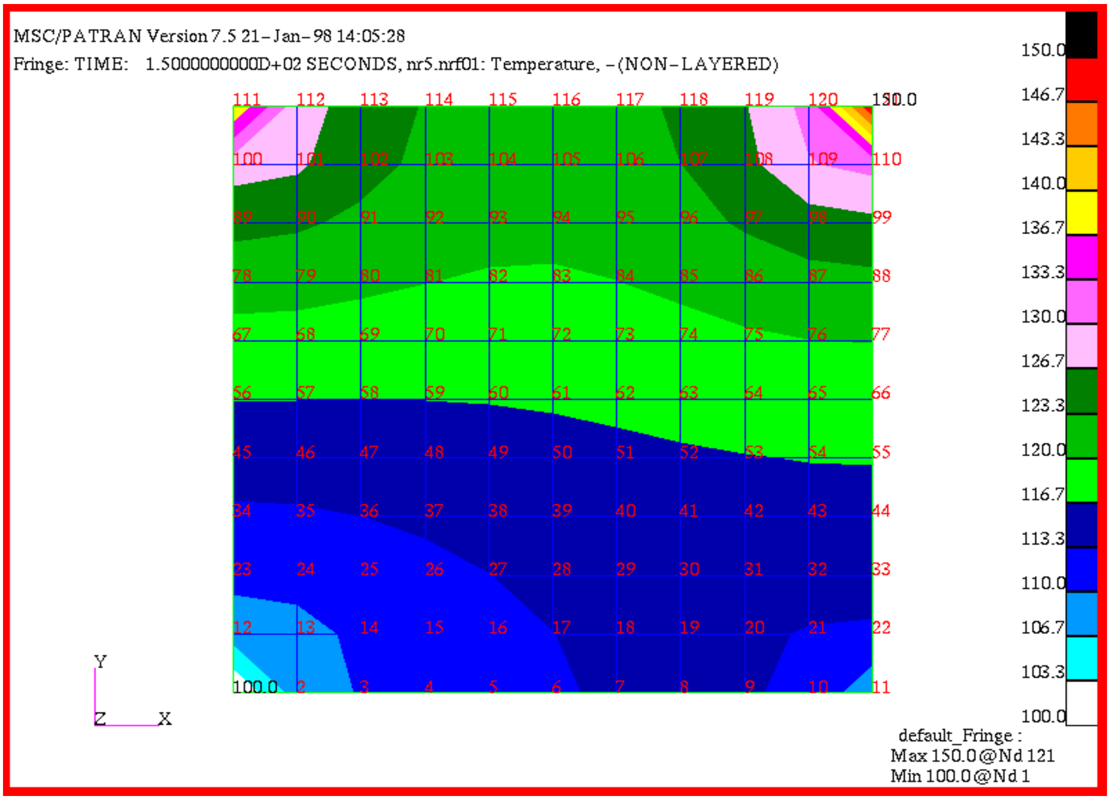
Exercise 10

Time Dependent Boundary Conditions



Objective:

- Model an aluminum plate.
- Use microfunctions to apply time dependent boundary conditions to the plate corners.
- Run a transient analysis to produce time dependent results.

Model Description:

In this exercise you will define MACROs and microfunctions. MACRO definitions are edited into a **template.dat.apnd** file which you create in the same directory as your database.

MACRO definitions link Template ID's (TID's) which are applied in the Loads/BCs form to Microfunction ID's (MFID's) which are defined in the Fields form. A microfunction can be a function of time or various temperature functions. This provides a mechanism for defining time or temperature varying heat load or temperature boundary conditions. Only constant or spatially varying loads or boundary conditions can be defined directly in the Loads/BCs forms.

In this exercise we will sample three of the available microfunctions: a sine wave, a flip-flop function and a linearly interpolated data table. These functions are applied to three of the four corners of an aluminum plate modelled from shell elements. The fourth plate corner will have a constant boundary temperature.

