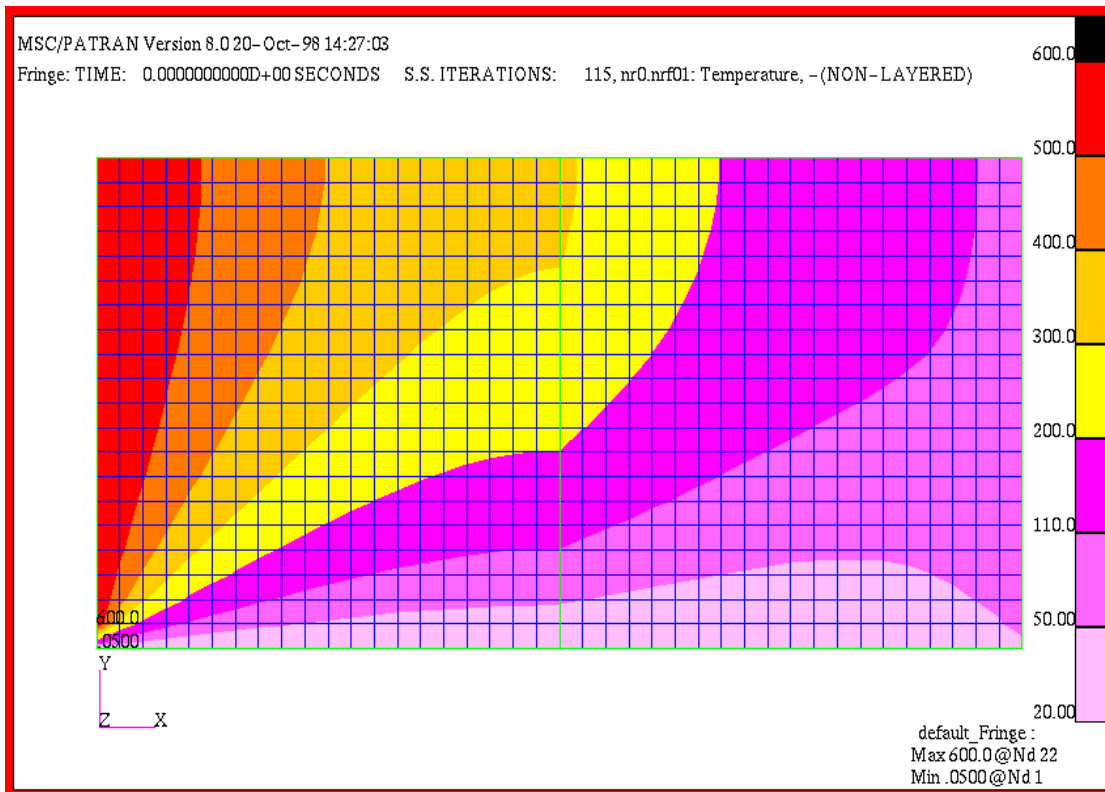


WORKSHOP 8

Temperature Dependent Material Properties



Objective:

