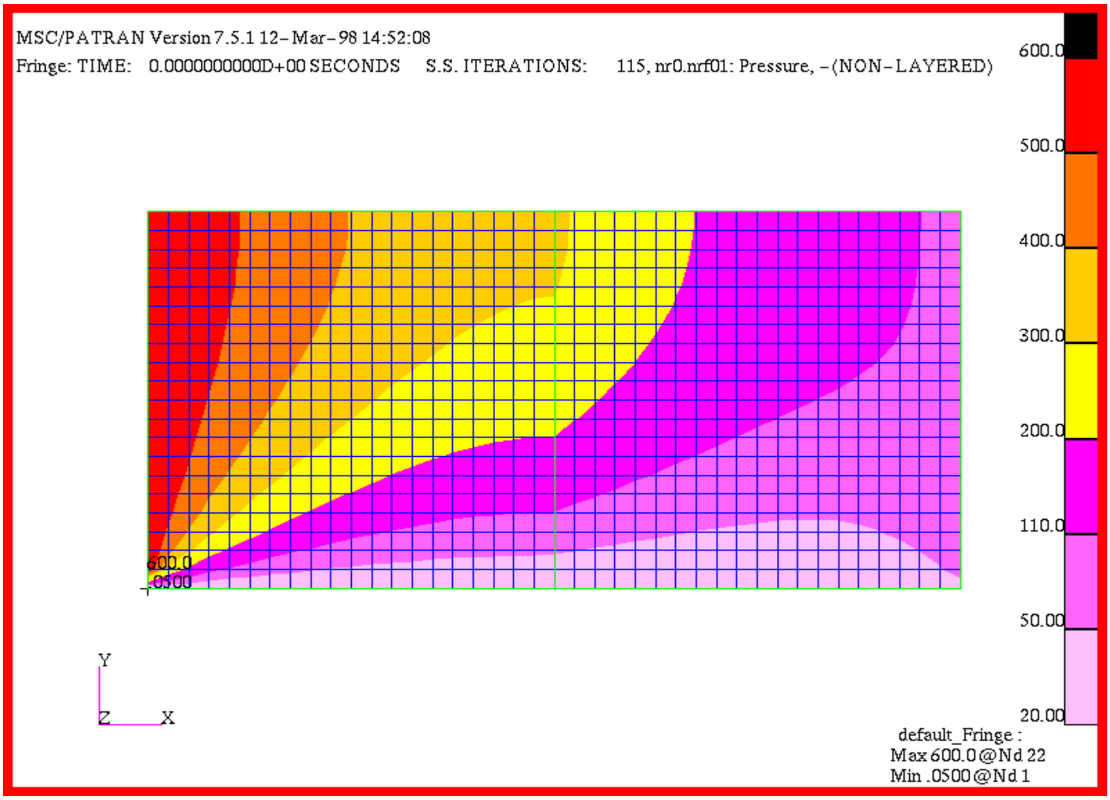
Exercise 8

Temperature Dependent Material Properties



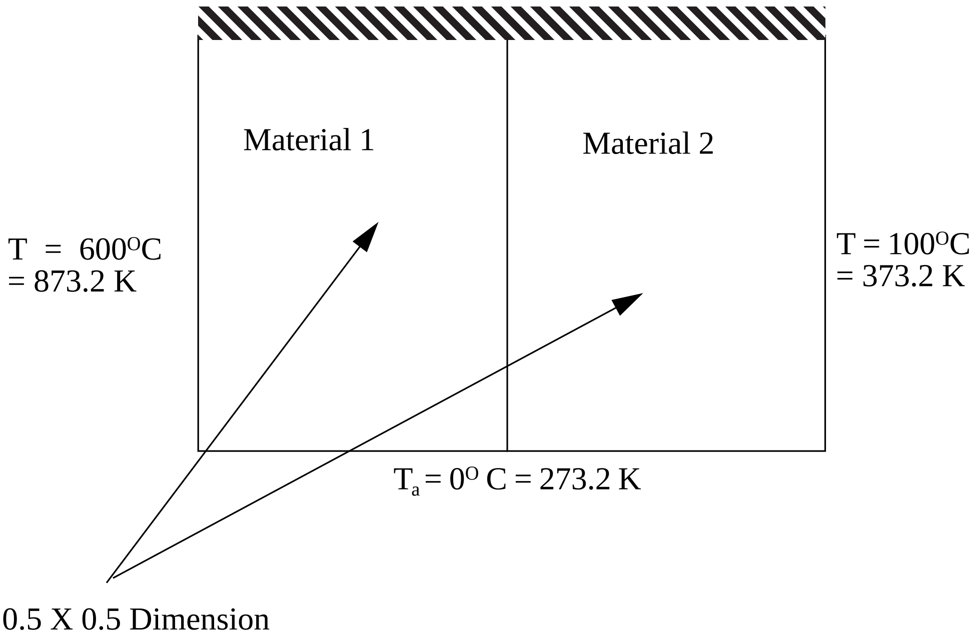
Objective:

- 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.



$K_1 = K_{10}(1 + \alpha_1 T)$ $\quad K_{10} = 0.060 \quad \alpha_1 = 0.0006$

$K_2 = K_{20}(1 + \alpha_2 T)$ $\quad K_{20} = 0.001 \quad \alpha_2 = 0.00001$

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.