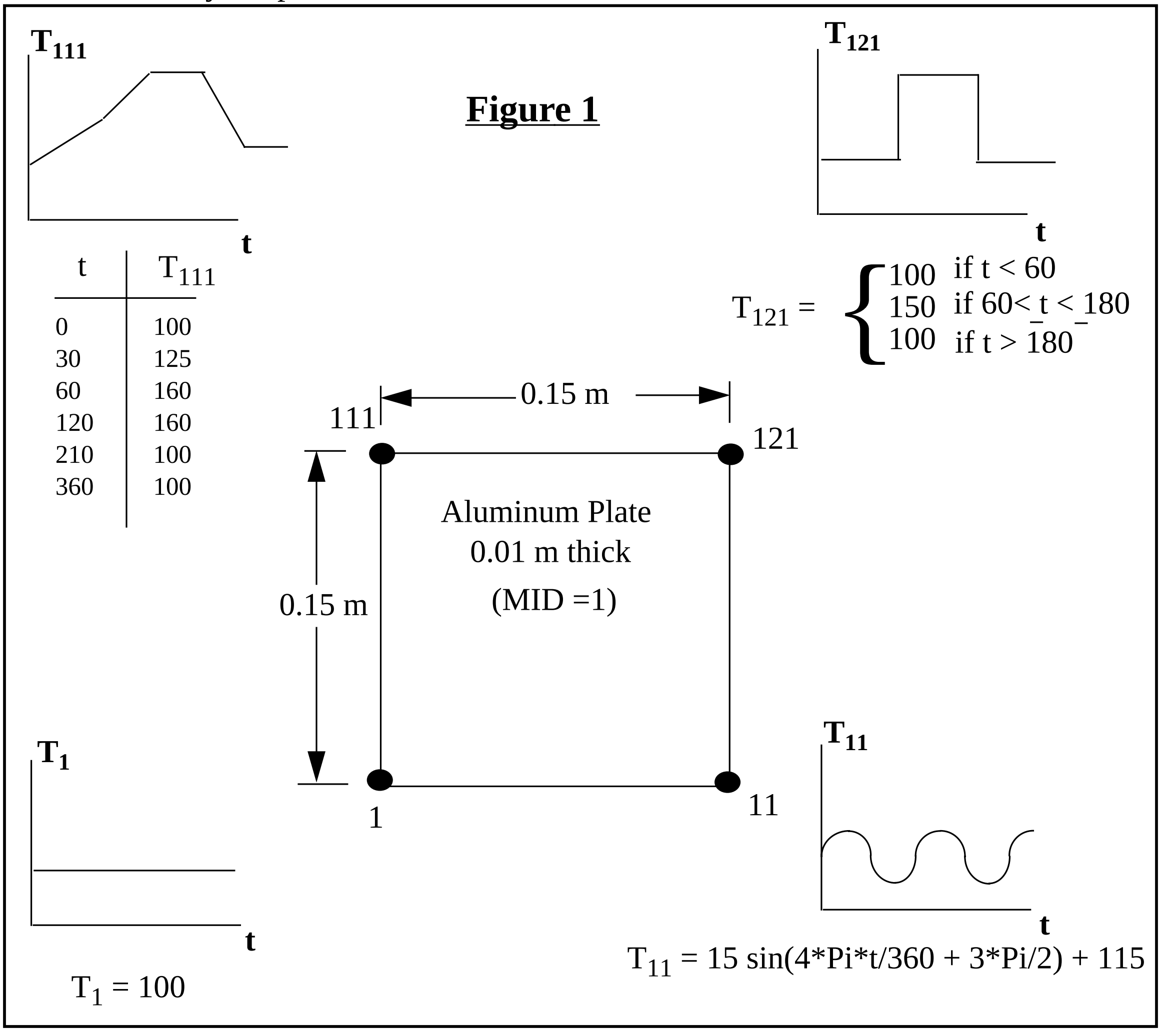


Figure 1

t	T_{111}
0	100
30	125
60	160
120	160
210	100
360	100

$$T_{121} = \begin{cases} 100 & \text{if } t < 60 \\ 150 & \text{if } 60 \le t \le 180 \\ 100 & \text{if } t > 180 \end{cases}$$

$$T_1 = 100$$

$$T_{11} = 15 \sin(4*Pi*t/360 + 3*Pi/2) + 115$$

Exercise Overview:

- Create a new database named **exercise_10.db.** Set *Tolerance* to **Default,** and the *Analysis Code* to **MSC/THERMAL**.
- Create a 0.15m x 0.15m plate surface.
- Mesh the surface with an IsoMesh of quad4 elements, global edge length of 0.015.
- Apply element properties to the quad4's defining them as shell elements having a material name (MID) of 1 and a thickness of 0.01m.
- Create 3 time dependent microfunctions using **Fields** and **Create/Non Spatial/General**.
- Define 4 temperature boundary condition in Loads/BC's, 1 fixed nodal temperature in the lower left corner of the plate and 3 variable nodal temperatures on the remaining corners.
- Open a new window (shell) and in the directory which contains the database and edit a file named template.dat.apnd creating the MACRO definitions.
- Prepare and submit the model for analysis specifying that it is a transient analysis from t=0s to t=360s with output each 30s, that the global initial temperature is 100^{o}C, and that all calculations and output should be ^{o}C.
- Read the results files using Shareware and plot results for several time steps. Do not delete the database from your directory since it will be used in a future exercise.
- **Quit** MSC/PATRAN.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_10.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 *Menu Bar* and select **New…** from the drop-down menu. Assign the name exercise_10.db to the new database by clicking in the *New Database Name* box and entering **exercise_10**.

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

File

New

New Database Name **exercise_10**

OK

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.

Tolerance ◆ **Default**

Analysis Code **MSC/THERMAL**

OK

2. Create a 0.15m x 0.15m plate surface.

Create 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.

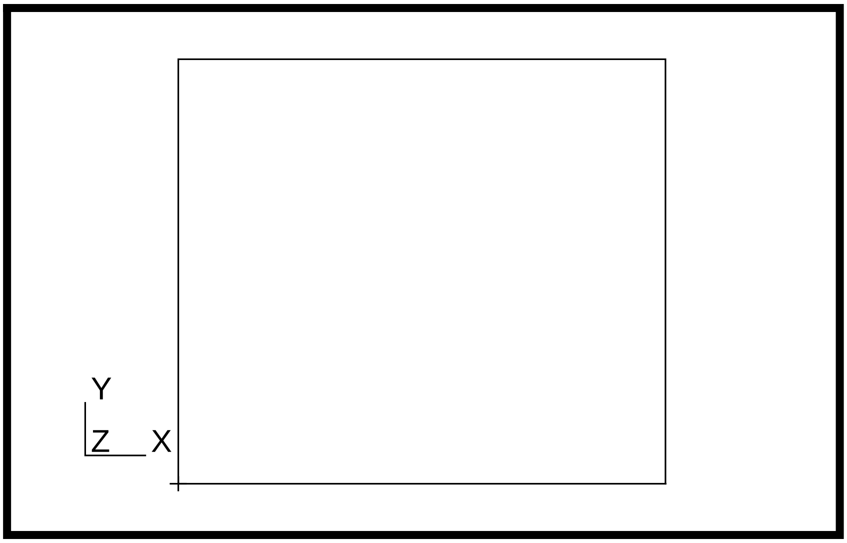
◆ **Geometry**

Create/Surface/XYZ

Vector Coordinate List **<0.15 0.15 0>**

Apply

The resulting model is shown below.



IsoMesh the surfaces

3. Mesh the surface with an IsoMesh of quad4 elements, global edge length of 0.015.

Select the **Finite Elements** *Applications radio button*. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Surface**. Change the *Global Edge Length* to **0.015** and select **Surface 1** for inclusion in the *Surface List*.

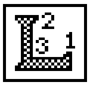
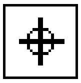
◆ Finite Elements

Create/Mesh/Surface

Global Edge Length	**0.015**
Surface List	**<click on Surface 1 in the viewport>**