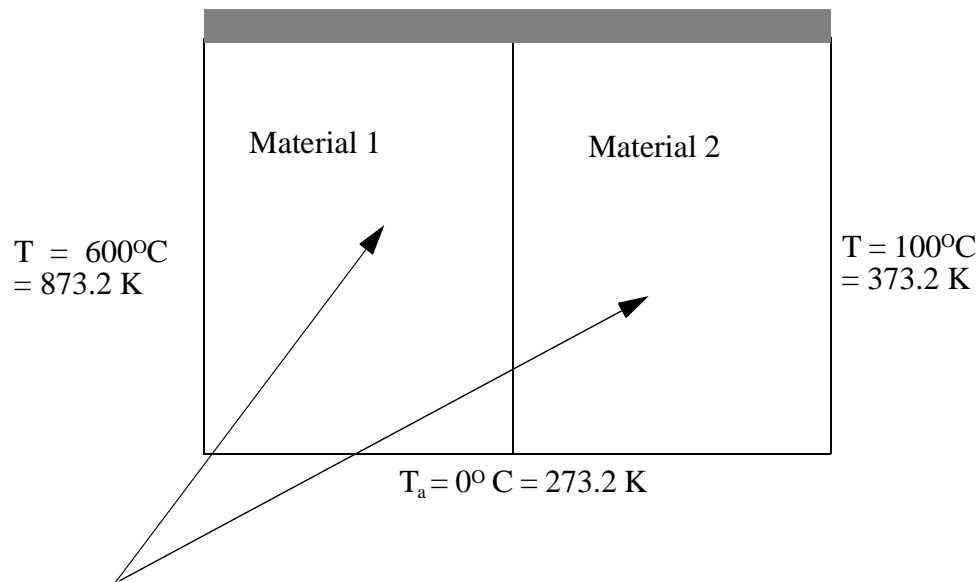
- You will create a 2D material slice consisting of two materials with temperature dependent material properties.
- You will visually and qualitatively compare the MSC/THERMAL results with the results of an analytical solution.



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.



0.5 X 0.5 Dimension

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

$$K_{10} = 0.060 \quad \alpha_1 = 0.0006$$

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

$$K_{20} = 0.001 \quad \alpha_2 = 0.00001$$

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:

Create a new database

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.

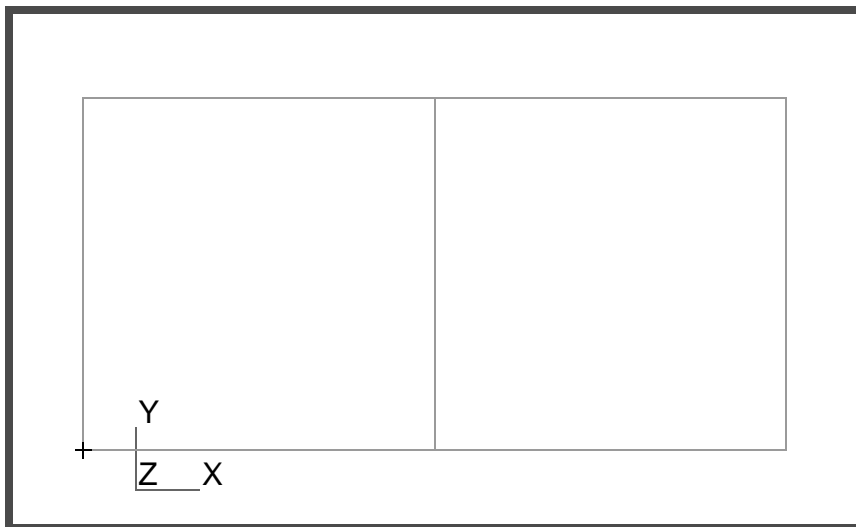
MSC.Patran will open a Viewport and change various *Main Form* selections from a ghosted appearance to a bold format. When the New Model Preferences form appears on your screen, set the *Tolerance* to **Default**, and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences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