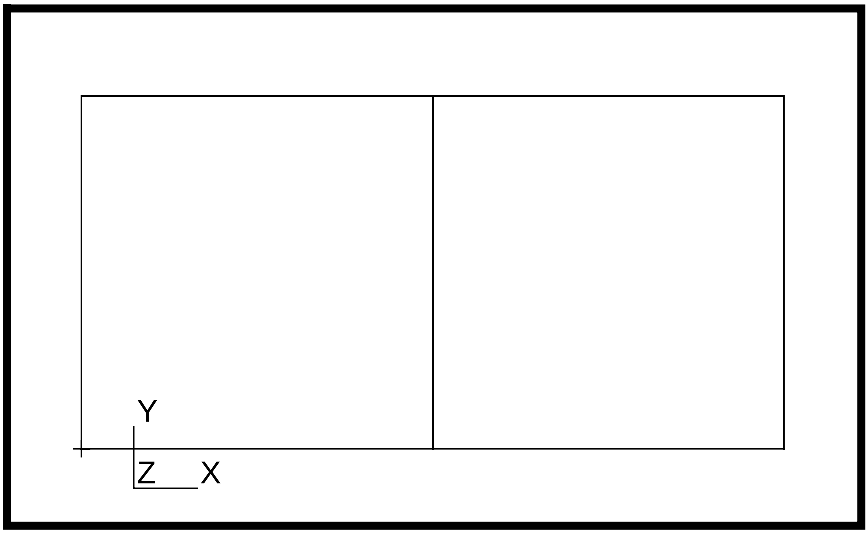
PATRAN will open a Viewport and change various *Main Form* selections from a ghosted appearance to a bold format. When the New Model Preferences form appears on your screen, set the *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**. Select **OK** to close the New Model Preferences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