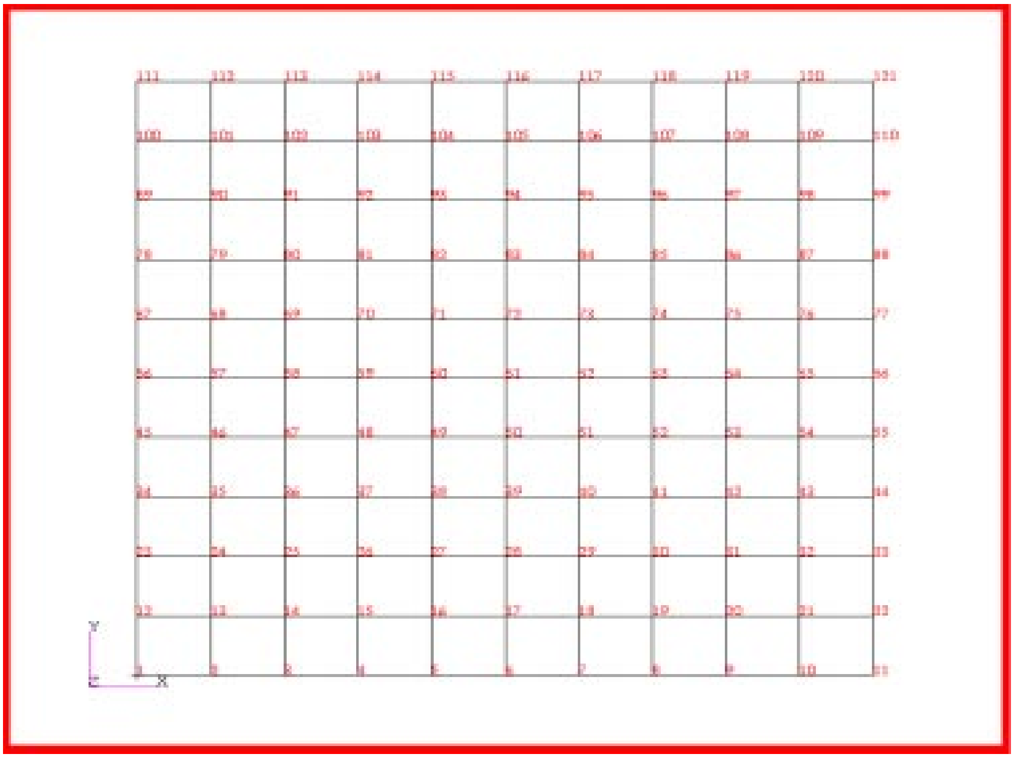
Apply

Use the *Tool Bar Label Control* icon to turn on node labels **only**.

First select  then 

Close

The display should now appear as shown below.



4. Apply element properties to the **Quad4's** defining them as shell elements having a *Material Name* (MID) of **1** and a thickness of **0.01**m.

Apply element properties

Select the **Properties** *Applications radio button*. Set the *Action, Dimension,* and *Type* to **Create/2D/Shell**. Enter *Property Set Name* **Prop1**. Select the *Input Properties...* box. Click in the *Material Name* box and enter **1**. Enter **0.01** in the *Shell Corner Thickness* list box. Select **OK** to close the form.Click in the *Select Members* box and select Surface 1 in the viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties

Create/2D/Shell

Property Set Name **Prop1**

Input Properties...

Material Name **1**

Shell Corner Thickness **0.01**

Ok

Select Members **<select Surface 1 in the viewport>**

Add

Apply

5. Create 3 time dependent microfunctions using **Fields** and **Create/Non Spatial/General**.

Create micro-functions

Microfunctions are created in the Fields form using the *Action/Object/Method* **Create/Non Spatial/General**. After selecting **Input Data.** The General Field Input Data form will show the complete list of microfunctions in the *Select Function Term:* list box.

Figure 1 contains the data required for entry into the various microfunction forms. The entries and selection below will guide you through the process of creating the microfunctions. An image of each completed microfunction form is included to facilitate microfunction entry.

◆ Fields

Create/Non Spatial/General

Field Name **Tsine**

Input Data...

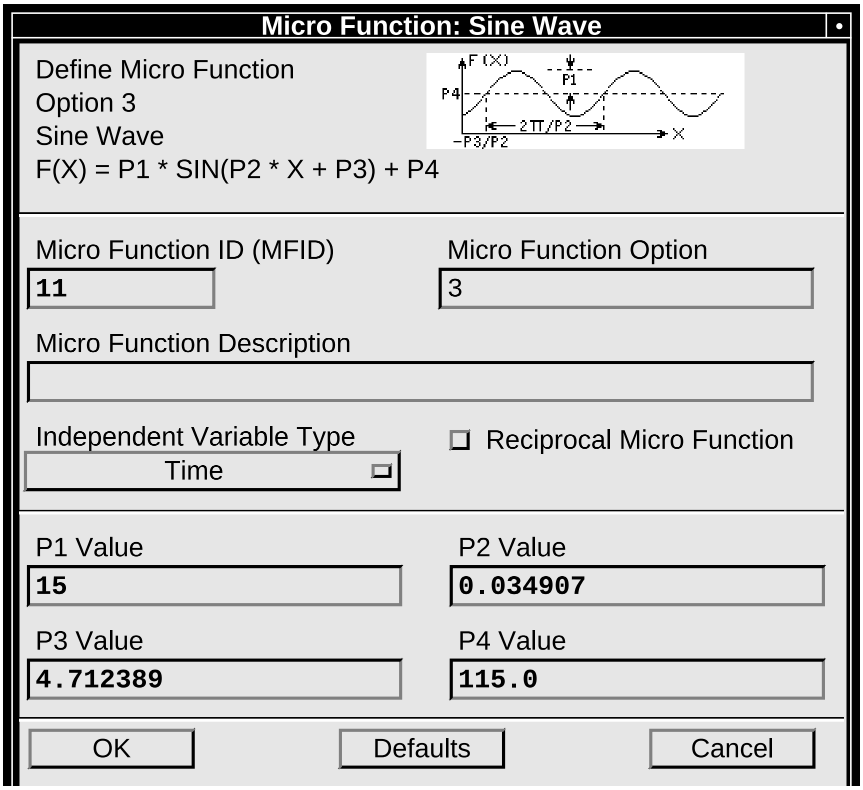
Select Function Term	**mfid_sine_wave**
Micro Function ID	**11**
P1 Value	**15.0**
P2 Value	**0.034907**
P3 Value	**4.71239**
P4 Value	**115.0**