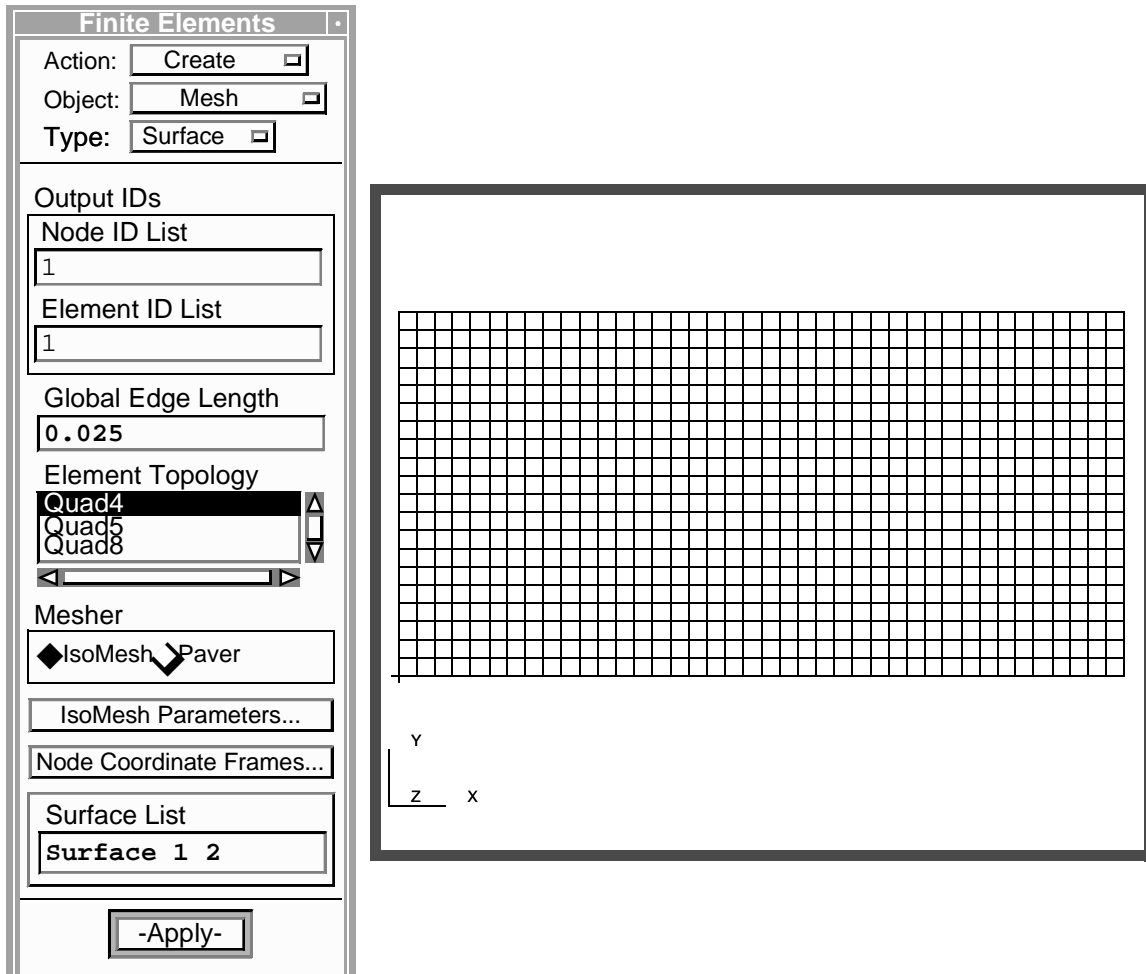
**Create the
two material
surfaces**



IsoMesh both surfaces

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.



The display should appear as shown above. If it does not, select the *undo* icon and analyze the error to a resolution.

Equivalence mesh nodes

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.

Verify element boundaries again.

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.

- 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**.

General Field Input Data

Select Function Term:

Function Term Type: P3 Functions

Term Sub-Type: MSC/THERMAL Matl Func

Select Function Term: mpid_arbt_plyn
mpid_bghm
mpid_cnst

Function Expression

Modify Highlighted Function

OK

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 K2** conductivity, (**$K2 = 0.001 + 0.00000001 T$**). First enter **0.001** in the *Coefficient,A(Index)* followed by a **carriage return**. Next enter **1.0E-8** followed by a **carriage return**. Before completing this form enter a description in the **Description** entry box.

Define the
material
property fields

The form should appear as shown below.

Arbitrary Order Polynomial

Define Material Property: Arbitrary Order Polynomial
 $P(X) = A(1) + A(2)*X + \dots A(n)*X^{(n-1)}$
Note: The temperature scale only indicates the valid units.
ICCALC units will be used in the evaluation.

Select Existing Table...