Create the two material surfaces

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

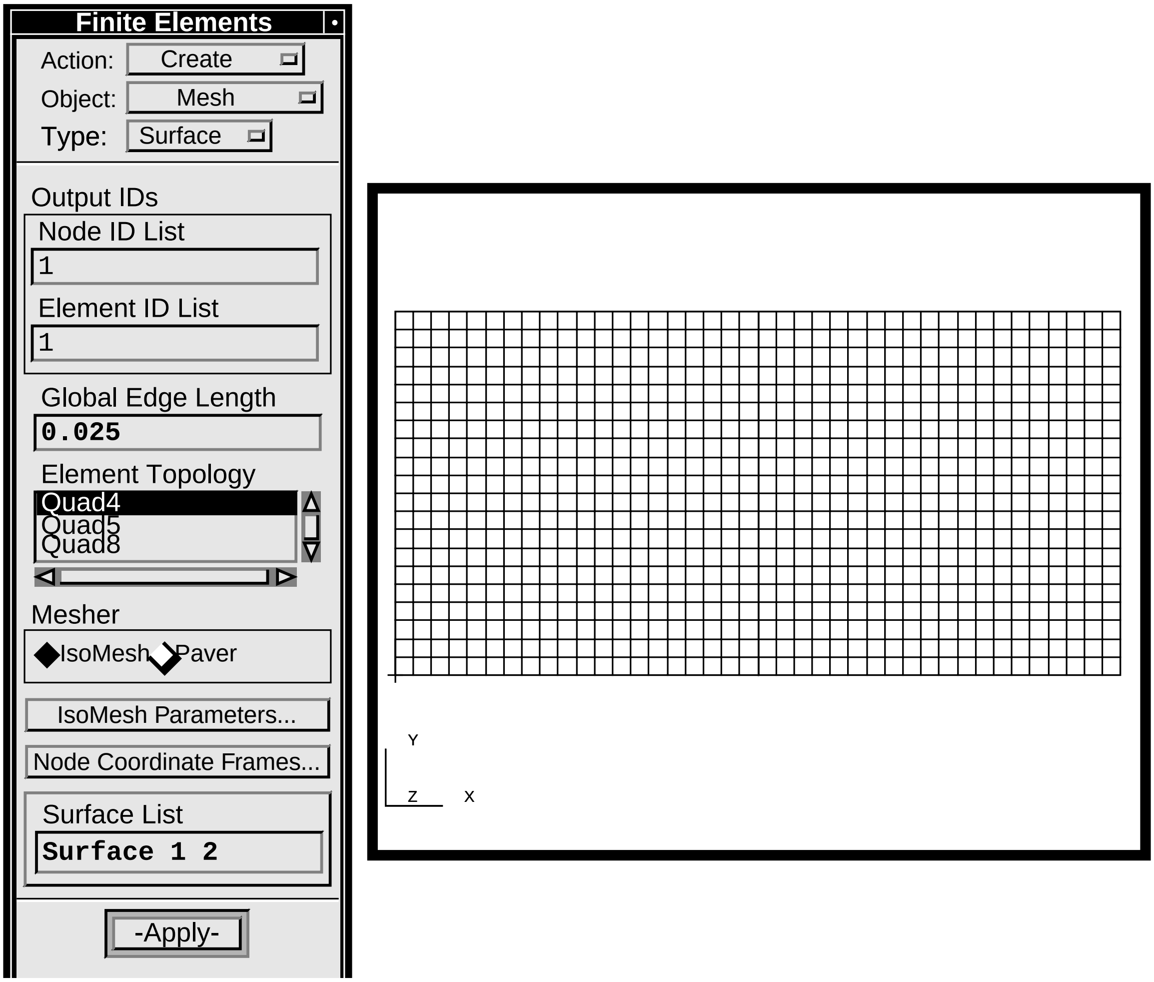
Change the *Origin Coordinates List* to **[0.5, 0, 0]**, and click on the **Apply** button to create the second surface.



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.

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.

