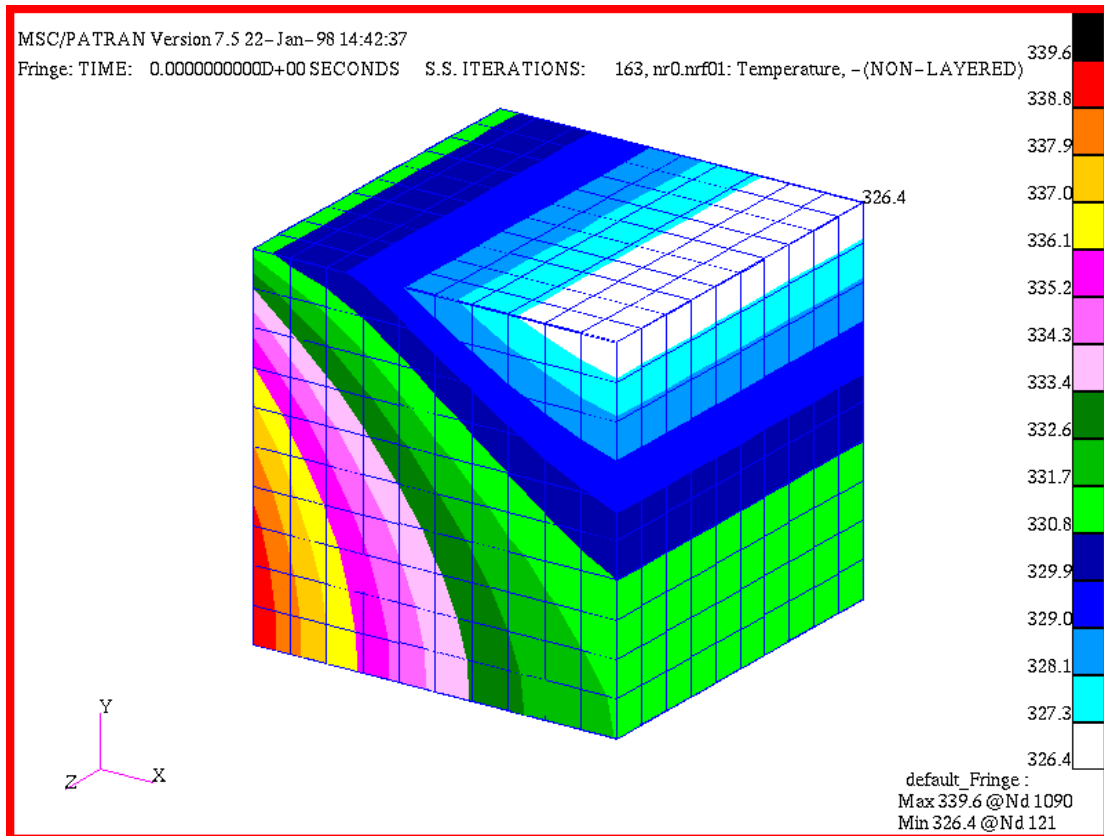


WORKSHOP 11

Using Convection Correlations



Objective:

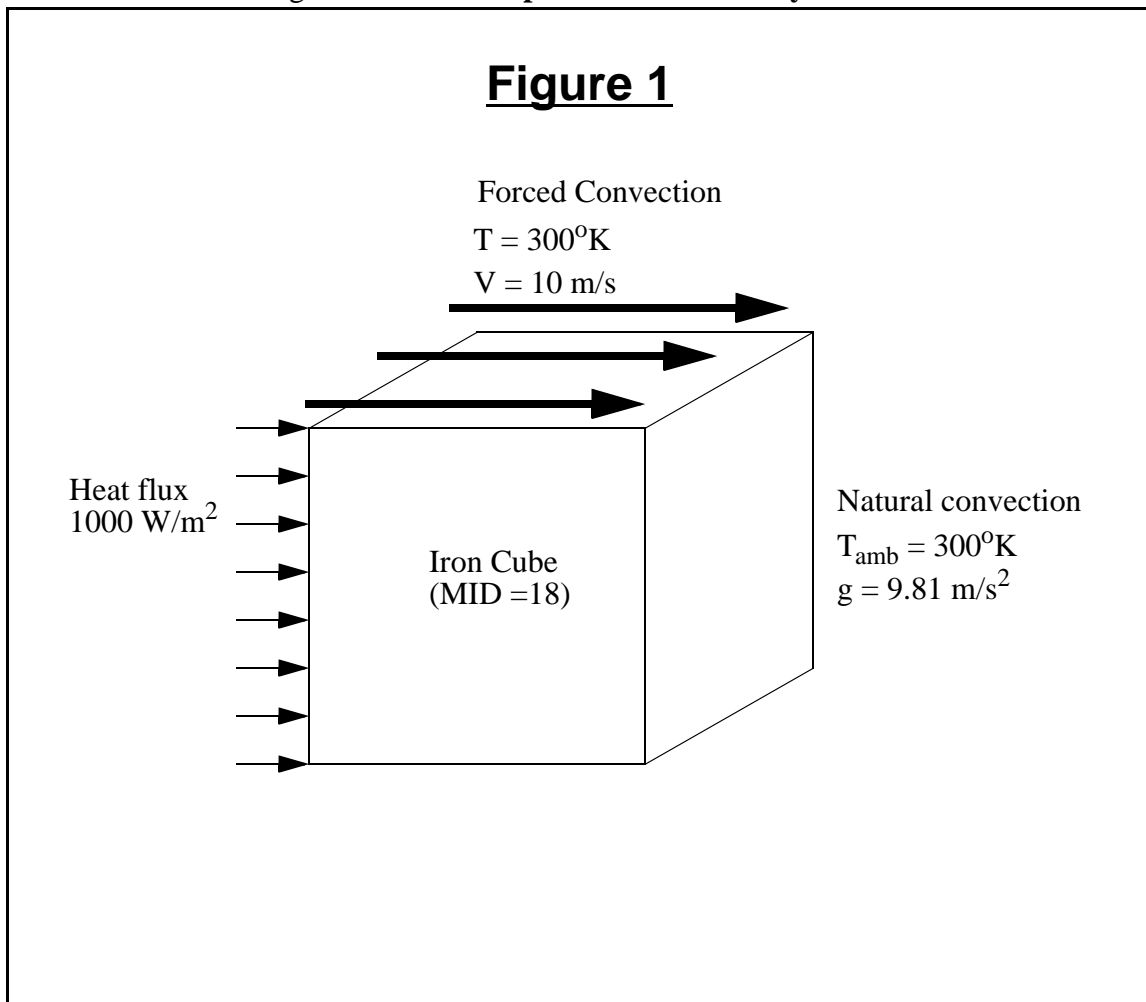
- Model an iron cube.
- Apply convective boundary conditions using correlations from the MSC.Thermal convection correlation library.
- Run a steady state analysis and display results.

Model Description:

In this exercise you will determine the steady state temperature distribution in an iron cube (MID 18). The temperature distribution will be driven by a heat flux on one vertical face, natural convection on another vertical face, and forced convection on the top horizontal face.

CONV definitions link convection Template ID's (TID's) which are applied in the Loads/BC's form to convection configurations and associated Material Property ID's (MPID's). CONV definitions for air will be placed in a **mat.dat.apnd** which you create using an editor. The mat.dat.apnd file is placed in the same directory as your database.

MSC.Patran contains an extensive library of convection coefficient configurations. The configurations are described in Chapter 9, Volume 1 of the MSC.Thermal Application Module User Manual. This volume can be accessed through the on-line **Help/Document Library..**



Exercise Overview:

- Create a new database named **exercise_11.db**. Set *Tolerance* to **Default**, and the *Analysis Code* to **MSC/THERMAL**.
- Create a 1m x 1m x 1m solid.
- Mesh the solid with an **IsoMesh** of **Hex8** elements, *Global Edge Length* of 0.100.
- Use **Finite Elements/Create/Node/Edit** to create a boundary node not associated with geometry.
- Apply element properties to the **Hex8** elements defining them as **Thermal 3D Solid** and having a *Material Name (MID)* of **18**.
- Create a spatial field which will provide distance-from-the-leading-edge data to the convection coefficient calculation.
- Define a fixed temperature and heat flux boundary condition in Loads/BCs.
- Define two convection boundary conditions assigning each a different *Convection Template ID* and supplying the distance from the leading edge in the *Convection Coefficient* data box.
- Use the new **Analysis/Build Template** form to create a **template.dat.apnd** containing the CONV definitions.
- In your xterm window (shell) and in the directory which contains the database, use **get_qtran** to copy an existing file, **mat.dat.apnd**, containing air MPID data to your directory.
- Prepare and submit the model for analysis specifying that it is a steady state, that all calculations and output should be °K, and that all eight columns of nodal results are included in the nodal results file.
- Read and plot the results.
- **Quit** MSC.Patran.