OK

OK

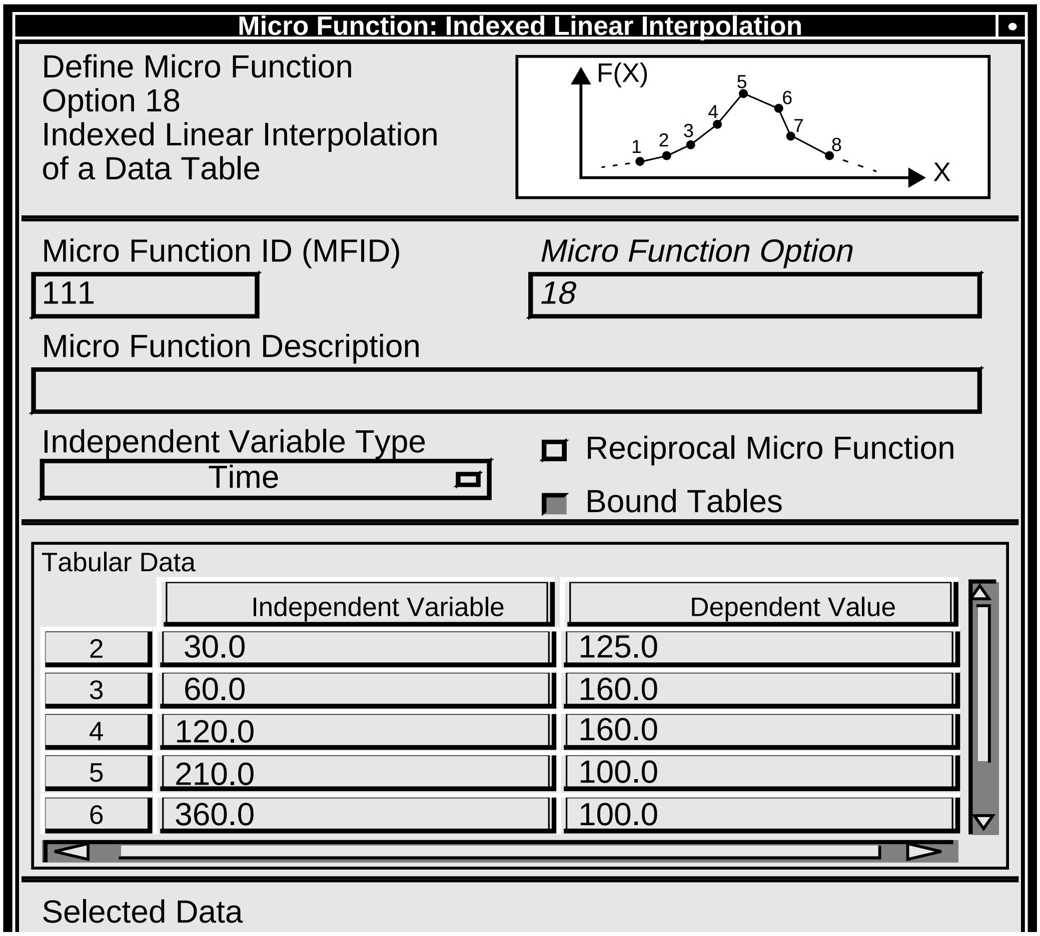
Apply

The Micro Function form should appear as shown below.