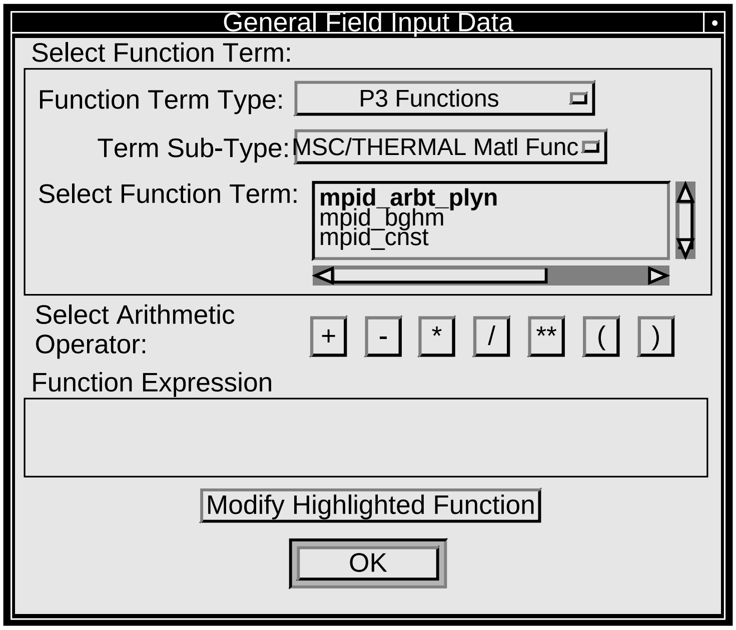
Define the material property fields

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**.

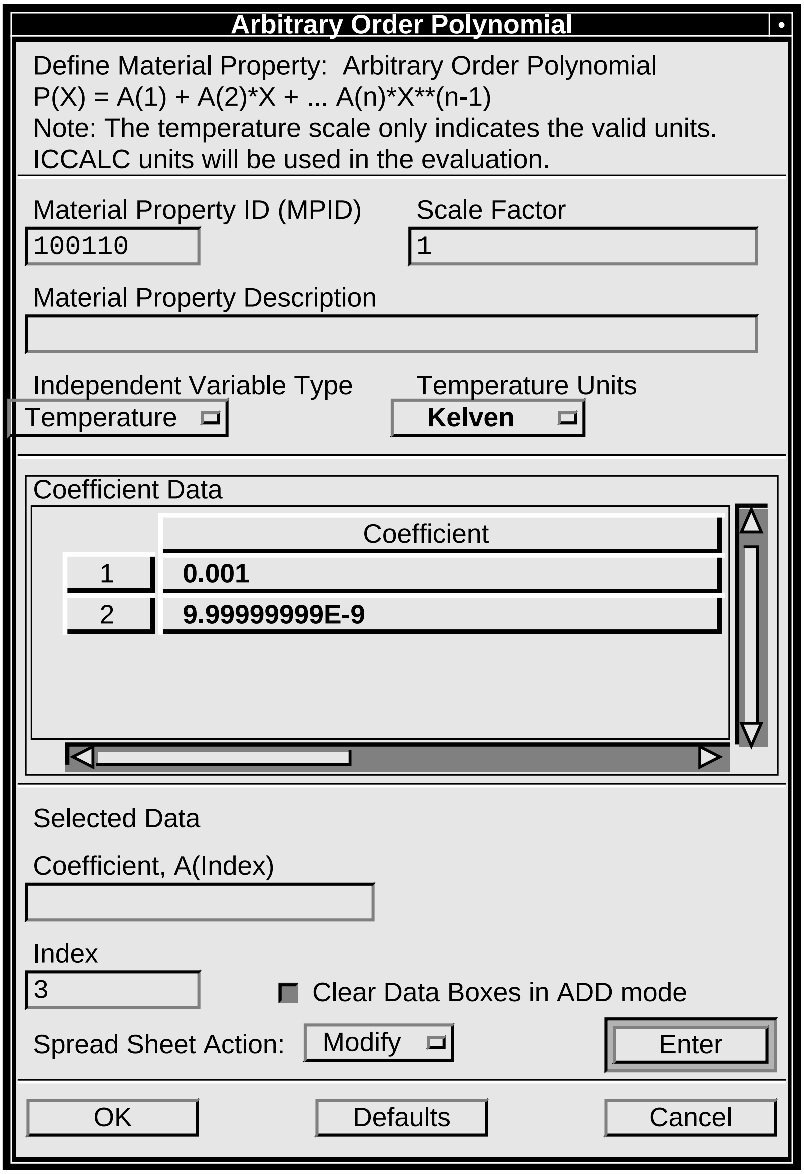
Enter a *Field Name* **K2** and select **Input Data...**

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.

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.