Build Arbitrary Order Polynomial Table

Description - Arbitrary Order Polynomial Table
Material K2

Material Property ID (MPID) Scale Factor
100001 1.0

Independent Variable Type Temperature Units
Temperature Kelvin

Input Coefficient, A(I) Value

	Coefficient, A(I)
1	0.001
2	9.99999999E-09
3	
4	

Clear Selected cell(s) Delete selected row (s)

Number of Rows to Insert 1 Insert row(s)

OK Defaults Cancel

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**. Click on the Coefficient 1 cell in the *Coefficient Data* frame and enter the *Coefficient, A(Index)* data for the thermal conductivity of Material 1, (**K1 = 0.06 + 0.000036T**). Change the Temperature units to **Kelvin** and add a description in the **Description** entry box.

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*.

Property Name	Value
Thermal Conductivity	K1
Density	1.0
Specific Heat	1.0
Phase change temperature	
Latent Heat	

Time, Temperature or Constant Fields:

- K1
- K2

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.

Define
material
properties

Apply
element
properties

Perform the same steps for **Surface 2**, the right surface, using **Prop2**, for the *Property Set Name*, and **Material_2** for the *Material Name*.

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

Left vertical edge of Surface 1:

Select the **Loads/BCs Applications** radio button. Set the *Action*, *Object*, and *Type* to **Create/Temperature (PThermal)/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.

The image displays three screenshots of software dialog boxes used for applying boundary conditions:

- Load/Boundary Conditions:** Shows settings for creating a new boundary condition. Action: Create, Object: Temperature(PThermal), Type: Nodal, Option: Fixed, Analysis Type: Thermal, Current Load Case: Default..., Type: Static, Existing Sets: (empty), New Set Name: Mat1_Edge_Temp, and an Input Data... button.
- Input Data:** Shows the Fixed Temperature dialog with the value 873.2 entered, a Select Spatial Field... button, a Reset button, and OK/Cancel buttons.
- Select Application Region:** Shows the Geometry Filter set to Geometry and FEM. The Application Region section shows a Select Geometry Entities box, an Add button, a Remove button, and the Application Region list containing Surface 1.1. An OK button is at the bottom.

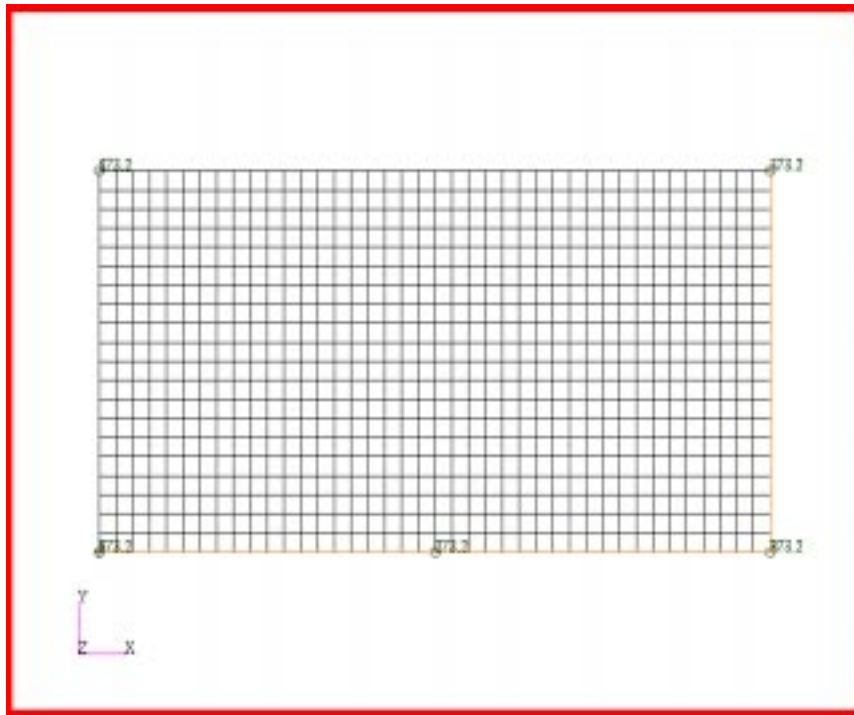
Apply boundary temperatures to 3 edges

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.