Exercise Procedure:

1. Open a new database named **exercise_11.db**.

Open a new database

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_11.db` to the new database by clicking in the *New Database Name* box and entering **exercise_11**

Select **OK** to create the new database

<input type="button" value="File"/>	
<input type="button" value="New"/>	
New Database Name	<input type="text" value="exercise_11"/>
<input type="button" value="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	<input type="button" value="◆ Default"/>
Analysis Code	<input type="text" value="MSC/THERMAL"/>
<input type="button" value="OK"/>	

2. Create a 1m x 1m x 1m solid.

Create solid geometry

Select the **Geometry Applications radio button**. Create a solid using the following *Action*, *Object*, and *Method*.

<input type="button" value="◆ Geometry"/>	
<input type="button" value="Create/Solid/XYZ"/>	
Vector Coordinate List	<input type="text" value="<1 1 1 >"/>
<input type="button" value="Apply"/>	

Select **Viewing** from the *Menu Bar* or use the *ToolBar Iso 1 View* icon to change to an isometric_view.

Viewing

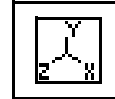
Named View Options...

Select Named View

isometric_view

Close

or,



IsoMesh the solid

3. Mesh the solid with an **IsoMesh** of **Hex8** elements, *Global Edge Length* of **0.1**.

Select the **Finite Elements Applications** radio button. Set the *Action*, *Object*, and *Type* to **Create/Mesh/Solid**. For the solid list, select **Solid 1**.

◆ Finite Elements

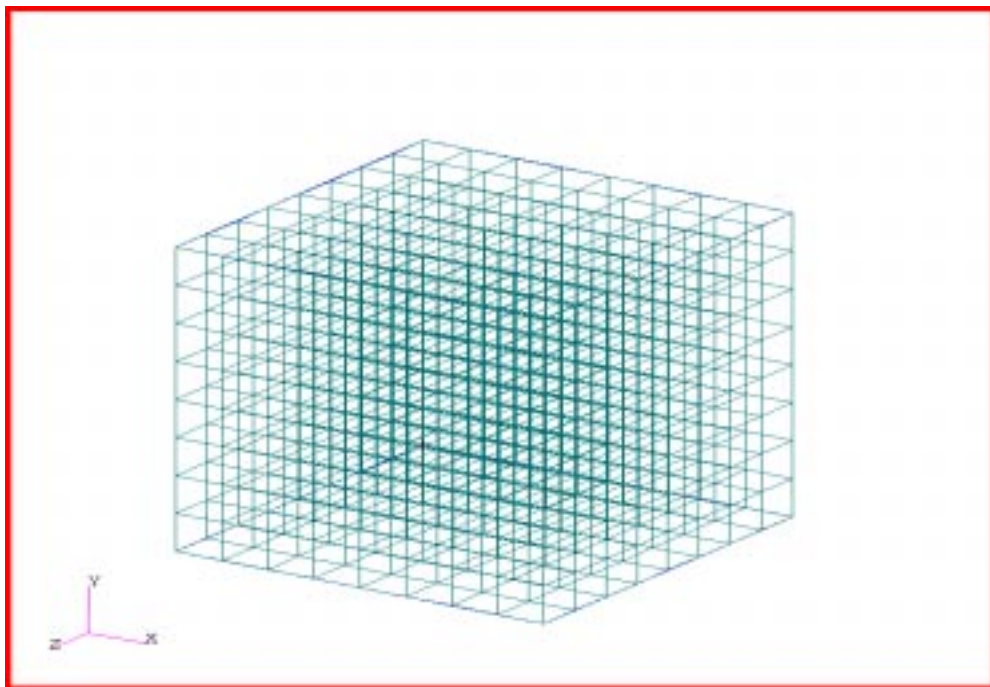
Create/Mesh/Solid

Solid List

<click on Solid 1 in the viewport>

Apply

The resulting model is shown below.



- Use **Finite Elements Create/Node/Edit** to create a boundary node **9999** not associated with geometry.

Create a boundary sink node

In the **Finite Elements** form create a boundary node which is **not associated with geometry**. The node is numbered **9999**. Locate the node at **[1.3 1.3 0.5]**.

