Exercise 8

Temperature Dependent Material Properties

Objective:

- You will create a 2D material slice consisting of two materials with temperature dependent material properties.
- You will visually and qualitatively compare the MSC/THERMAL results with the results of an analytical solution.
**Model Description:**

In this exercise you will learn to create temperature dependent material properties.

There are very few analytical solutions available for composite materials with temperature dependent conductivities. Recently, K. C. Chang and V. J. Payne published an analytic solution for the problem you will analyze in this exercise (Journal of Heat Transfer, Feb. 1991, Vol. 113, pp. 237). Results of their work have been included at the end of this exercise to allow you to qualitatively compare your solution to theirs.

\[
T = 600^\circ C = 873.2 \text{ K}
\]

\[
T = 100^\circ C = 373.2 \text{ K}
\]

\[
T_a = 0^\circ C = 273.2 \text{ K}
\]

Material 1  
Material 2

0.5 X 0.5 Dimension

\[
K_1 = K_{10} (1 + \alpha_1 T)
\]

\[
K_2 = K_{20} (1 + \alpha_2 T)
\]

\[
K_{10} = 0.060 \quad \alpha_1 = 0.0006
\]

\[
K_{20} = 0.001 \quad \alpha_2 = 0.00001
\]
Exercise Overview:

- Create a new database named `exercise_08.db`. Set the Tolerance to Default, and the Analysis Code to MSC/THERMAL.
- Create two surfaces which model the two adjoining material slabs.
- Mesh the surfaces with an IsoMesh.
- Identify “cracks” in the model and Equivalence the nodes at the mating surface edges.
- Define the two materials using Fields/Create/Material Property/General.
- Using the fields just defined create Material 1 and Material 2.
- Apply element properties to the elements referencing the two material properties just defined.
- Apply the three temperature boundary conditions to the edges of your model.
- Prioritize temperature boundary conditions at the lower corners.
- Prepare and submit the model for analysis.
- Read results file and plot results.
- Compare the results to the analytical solution.
- Quit MSC/PATRAN.

Exercise Procedure:

1. Create a new database named `exercise_08.db`. Set the Tolerance to Default, and the Analysis Code to MSC/THERMAL.

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 Menu Bar Menu Bar and select New… from the drop-down menu.
Assign the name exercise_08.db to the new database by clicking in the New Database Name box and entering exercise_08 (.db will automatically be appended).

Select OK to create the new database.

PATRAN will open a Viewport and change various Main Form 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 two surfaces which model the two adjoining material slabs.

Select the Geometry Applications radio button. Set the Action, Object, and Method to Create/Surface/XYZ. Change the Vector Coordinates List to <0.5, 0.5, 0> and click on the Apply button to create the first patch.

Change the Origin Coordinates List to [0.5, 0, 0], and click on the Apply button to create the second surface.
3. Mesh the surfaces with an IsoMesh.

Select the **Finite Elements Applications radio button**. Set the **Action**, **Object**, and **Type** to **Create/Mesh/Surface**. Set the **Global Edge Length** to **0.025**. Click in the **Surface List** box and drag a rectangle around both surfaces. Select **Apply** to complete the meshing function. The completed form and resulting display are shown below.

4. Identify “cracks” in the model and equivalence the nodes at the mating surface edges.

In the **Finite Elements** form set the **Action**, **Object**, and **Test** to **Verify/Element/Boundaries**. Select **Apply**.

In the **Finite Elements** form set the **Action**, **Object**, and **Method** to **Equivalence/All/Tolerance Cube**. Select **Apply** to complete the function.
The nodes bounding the interior edges will be circled in the display and the Command Window will indicate that a number of nodes are deleted. All gaps or cracks have now been eliminated from the mesh.

5. Define the two materials using Fields/Create/Material Property/General.

Select the Fields Applications radio button. Set the Action, Object, and Method to Create/Material Property/General.

Enter a Field Name K2 and select Input Data...

In the General Field Input Data form Select Function Term mpid_arbt_plyn.

An Arbitrary Order Polynomial form will be displayed. On this form, change the Temperature Units option menu to Kelvin. Then enter Coefficient Data for Material 2 conductivity, \( (K2 = 0.001 + 0.00000001 T) \). First enter 0.001 in the Coefficient, A(Index) databox then select the Enter button, next enter 1.0E-8 and select the Enter button.
Define the material property fields

The form should appear as shown below.

Select OK in the Arbitrary Order Polynomial form. Select OK in the General Field Input Data form. Select Apply button on the Fields form to complete the function.

In the Field form change Field Name to K1. Again choose the mpid_arbt_plyn. Using the Spread Sheet Action Modify, click on the Coefficient 1 cell in the Coefficient Data frame, and edit the Coefficient,A(Index) data box to change the Arbitrary Order Polynomial form for the thermal conductivity of Material 1, (K1 = 0.06 + 0.000036T).
6. Using the fields just defined create Material 1 and Material 2.

Select the Materials Applications radio button. Set the Action, Object, and Method to Create/Isotropic/Manual Input. Enter Material_1 in the Material Name databox. Select Input Properties... In the Input Options form click into the Thermal Conductivity data box.

The form should be modified to include a Time, Temperature, or Constant Fields: list box. Select K1 from the listbox. Enter unit values for Density and Specific Heat.

Repeat the same procedure for Material_2; this time selecting K2 for Thermal Conductivity. After creating both materials select Cancel to close the Input Options form.

7. Apply element properties to the elements selecting the two material properties just defined.

Select the Properties Applications radio button. Set the Action, Dimension, and Type to Create/2D/Thermal 2D. Enter Property Set Name Prop1. Select the Input Properties... box. In the Input Properties form, click in the Material Name box and select Material_1 from the Material Properties Sets list. Select OK to close the form.