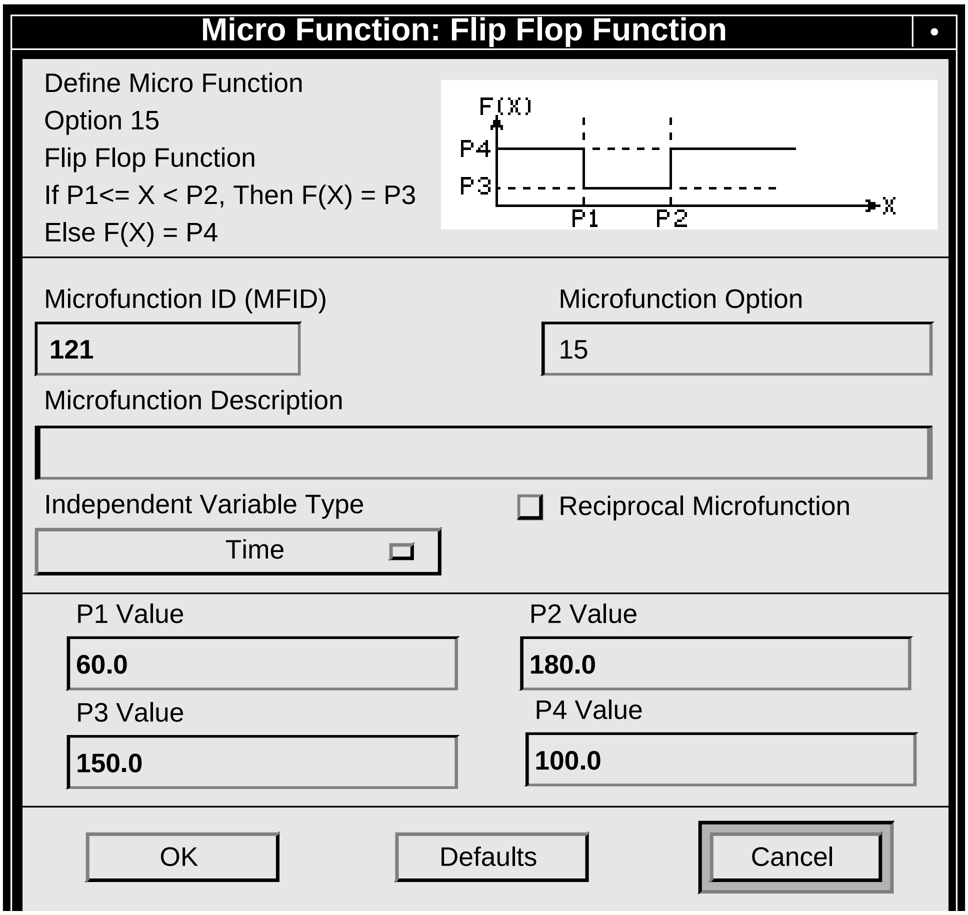
Field Name	**Ttable**
Input Data...	
Select Function Term	**mfid_indx_linr_tabl**
Micro Function ID	**111**
Independent Variable, (X)	**0.0**
Value, Function(X)	**100.0**
Enter	
Independent Variable, (X)	**30.0**
Value, Function(X)	**125.0**
Enter	
Independent Variable, (X)	**60.0**
Value, Function(X)	**160.0**
Enter	
Independent Variable, (X)	**120.0**
Value, Function(X)	**160.0**
Enter	
Independent Variable, (X)	**210.0**
Value, Function(X)	**100.0**
Enter	
Independent Variable, (X)	**360.0**
Value, Function(X)	**100.0**
Enter	
OK	
OK	
Apply	

The Micro Function form should appear as shown below.



Field Name	**Tflip_flop**
Input Data...	
Select Function Term	**mfid_flip_flop**
Micro Function ID	**121**
P1 Value	**60.0**
P2 Value	**180.0**
P3 Value	**150.0**
P4 Value	**100.0**
OK	
OK	
Apply	

The Micro Function form should appear as shown below.



6. Define 4 temperature boundary condition in Loads/BCs, 1 fixed nodal temperature in the lower left corner of the plate and 3 variable nodal temperatures on the remaining corners.

Apply boundary conditions

Begin applying boundary conditions. Select the **Load/BCs** *Applications radio button*. Create a fixed **100$^{\mathbf{o}}$C** nodal boundary temperature named **T1**. In the Input Data form define the fixed temperature. In the Select Applications Region form pick **Node 1** located in the lower left corner of the plate.

