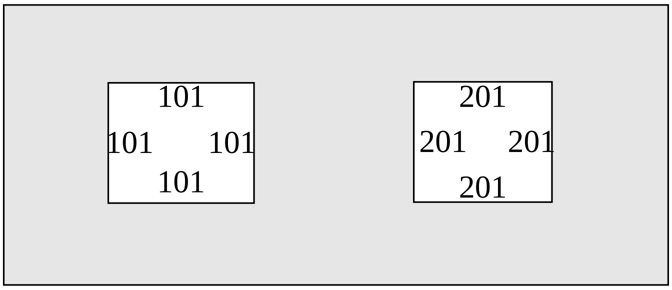
◆ **Geometry**

Select Surfaces s or Edges /*Select Menu* — **<select *Edge* icon>**



Select Surfaces or Edges	**<refer to the diagram below and drag a box around the interior surfaces corresponding to 101>**
Add	
OK	
Apply	
New Set Name	**Encl_201**
Input Data...	
Vfac Template ID	**200**
Participating Media Node ID	**1000**
Enclosure ID	**2**
OK	
Select Application Region...	
Select Surfaces or Edges	**<refer to the diagram below and drag a box around the interior surfaces corresponding to 201>**
Add	
OK	
Apply	

Use the diagram below to determine the required geometric information for the two enclosures.



You will now complete the View Factor definitions by entering the Emissivity and Transmissivity information into the *template.dat* file. Create a separate x-window shell in the directory you are running P3/PATRAN and edit the file named **template.dat.apnd**. Next, enter the required VFAC commands to define the Emissivity and Transmissivity for Enclosures 1 and 2. The syntax of the command is,

VFAC TID NBANDS
ε τ ε_{id} τ_{id} λ_1 λ_2 K-flag

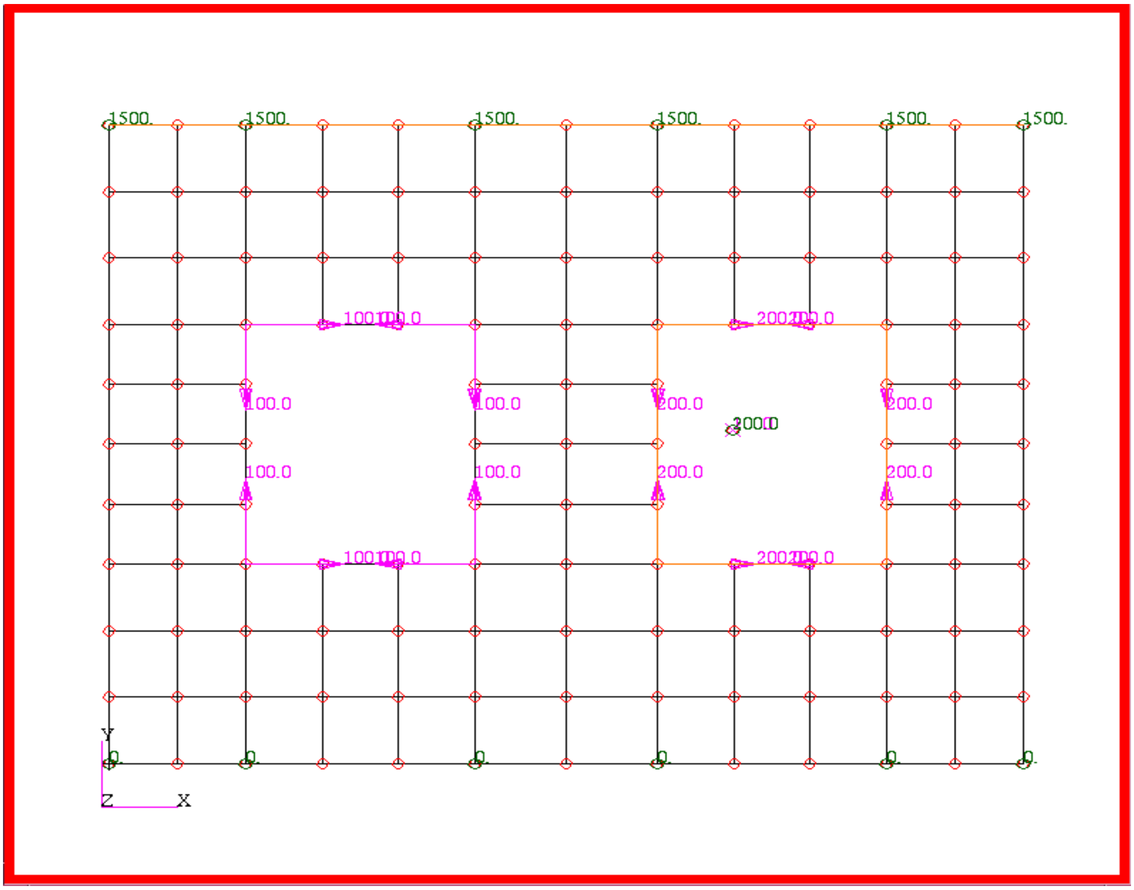
Each term of the command is defined in the P/THERMAL Users Manual. Shown below is a Table that lists the required information for the two VFAC commands and the template.dat.apnd file created with this information for your reference.