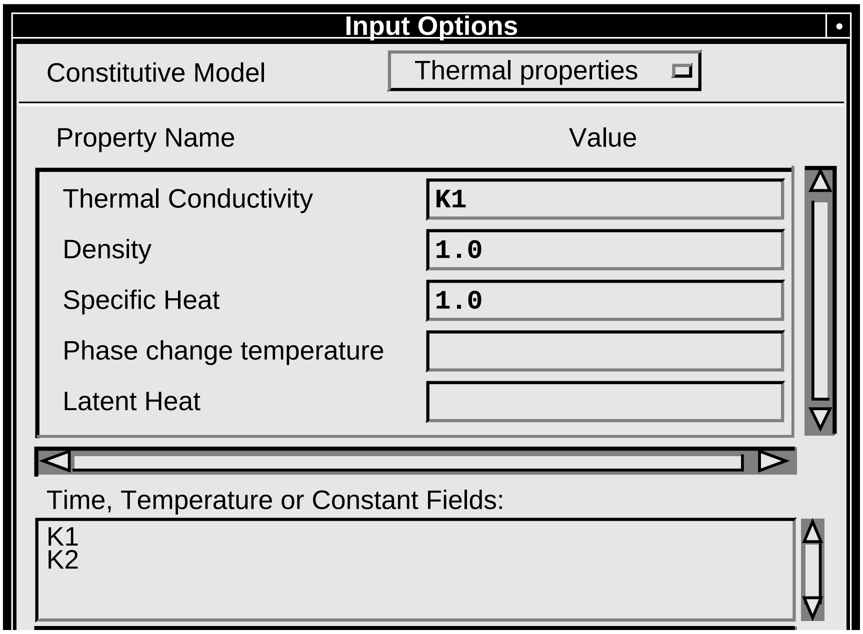
In the Field form change *Field Name* to **K1**. Again choose the **mpid_arbt_plyn**. Using the *Spread Sheet Action* **Modify**, click on the Coefficient 1 cell in the *Coefficient Data* frame, and edit the *Coefficient,A(Index)* data box to change the Arbitrary Order Polynomial form for the thermal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

Define material properties

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

Apply element properties

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*.