<i>New Set Name</i>	<i>Fixed Temp</i>
Mat1_2_Bottom_Edge_Temp	273.2
Mat2_Edge_Temp	373.2

Your model should now look like the one shown below.

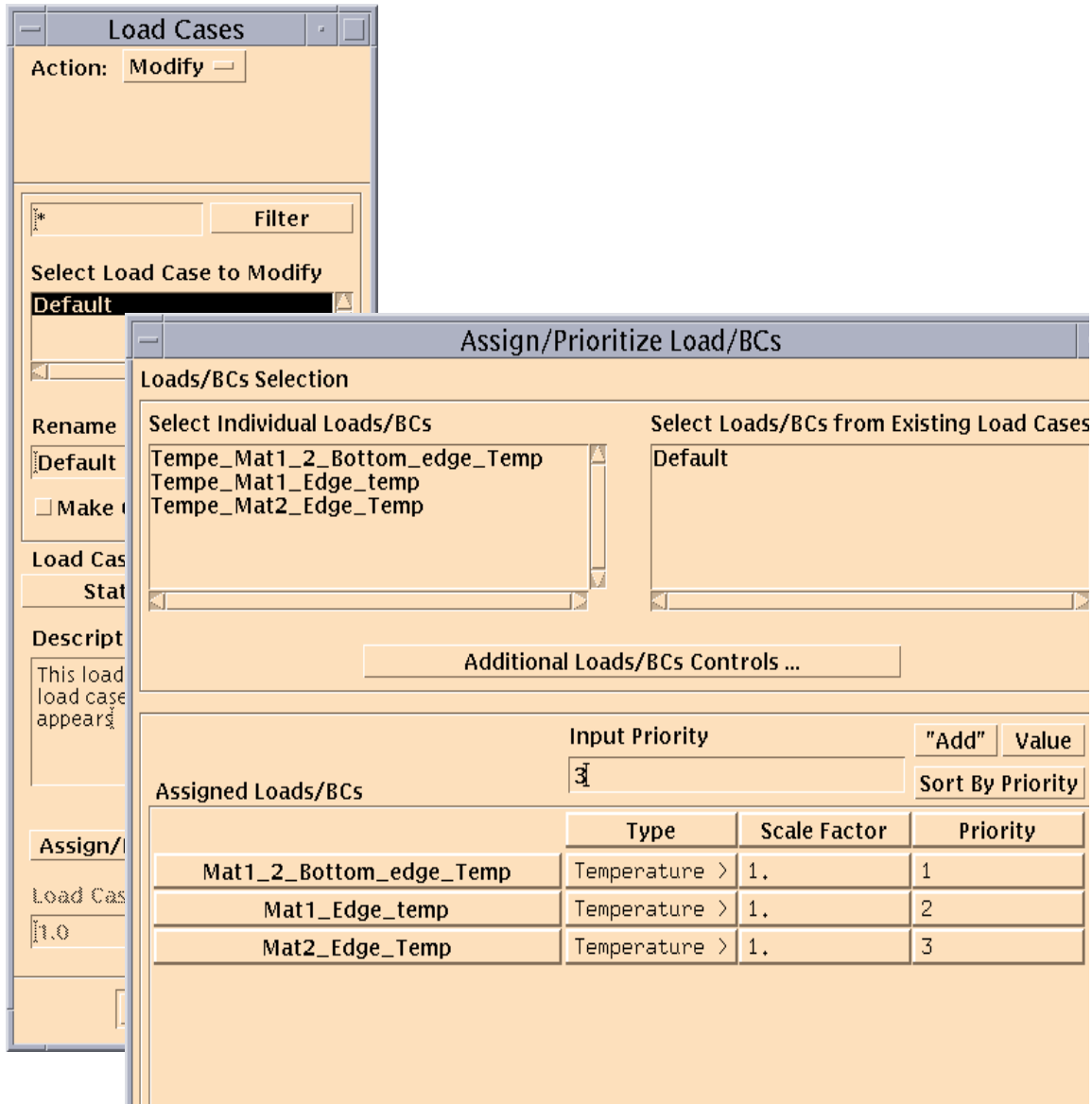


Applying the temperature boundary conditions to the various edges of your model created a conflict at the two lower corner points. At the lower left corner both the **873.2** and **273.2** temperature boundary conditions were applied. At the lower right corner both the **373.2** and **273.2** temperature 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.

Prioritize temperature BC's

- 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 **Priority cell** for LBC 1, **Mat1_2_Bottom_Edge_Temp**. Select the **Value** button. A value of **1** should appear in the **Priority cell**. Select the **Mat1_Edge_Temp** LBC then again set the **Priority** to **2** using the **Value** cell. 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. If the **Message** box, "Do you wish to overwrite?" appears, 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 MSC.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.

