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 wavelength 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:  \( 0.0 \leq \lambda \leq 5.0 \)  \( \varepsilon(\lambda)=0.9 \)  \( \tau=0.4 \)

\( 5.0 < \lambda \leq \infty \)  \( \varepsilon(\lambda)=0.2 \)  \( \tau=0.4 \)
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:

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.
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.

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

Transform/Surface/Mirror

Define Mirror Plane Normal

Coord 0.1

Offset Parameters

1.0

Auto Execute (off)
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.
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.
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 Interface](image)

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.

![Display with IsoMesh](image)
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.

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

<table>
<thead>
<tr>
<th>Create/Node/Edit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Node ID List</td>
</tr>
<tr>
<td>Associate with Geometry</td>
</tr>
<tr>
<td>Node Location List</td>
</tr>
<tr>
<td>Apply</td>
</tr>
</tbody>
</table>

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

6. Apply Temperature boundary conditions.

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

** Load/BCs

<table>
<thead>
<tr>
<th>Create/Temperature/Nodal</th>
</tr>
</thead>
<tbody>
<tr>
<td>Option: Fixed</td>
</tr>
<tr>
<td>New Set Name: Temp_Part_Med</td>
</tr>
<tr>
<td>Input Data...</td>
</tr>
</tbody>
</table>
Next, assign fixed temperatures of 1500°C and 0°C respectively to the top and bottom geometry edges of the model. Use $T_{\text{top}}$ and $T_{\text{bottom}}$ for their respective New Set Names.

Fixed Temperature

<table>
<thead>
<tr>
<th>200</th>
</tr>
</thead>
</table>

Select Application Region...

◆ FEM

Select Node

Node 1000

Add
OK
Apply

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...
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:
New Set Name
Target Element Type:
Input Data...
Vfac Template ID
Enclosure ID
OK
Select Application Region...
◆ Geometry
Select Surfaces or Edges
/Select Menu

<drag a box around the bottom edge of the Entity>
```

View Factors

<table>
<thead>
<tr>
<th>Encl_101</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D</td>
</tr>
</tbody>
</table>

| 100 |
| 1    |
Apply View Factor boundary conditions

Select Surfaces or Edges

Add

OK

Apply

New Set Name

Encl_201

Input Data...

Vfac Template ID

200

Participating Media Node ID

1000

Enclosure ID

2

Select Application Region...

Select Surfaces or Edges

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,

\[
\text{VFAC TID NBANDS} \quad \varepsilon \quad \tau \quad \varepsilon_{id} \quad \tau_{id} \quad \lambda_{1} \quad \lambda_{2} \quad \text{K-flag}
\]

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.

<table>
<thead>
<tr>
<th>TID</th>
<th>NBANDS</th>
<th>$\varepsilon$</th>
<th>$\tau$</th>
<th>$\varepsilon_{id}$</th>
<th>$\tau_{id}$</th>
<th>$\lambda_{1}$</th>
<th>$\lambda_{2}$</th>
<th>K flag</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>0</td>
<td>0.9</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>200</td>
<td>2</td>
<td>0.9</td>
<td>0.4</td>
<td>0</td>
<td>0</td>
<td>0.0</td>
<td>5.0</td>
<td>0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>0.2</td>
<td>0.4</td>
<td>0</td>
<td>0</td>
<td>5.0</td>
<td>1E6</td>
<td>0</td>
</tr>
</tbody>
</table>

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.

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.
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**

9. Prepare and submit the model for 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.
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.

To plot the results to posted FEM use the Results Application radio button.
Select the *Fringe Attributes* icon.

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

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.

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.

12. **Quit MSC/PATRAN.**

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