Apply boundary temperatures to 3 edges

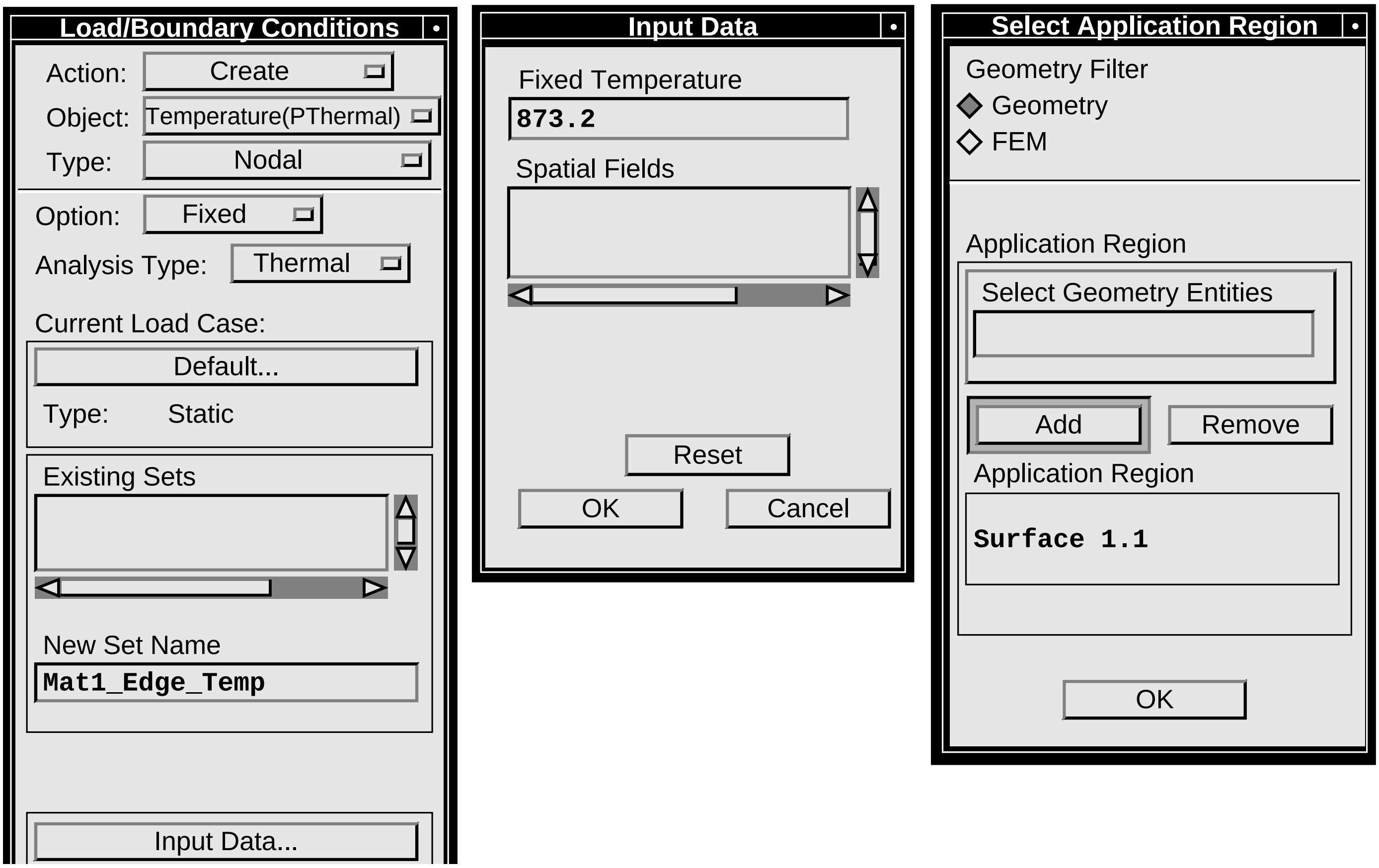
8. You will now apply the three temperature boundary conditions to the edges of your model.

Left vertical edge of Surface 1:

Select the **Load/BCs** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Tempature (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.

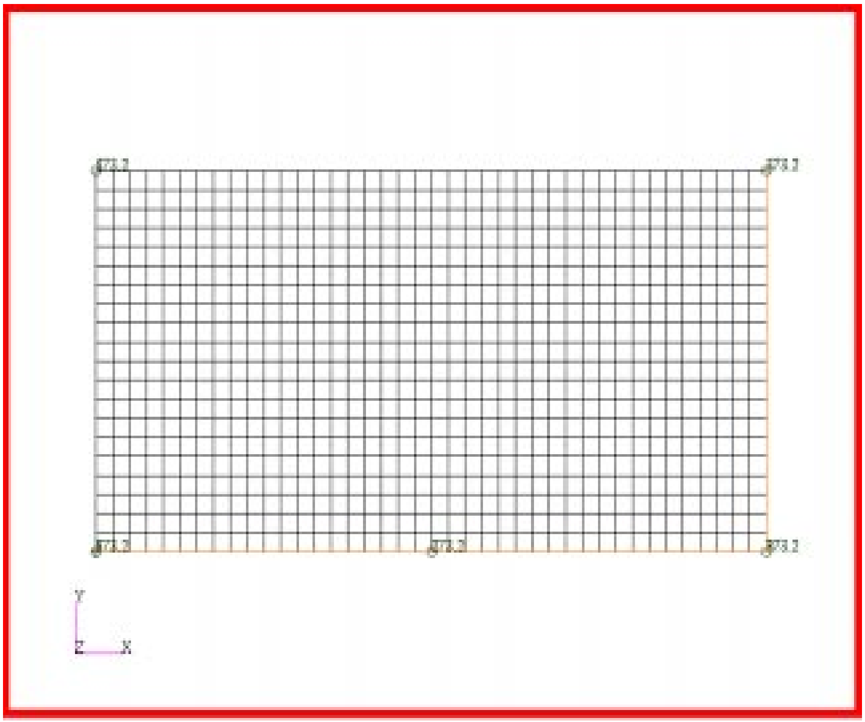


Select **Apply** to create the temperature boundary condition.