Prepare and run analysis

Read and plot results

To plot the results use the **Results Application radio button**. The default Action/Object should be **Create/Quick Plot**. Select **Fringe Result: Temperature**. Hit **Apply** to quick plot the default Result Case and Fringe Result.

To affect a better comparison use the *Fringe Attributes* 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 spread sheet at the bottom of the form, select the **0th cell in the From column**. In the **Spreadsheet Input** data line, type **600.0** and **carriage return**. Move to the next cell down and repeat these steps for **500, 400, 300, 200, 110**, and **50**. Select **Calculate**. Hit **Apply**.

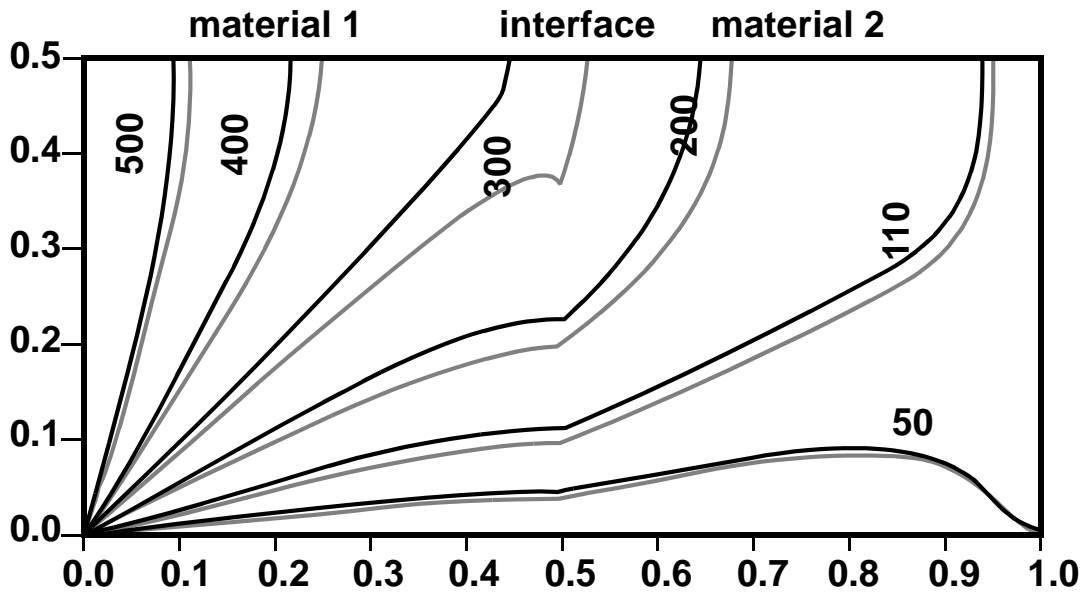
Finally select **Assign Target Range to Viewport**. Close all the sub-forms and click **Apply** on the Results form.

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



13. **Quit** MSC.Patran.

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

Quit
 MSC.Patran