◆ Load/BCs

Create/Temperature/Nodal

Option:	**Fixed**
New Set Name	**T1**

Input Data...

Fixed Temperature	**100.0**

OK

Select Application Region...

Geometry Filter ◆ **FEM**

Select Nodes **<select Node 1>**

Add

OK

Apply

Create a variable boundary temperature named **T11** with *Template Id* **2** in the Input Data form. In the Select Applications Region form pick **Node 11** located in the lower right corner of the plate. Repeat these steps for *New Set Name* **T111** with *Template ID* **3** on **Node 111** and **T121** with *Template ID* **4** on **Node 121**.

◆ **Load/BCs**

Create/Temperature/Nodal

Option: **Variable**

New Set Name **T11**

Input Data...

Template ID **2**

OK

Select Application Region...

Select Nodes **<select Node 11>**

Add

OK

Apply

Repeat these steps for *New Set Name* **T111** with *Template ID* **3** on **Node 111** and or *New Set Name* **T121** with *Template ID* **4** on **Node 121**.

New Set Name **T111**

Input Data...

Template ID **3**

OK

Select Application Region...

Select Nodes **<select Node 111>**

Add

OK

Apply

New Set Name **T121**

Input Data...

Template ID **4**

OK

Select Application Region...

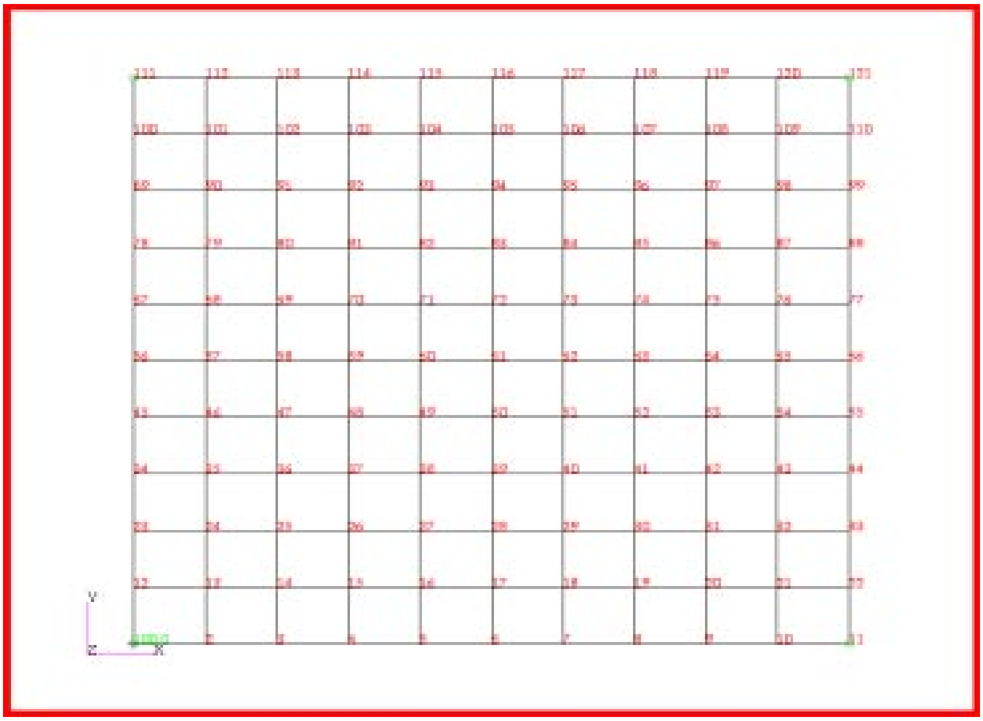
Select Nodes **<select Node 121>**

Add

OK

Apply

With boundary conditions applied the model should appear as shown below



In unix create template.dat. apnd file

7. Open a new window (shell) and in the directory which contains the database vi edit a file named **template.dat.apnd** creating the MACRO definitions.

Using the system editor, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC/PATRAN is running (You may need to open a new window.). The format for the MACRO functions are as follows:

MACRO tid micro_function_count node1 node2 scale_factor

mfid(1) mfid(2)... mfid(n)

tid: where,

This is a Template ID number (entered in the Loads/BCs form).

micro_function_count:

Number of microfunctions that will be assembled (grouped) in this MACRO function.

node1 & node2:

Node ID numbers used if temperature or a temperature difference is used for the microfunction argument. {e.g. for radiosity difference $\sigma*(T_1^4-T_2^4)$ (T_1 and T_2 temperatures at Node 1 and 2 respectively)}

scale_factor:

Optional scale factor for the MACRO function.

mfid#:

Integer ID number (pointer) of the microfunction. This is the MFID number you used in the microfunction definition.