TID	NBANDS	ε	τ	ε_{id}	τ_{id}	λ_1	λ_2	K flag
100	0	0.9	-	-	-	-	-	-
200	2	0.9	0.4	0	0	0.0	5.0	0
		0.2	0.4	0	0	5.0	1E6	0

The text in the **template.dat.apnd** file should be as follows.

***Information for enclosure 1**
VFAC 100 0
0.9
***Information for enclosure 2**
VFAC 200 2
0.9 0.4 0 0 0.0 5.0
0.2 0.4 0 0 5.0 1.0E6

Your model with its applied boundary conditions should now look like the one shown below.



Define Element Properties

8. Define the Element Properties for the models Iron material.

To do this click on the **Element Props** toggle in the *Main Window*. When the form appears set its *Action*, *Dimension*, and *Type* option menus respectively to **Create**, **2D**, and **Thermal 2D**. Enter **Iron**, for the *New Set Name* and then click on the **Input Properties…** button. Enter **18** in the *Material Name* databox and then click on the **OK** button to close the form. Next, click in the *Select Members* box and select all the models surfaces in the viewport. Finally click on the **Apply** button in the *Element Properties* form.

◆ Properties

Create/2D/Thermal 2D

Property Set Name **Iron**

Input Properties...

Material Name **18**

OK

Select Members **<Select all entities, (Surface 1:13)>**

Add

Apply

9. Prepare and submit the model for analysis.

Prepare and run analysis

Select the **Analysis** *Applications Radio Button* to prepare the analysis. Select the parameter forms reviewing and changing the settings as shown below. The analysis is submitted by selecting **Apply** in the Analysis form.

◆ Analysis

Analyze/Full Model/Full Run

Translation Parameters...

Model Dimensionality **◆ 2D Plane Geometry, X Y Co-ord**

OK

Solution Type... **◆ Perform Viewfactor Analysis**

OK

Solution Parameters...

Calculation Temperature Scale **◆ Celsius**

OK

Output Requests...

Units Scale for Output Temperatures **◆ Celsius**

OK

Submit Options...

Create Viewfactor Control File **(if not already selected)**

Execute Viewfactor Analysis **(if not already selected)**

OK

Apply

Read and plot results

10. Read and plot the results.

From within MCS/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.

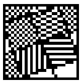
P3 was initiated from a working directory which contained the exercise_22.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*, exercise_22. By using **Read Result** in the Analysis form and Selecting **Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of a results file.

◆ Analysis	
Read Results/Result Entities	
Select Results File...	
Directories	**<path>/exercise_22**
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*.