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

New Set Name	*Fixed Temp*
Mat1_2_Bottom_Edge_Temp	**273.2**
Mat2_Edge_Temp	**373.2**

Your model should now look like the one shown below.

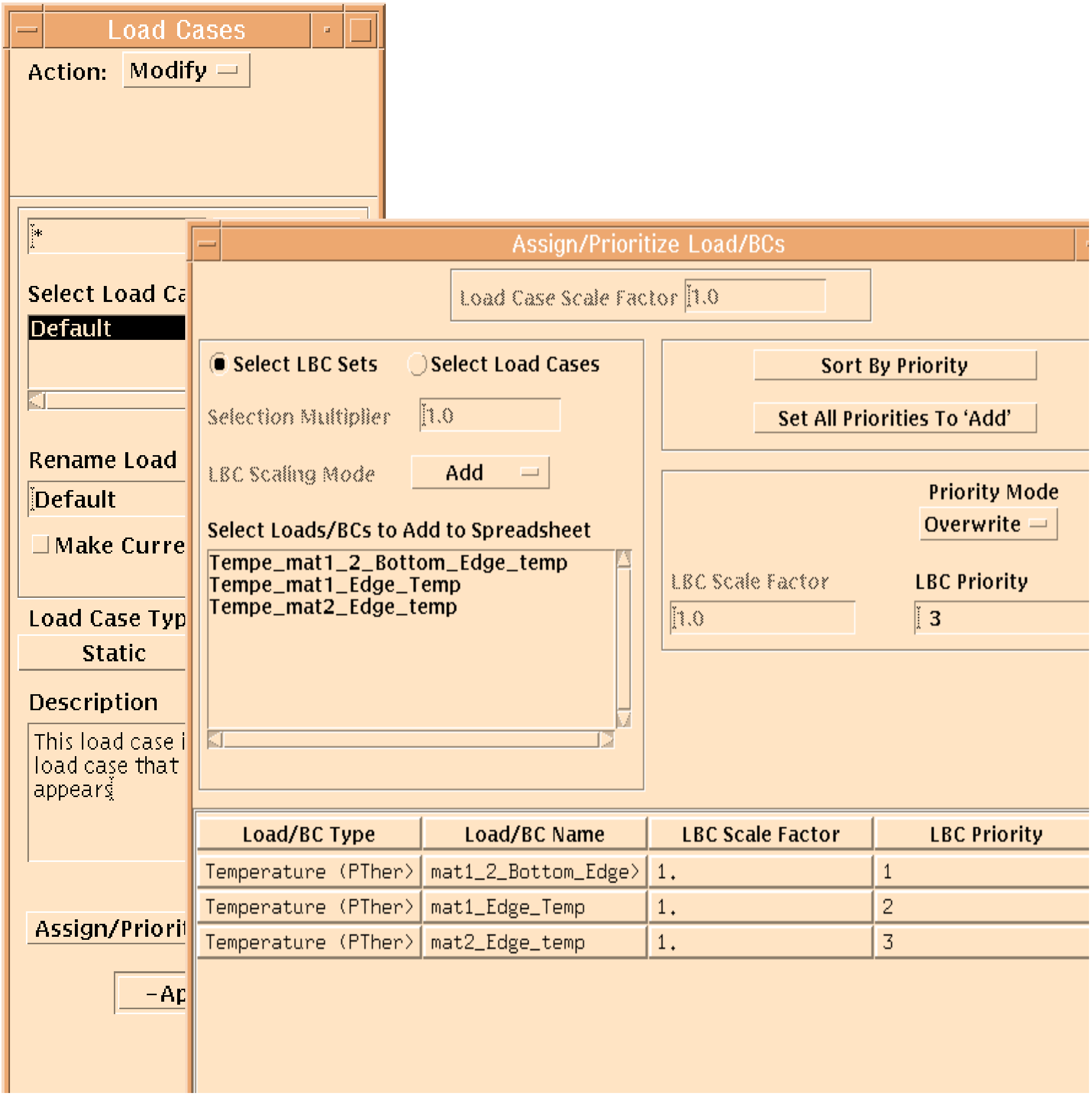


Applying the temperature boundary conditions to the various edges of your model created a conflict at the two lower corner points. At the lower left corner both the **873.2** and **273.2**temperature boundary conditions were applied. At the lower right corner both the **373.2** and **273.2** temperature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

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.

Prepare and run 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.

Read and plot results