◆ **Finite Elements**

Create/Node/Edit	
Node ID List	9999
<input type="checkbox"/> Associate with Geometry	<turn it off>
Node Location List	[1.3 1.3 0.5]
Apply	

Increase the display size of nodes. Use either **Display Finite Element** or the associated *ToolBar* icon to change the node size.

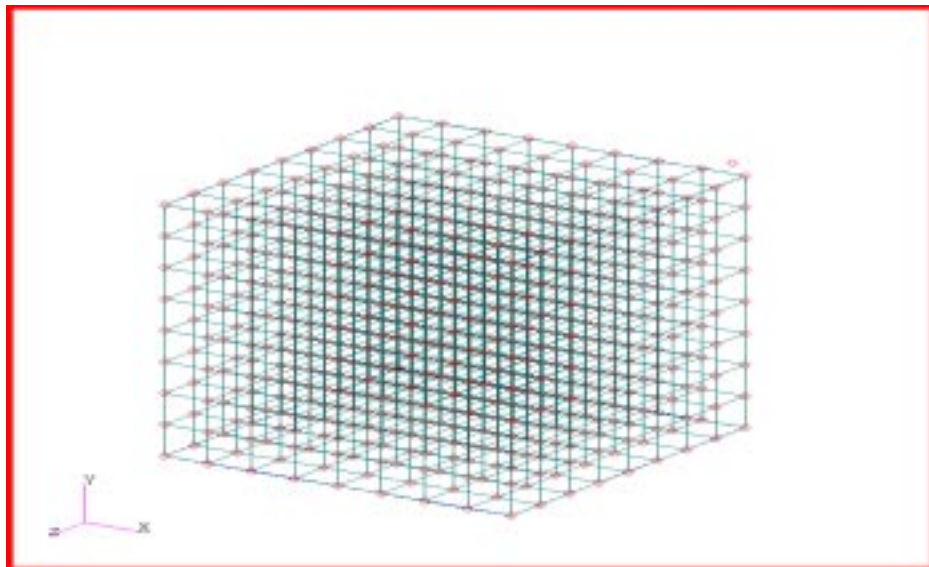
Display	
Finite Elements...	
Node Size	6 <use slider bar>
Apply	
Cancel	

or,

ToolBar Node size



The display should now appear as shown below.



Apply element properties

- Apply element properties to the **Hex8**'s defining them as **Thermal 3D Solid** and having a material name (MID) of **18**.

Select the **Properties Applications** radio button. Set the *Action*, *Dimension*, and *Type* to **Create/3D/Thermal 3D Solid**. Enter *Property Set Name* **Prop1**. Select the *Input Properties...* box. Click in the *Material Name* box and enter **18**. Select **OK** to close the form. Click in the *Select Members* box and select **Solid 1** in the viewport. Select **Add** then **Apply** in the Element Properties form to complete the element property definition.

◆ Properties	
Create/3D/Thermal 3D Solid	
Property Set Name	Prop1
Input Properties...	
Material Name	18
OK	
Select Members	<select Solid 1 in the viewport>
Add	
Apply	

- Create a spatial field which will provide distance-from-the-leading-edge data to the convection coefficient calculation.

In the forced convection boundary conditions the heat transfer coefficient varies as the thermal boundary layer develops from a leading edge. One input to the convective correlation is the distance from the leading edge of the surface. MSC.Thermal provides for spatial fields to define the element distance from the leading edge. You will create a spatially varying field that will define the required distance.

Click on the **Fields** toggle. Set the *Action*, *Object*, and *Method* to **Create/Spatial/PCL Function**. Enter, **X_dist**, in the *Field Name* box. Next, click in the *Scalar Function* text box and select **'X** from Independent Variable list box. Since the global coordinate system's origin is located at a lower left corner of the Solid the simple function, $f(x)=x$, represents the horizontal distance from the leading edge of the top surface where the forced convection heat transfer will occur. Click on **Apply** to create the field.

The natural convection coefficient calculation uses the characteristic length L of the vertical side. This will be input in the **template.dat.apnd** file GP list which will be described later in the exercise.

◆ Fields	
Create/Spatial/PCL Function	
Field Name	X_dist
Independent Variable	'X
Apply	

The field name should appear in the *Existing Fields* list box.