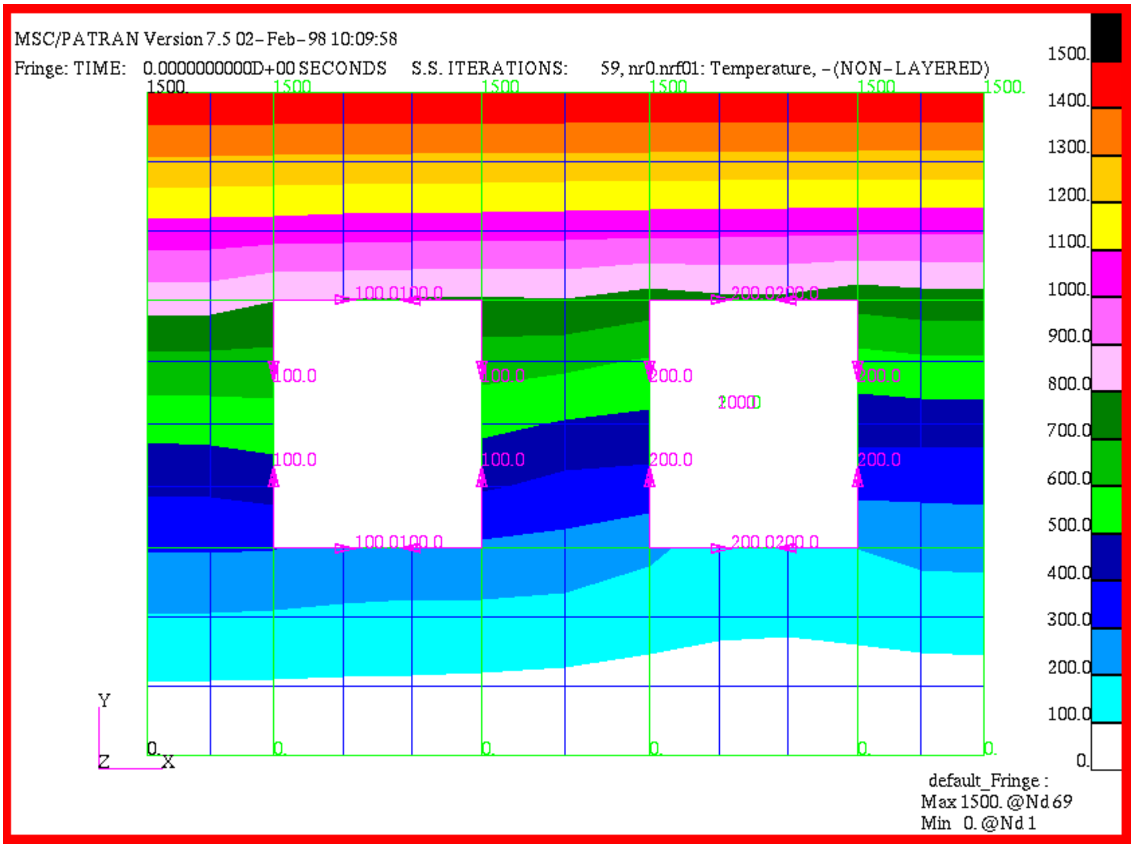
◆ Results	
Create/Quick Plot	
Select Result Cases	**TIME: 0.0000000000D+00 S...**
Select Fringe Result	**Temperature,**
Apply	

Select the *Fringe Attributes* icon.



Display:	**Element Edges**
Label Style...	
Label Format:	**Fixed**
Significant figures	**4 <use slider bar>**
OK	
Apply	

The model should now appear as shown below.



As expected the temperature distribution is not horizontally symmetrical due to the different radiation boundary conditions in each enclosure.