Click in the Select Members box and select Surface 1, the left surface. Select Add then Apply in the Element Properties form to complete the element property definition for Surface 1.
Perform the same steps for **Surface 2**, the right surface, using **Prop2**, for the **Property Set Name**, and **Material_2** for the **Material Name**.

8. You will now apply the three temperature boundary conditions to the edges of your model.

**Left vertical edge of Surface 1:**

Select the **Load/BCs Applications radio button**. Set the **Action**, **Object**, and **Type** to **Create/Tempature (PTermal)/Nodal** with an **Option:** of **Fixed**. Enter the name, **Mat1_Edge_Temp**, into the **New Set Name** data box.

Click on the **Input Data...** button and enter a **Fixed Temperature** of **873.2**. Click on the **OK** button to close the **Input Data** form.

Select the **Select Application Region...** button and set the **Geometry Filter to Geometry**. Click on the **Select Geometry Entities** box, select the **Curve or Edge icon** in the **Select Menu**. Select the left-hand vertical edge of Surface 1. Select **Add** then **OK** to affect and close the **Select Application Region** form. The completed forms are shown below.
Select **Apply** to create the temperature boundary condition.

Perform similar steps to assign the remaining temperature boundary conditions to your model. Use the following **New Set Name**, and **Fixed Temperature** values.

<table>
<thead>
<tr>
<th>New Set Name</th>
<th>Fixed Temp</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mat1_2_Bottom_Edge_Temp</td>
<td>273.2</td>
</tr>
<tr>
<td>Mat2_Edge_Temp</td>
<td>373.2</td>
</tr>
</tbody>
</table>

Your model should now look like the one shown below.

Applying the temperature boundary conditions to the various edges of your model created a conflict at the two lower corner points. At the lower left corner both the 873.2 and 273.2 temperature boundary conditions were applied. At the lower right corner both the 373.2 and 273.2 temperature boundary conditions were applied. By default MSC/PATRAN adds overlapping boundary conditions. To fix the lower corner temperature to 273.2 you must tell MSC/PATRAN that the boundary condition you applied to the bottom edge of the model has priority over the conflicting vertical edge boundary conditions.
9. Prioritize temperature boundary conditions at the lower corners.

Select the **Load Cases Applications radio button**. Change the **Action** to **Modify**. In the **Load Cases** form highlight the **Default** load case in the **Existing Load Cases** list box, if necessary. Select the first LBC, **Mat1_2_Bottom_Edge_Temp**, and then in the **Prioritize Load/BCs** form set the **Priority Mode** to **Overwrite**. Check that the **Mat1_2_Bottom_Edge_Temp** boundary condition has first priority. Select the **Mat_1_Edge_Temp** LBC then again set the **Priority Mode** to **Overwrite**. Repeat for the last LBC, **Mat2_Edge_Temp**. The completed forms are shown below.

Select **OK** in the **Prioritize Load/BCs** form and **Apply** in the **Load Cases** form. In **Message** box, “**Do you wish to overwrite?**”, answer **Yes**.
10. Prepare and submit the model for analysis.

Select the Analysis Applications radio button to prepare the analysis.

Select Translation Parameters... Select the 2D Plane Geometry, XY Co-ordinates (Unit Thickness in Z) radio button in Model Dimensionality. Select OK to close the P/Thermal Translation Parameters form.

Select Solution Parameters... Select the Kelvin radio button in Calculation Temperature Scale. Select OK to close the P/Thermal Solution Parameters form.

Select Output Requests... Select the Celsius radio button in Unit Scale for Output Temperatures. Select OK to close the P/Thermal Output Requests form.

Since all other defaults are acceptable submit the analysis by selecting Apply in the Analysis form.

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.

11. Read results file and plot results.

Recall that p3 was initiated from a working directory which contained the exercise_08.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 exercise_08/. By using Read Result in the Analysis form and Select Results File... you can filter down to the Job Name subdirectory and check for the existence of the results file.

Select the nr0.nrf.01 results file in the Available Files list box. Select OK.

Select the Select Rslt Template File... in the Analysis form. In the Template to Import P/Thermal Nodal Results form select the template named pthermal_1_nodal.res_tmpl from the Files list. Select OK.

Select Apply in the Analysis form to read the chosen results file with the selected template.
To plot the results use the **Results Application radio button**. The default Action/Object should be **Create/Quick Plot**. Hit **Apply** to quick plot the default Result Case and Fringe Result.

To affect a better comparison use the **Fringe Attribute** icon to change the display and range.

Select **Display: Element Edges**. Select **Label/Style...**. Under **Label/Style...** select **Label/Format: Fixed** and use the slider bar to select **4** Significant figures, then select **OK**, and **Apply**.

Select **Range.../Define Range.../Create...** Use a new **Range Name: Compare** with **Number of Sub-Ranges**: **7**. Select **OK**.

In the **Range** form select **Data Method/From**. In the spreadsheet at the bottom of the form, select the **0th cell** in the From column. In the **Spreadsheet Input** data line, type **600.0** and **Enter**. Move to the next cell down and repeat these steps for **500, 400, 300, 200, 110**, and **50**. Select **Calculate**. Hit **Apply**.

Finally select **Assign Target Range to Viewport**. Close all the sub-forms and click **Apply** on the **Results** form.

**12. Compare the results to the analytical solution.**

Shown below is the temperature contours derived by K. C. Chang and V. J. Payne.

---

**Compare results**

- **temperature-independent**
- **temperature-dependent**

---

**material 1**

**interface**

**material 2**

---

**0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1.0**

**0.0 0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1.0**

---

**773**

**673**

**573**

**473**

**383**

**50**
13. **Quit** MSC/PATRAN.

Select **File** on the *Menu Bar* and select **Quit** from the drop-down menu.