Note: MACRO is a key word and must be typed in uppercase.

Use the following chart to help you define the MACRO functions for the temperature boundary conditions assigned to Nodes 11, 111, and 121.

Table 1:

TID#	Micro_function_count	Node 1	Node 2	scale_factor	mfid#
2	1	0	0	1	11
3	1	0	0	1	111
4	1	0	0	1	121

Note: Nodes 1 and 2 are set to zero since the argument is time.

Shown below is the final form of the **template.dat.apnd** file created for this exercise. Note that any comment lines must be started with an * in column 1 and make sure that there are no blank lines especially at the end of the file.

```
*================
MACRO 2 1 0 0 1.0
11
MACRO 3 1 0 0 1.0
111
MACRO 4 1 0 0 1.0
121
*================
```

8. 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

Solution Type...

Select Thermal Solution	◆ **1, Transient Run**

OK

Solution Parameters...

Calculation Temperature Scale	◆ **Celsius**

Run Control Parameters...

Stop Time	**360.0**
Initial Temperature =	**100.0**

OK

OK

Output Requests...

Units Scale for Output Temperatures	◆ **Celsius**

Print Interval Controls...

Initial Print Interval **30.0**

OK

OK

Apply

Read and plot results

9. Read the results files using **Utilities** and plot results for several time steps. Do not delete the database from your directory since it will be used in a future exercise.

From within MSC/PATRAN the only indication that the analysis has successfully finished is the existence of **nrX.nrf.01** ("x" will range from 1-12) results files in a subdirectory one level below your working directory.

P3 was initiated from a working directory which contained the exercise_010.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*, exercise_10/. By using **Read Result** in the <u>Analysis</u> form and Selecting **Results File...** you can filter down to the *Job Name* subdirectory and check for the existence of a results file

Utilities provides a results reader under **Thermal** which facilitates reading the results of transient runs. If you do not have Shareware installed or is unavailable then skip to the instructions below which tell you how to read the results from the <u>Analysis</u> form.

Utilities

Thermal

Thermal Results Reader...

Disclaimer

OK

Directories **<path>/exercise_10**

Filter

All your **nr.nrf.01** files should appear in the *Selected Files* list box. Edit the end of the path and file in the *Selected File* box to end in nr*.

Selected File **<path>/exercise_10/nr***

OK

Apply

All the files will be read into the database. Since the analysis does not involve convection template ID's, the results template that is used in reading the files detects that there are no such entries and creates a warning message for each occurrence. This warning is irrelevant to this analysis. When the warning messages cease. Click **OK** in the Warning form.

Select **Close** to exit the **Utilities**.

Warning
OK
Close

[OPTIONAL METHOD follows. Use only if necessary]

If for whatever reason **Utilities** are unavailable you can use the standard MSC/PATRAN method of reading the results files. However, a Global Variable for Time will not be created in the Result Cases. This will adversely affect the animation subsequently created in Exercise 19.

◆ Analysis
Read Results/Result Entities
Select Results File...
Directories **<path>/exercise_10**
Filter
Available Files **nr0.nrf.01**
OK
Select Rslt Template File...
Files **pthermal_1_nodal.res_tmpl**
OK
Apply

Once you have selected a *Results Template* you can repeat the **Select Results File...** step and hit **Apply** on the Read Results form without again selecting a template.

Select Results File...
Available Files **nr1.nrf.01**
OK

Apply

Repeat these steps for each **nr#.nrf.01** file until all results are read.

[END OPTIONAL METHOD]

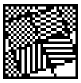
After results are read in by either method, plot the results. To plot the results use the **Results** *Application radio button.* Select results for t=150s.

◆ Results

Create/Quick Plot

Select Result Cases	**TIME: 1.5000000000D+02 S...**
Select Fringe Result	**Temperature,**

Select the *Fringe Attributes* icon.



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

OK

Apply

The posted results are for t=150s. The model should now appear as shown on the front panel of this exercise. Feel free to post results from other times to verify your results.

Do not delete the database when you finish this exercise it will be used in a future exercise. In that exercise we will animate the results to display the model's behavior through time.

Quit MSC/ Patran

10. **Quit** MSC/PATRAN

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