Create Contours

11. Create Temperature and Insight Contours.

To do this click on the **Insight** toggle in the *Main Window*. When the *Insight Imaging* form appears set the *Action* and *Tool*, to **Create** and **Contour** respectively. Click on the **Results Selection…** button and select **1.1 Temperature, (nodal)** from the *Contour Results List Box*. Click on the **OK** button to close the from. Click on the **Apply** to create the Temperature Contours.

◆ **Insight**

Create/Contour

Results Selection...

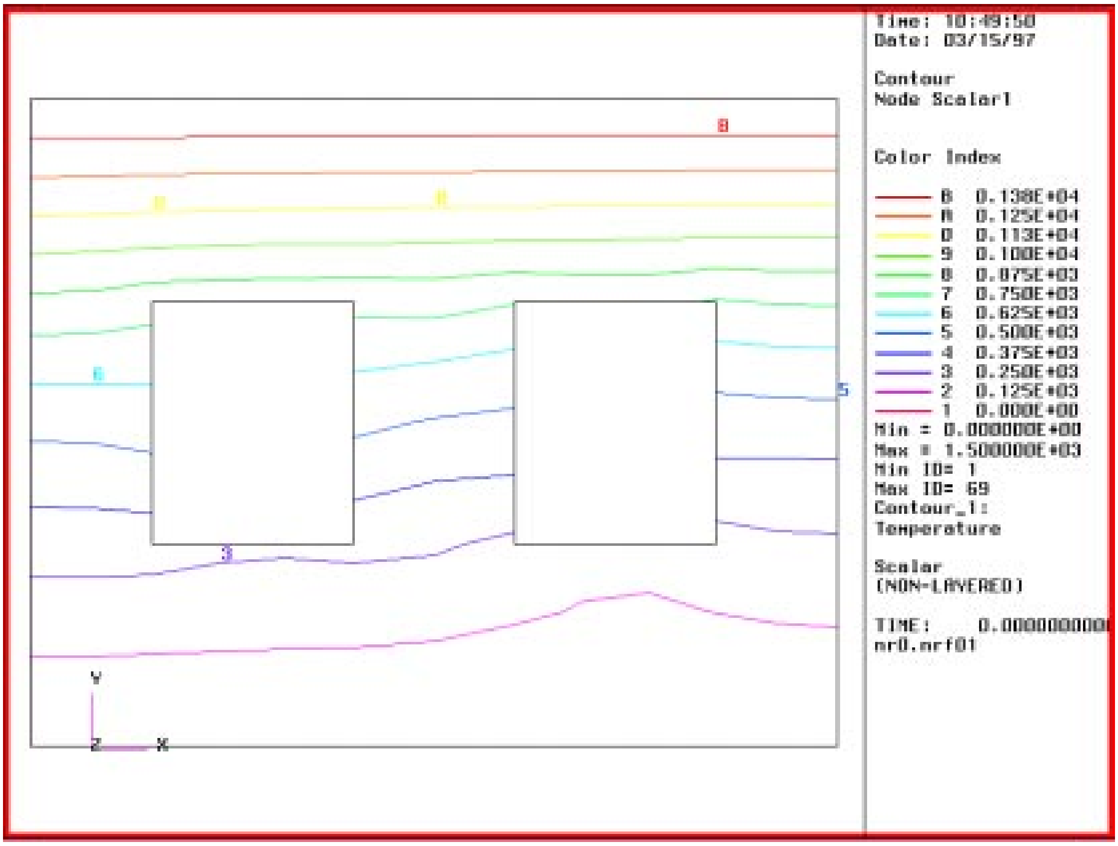
Update Results

Contour Result **1.1-Temperature,**

OK

Apply

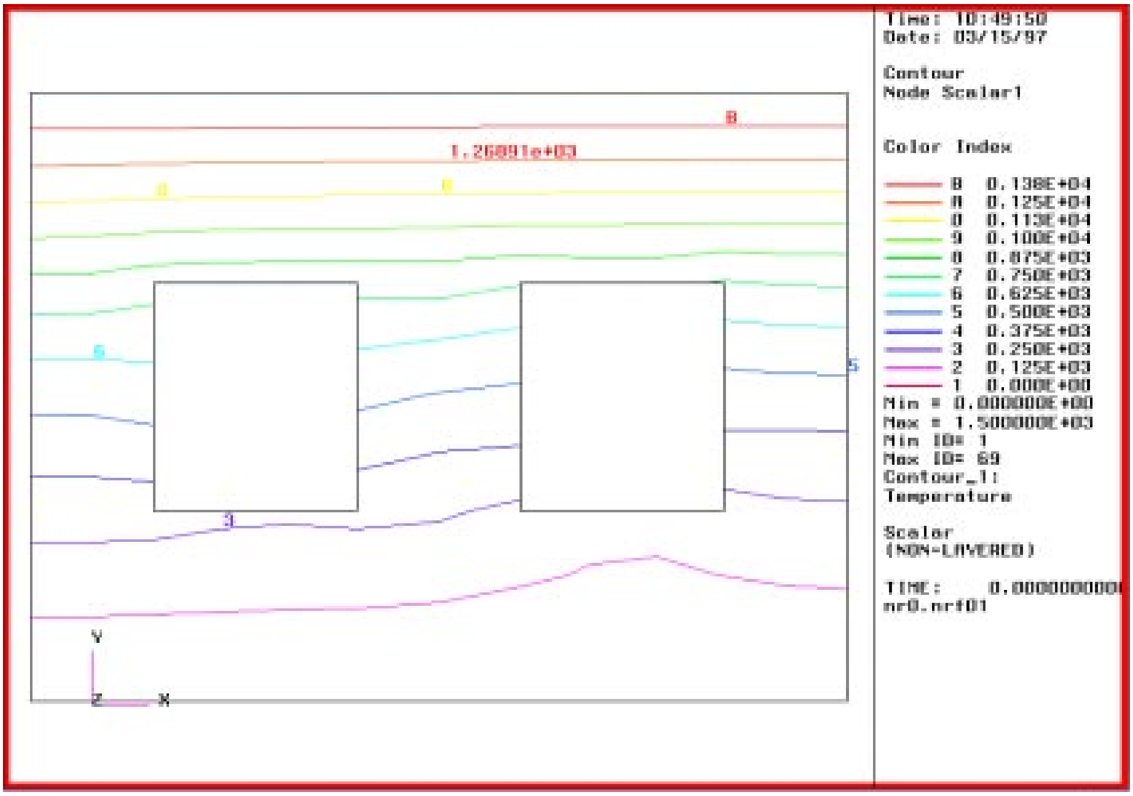
Your model should now look like the one shown below.