Recall that p3 was initiated from a working directory which contained the excercise_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 comparision 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.**

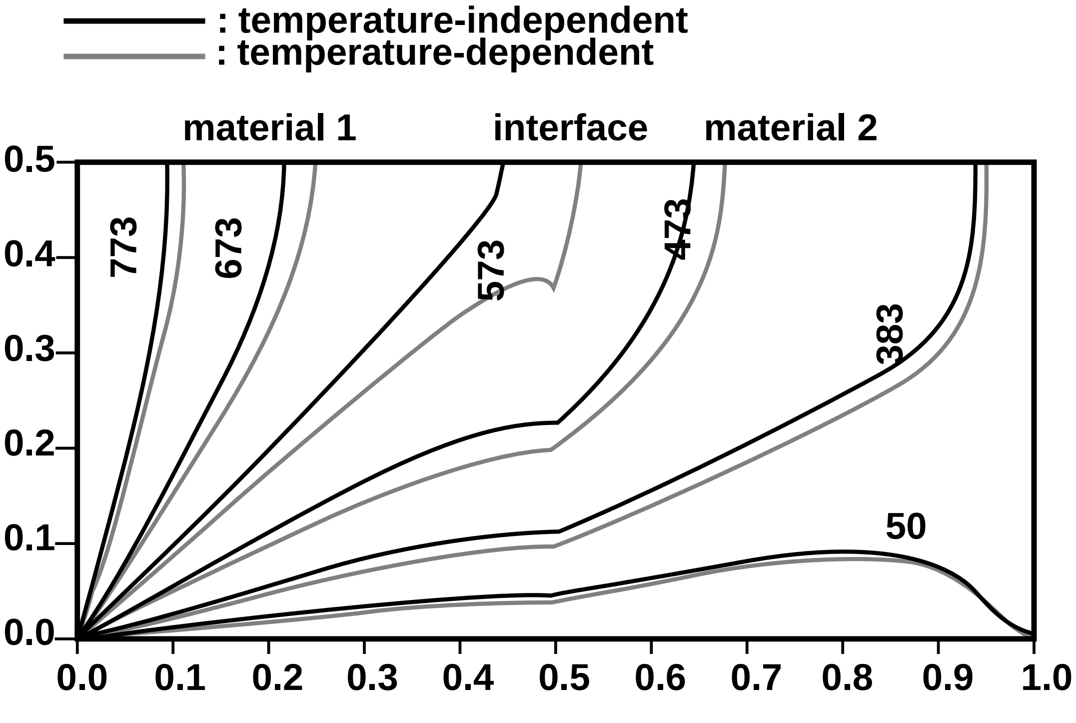
In the Range form select **Data Method/From**. In the spread sheet at the bottom of the form, select the **0th cell in the From column**. In the **Spreadsheet Input** data line, type **600.0** and **Enter**. Move to the next cell down and repeat these steps for **500, 400, 300, 200, 110,** and **50**.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

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

Compare results

12. Compare the results to the analytical solution.

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



13. **Quit** MSC/PATRAN.

Quit MSC/ Patran

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