7. Create fixed temperature and heat flux boundary condition in Loads/BC's.

Select the **Loads/BCs Applications** radio button. Create a fixed **300°K** nodal boundary temperature named **Tamb**. In the Input Data form define the fixed temperature. In the Select Application Region form pick **Node 9999**.

Apply temperature and flux boundary conditions

◆ Loads/BCs	
Create/Temperature/Nodal	
Option:	Fixed
New Set Name	Tamb
Input Data...	
Fixed Temperature	300.0
OK	
Select Application Region...	
Geometry Filter	◆ FEM
Select Nodes	<select Node 9999>
Add	
OK	
Apply	

Create a set name **Flux** of 1000 W/m^2 . Apply the boundary condition to the left facing (-X normal) surface of the solid, Solid 1.1, as shown in Figure1.

◆ **Loads/BCs**

Create/Heating/Element Uniform

Option:

Flux, Fixed

New Set Name

Flux

Target Element Type

3D

Input Data...

Heat Flux

1000

OK

When selecting a surface, the surface chosen will be highlighted. Hold down the *<Shift>* key and use the right mouse button to cycle through surfaces that may overlap or share an edge.

Select Application Region...

Geometry Filter

◆ **Geometry**

Select Solid Faces

<select (-X normal) face of Solid 1, (Solid 1.1), use shift-right mouse button to cycle pick, if necessary>

Add

OK

Apply

- Define two convection boundary conditions assigning each a different *Template ID* and supplying the distance from the leading edge in the *Convection Coefficient* data box.

Create the convection coefficient boundary conditions with the **Use Correlations** option, *New Set Name* **forced_convection**, with a fluid node **9999**, and a *Template ID* of **93**. Apply the boundary condition to the top face (+Y normal) of Solid 1 as shown in Figure1.

**Create two
convection
Template ID's**

Spatial field **X_dist** will be used in the *Convection GP2/GP3* data field to supply the leading edge distance to the correlation.

◆ **Loads/BCs**

Create/Convection/Element Uniform	
Option:	Template, Convection
New Set Name	forced_convection
Target Element Type	3D
Input Data...	

In the Input Data form provide the *Convection Template ID*, fluid node, and Geometric Properties 2 and 3 via the *Convection GP2/GP3* data box.

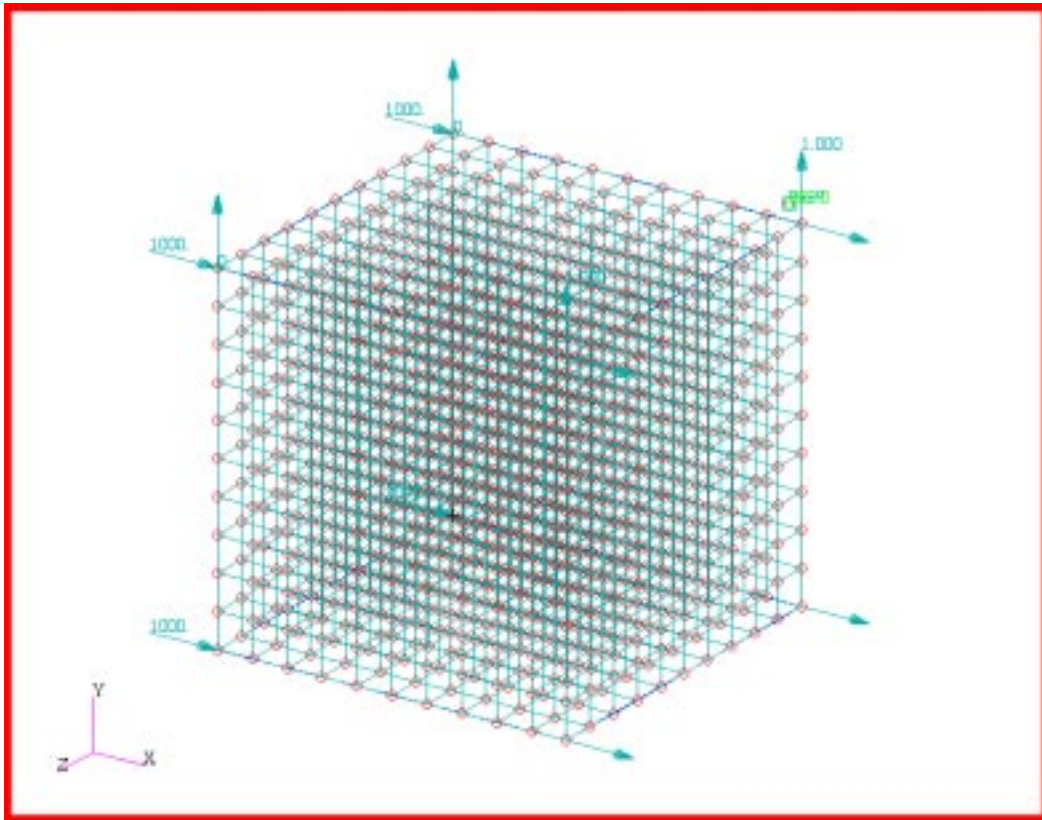
<input type="checkbox"/> Fixed	<deselect>
Select Spatial Field...	
CONV GP2/GP3	<select X_dist from the Spatial Fields list box>
Close	
Convection Template ID	93
Fluid Node ID	<select node 9999>
OK	
Select Application Region...	