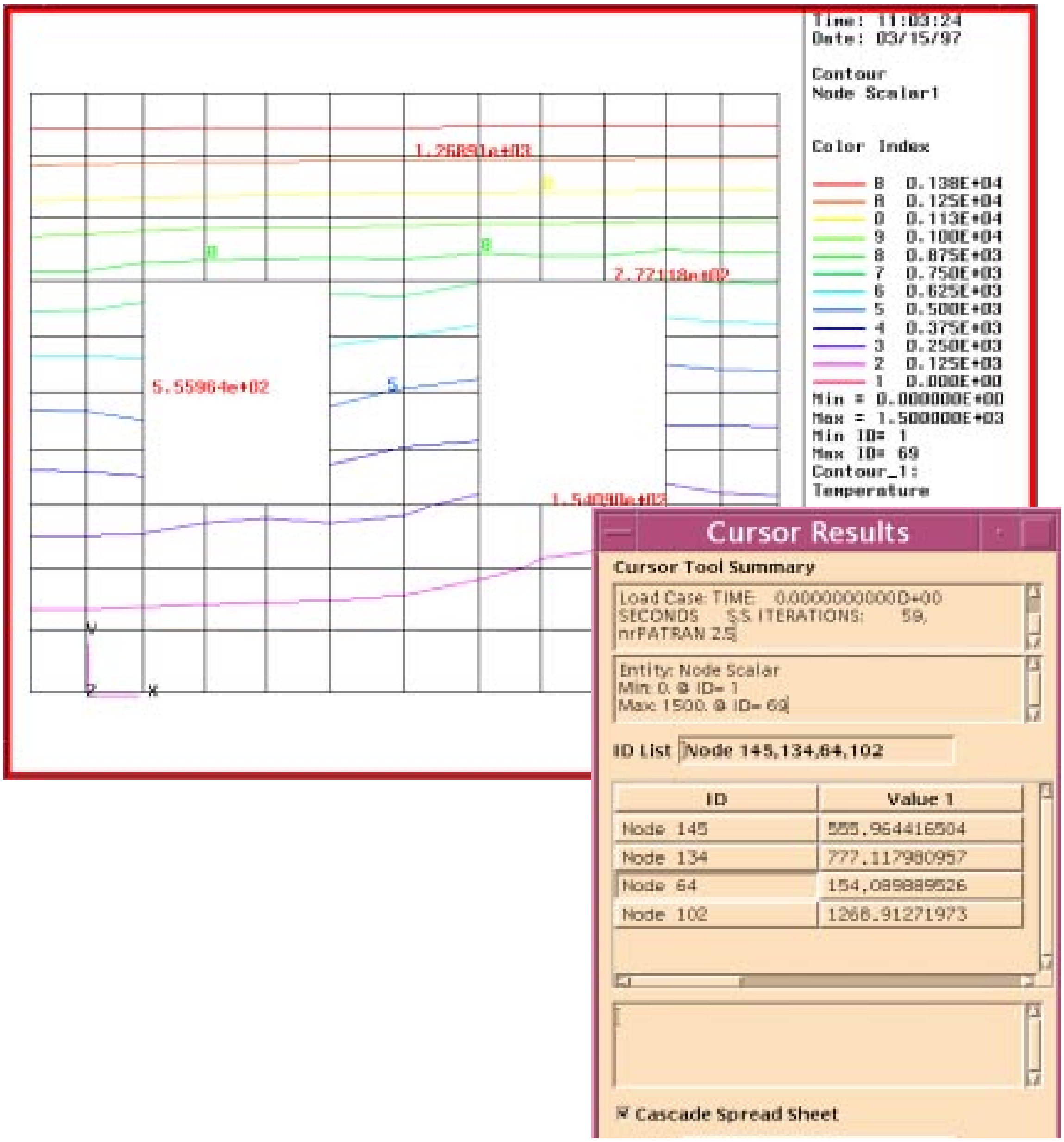
To create Cursor tool change the *Tool* to **Cursor** and then click on the **Results Selection…** button. Again select **1.1-Temperature, (nodal)** in the *Cursor Results* list and click on **OK** to close the form. Click on the **Apply** button to create the Cursor Tool. When the *Cursor Tool* from appears click on the **Cascade Spread Sheet** button. Next, click some where on the model. You should see the temperature of the Node nearest to the mouse cursor printed on the model and in the *Cursor Results* form.

Create/Cursor	
Results Selection...	
Contour Result	**1.1-Temperature,**
OK	
Apply	
Cursor Result Form	◆ **Cascade Spread SHeet**

Your model should now look similar to the one shown below.



To obtain an indication of where the models Nodes are located click on **Preferences** in the *Main Window* and select **Insight...** from the pull-down menu. When the *Insight Preferences* form appears change the *Display Method* to **Wireframe**. Click on the **Apply** and **Cancel** buttons rerender the model and to close that from. You can now click on the element corners (where the nodes are located) and determine the specific temperature values at those nodes. An example *Cursor Results* form and its corresponding temperature locations are shown as follows, for your reference.



Quit MSC/ Patran

12. **Quit** MSC/PATRAN.

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