In the Select Applications Region form select the top face (+Y normal) of Solid 1.

Geometry Filter	◆ Geometry
Select Surfaces or Edges	<select the top face (+Y normal) of Solid 1, (Solid 1.4), as shown in Figure 1>
Add	
OK	
Apply	

Repeat these steps for a New Set Name natural_convection. Click in the *Convection GP2/GP3* data box. Leave the *CONV GP2/GP3* blank. Use a *Convection Template ID* of **913**, and a *Fluid Node* of **9999**. In the Select Applications Region form select the right face (+X normal) of **Solid 1**, (**Solid 1.2**).

With boundary conditions applied the model should appear as shown below



You may also choose to review your loads and boundary conditions using **Utilities/Thermal/Thermal BCs Display**.

Utilities

Thermal

Thermal BC Display...

A disclaimer message may appear, select **OK**..

OK

Apply

Use **Clear** and **Close** in the Thermal BC Display form to revert to normal display.

Clear

Close

- Use the new **Analysis/Build Template** form to create a **template.dat.apnd** containing the CONV definitions.

Using **Analysis/Build Template**, create and edit the file **template.dat.apnd** in the directory which contains your database and where MSC.Patran is running.

**Create
template.dat.
apnd file**

Use the following table to help you define the convection templates for the forced and natural convection boundary conditions. You should also review the definition of the convection configuration 3 and 13 in the MSC.Thermal USERS MANUAL, Volume 1, Chapter 9.

Conv Type	TID#	config#	GP Value	MPID's (order is important)
Forced	93	3	10.0	550100, 550101, 550103, 550105
Natural	913	13	1.0, 1.0, 0.0, 9.8	550100, 550101, 550106, 550103, 550105

◆ Analysis

Build Template

Create Template File...

Create/CONV/Data Entry

CONV ID:

93

CFIG ID

3

Info

GP Values

10.0

MPID's

550100 550101 550103 550105

Apply

Cancel

CONV ID:

913

CFIG ID

13

Info

GP Values

1.0 1.0 0.0 9.8

MPID's

**550100 550101 550106 550103
550105**

Apply

Write File...

OK

Cancel

Close

Cancel

<close Configuration Info window>

<close Template Data Form>

Shown below is the final form of the **template.dat.apnd** file created for this exercise.

```
*=====
CONV 93 3 1 4
10.0
550100 550101 550103 550105
CONV 913 13 4 5
1.0 1.0 0.0 9.8
550100 550101 550106 550103 550105
*=====
```

**In unix copy a
mat.dat.apnd
file**

10. In your xterm window (shell) and in the directory which contains the database, use **get_qtran** to copy an existing file, **mat.dat.apnd**, containing air MPID data to your directory.

Copy a **mat.dat.apnd** file into the same directory in which you've created the **template.dat.apnd** file. The commands are as follows:

> get_qtran

Note: If you receive a "command not found" error then it indicates that you do not have a path to P3.HOME. In this case type:

> which p3

(response) > <path>/bin/p3

Then type in:

> <path>/bin/get_qtran

(response) > Enter a problem directory name or <cr> to exit:

> prob4

(response) > Enter a filename (* to copy all files) or <cr> to exit:

> mat.dat.apnd

(response) > Copying mat.dat.apnd

A **mat.dat.apnd** should now reside in your database subdirectory. This file contains more material properties than required. This will not adversely affect the analysis. Feel free to review the format and syntax of the mat.dat.apnd file. You can use this file as a boiler plate for creating your own material properties file data.

The **template.dat.apnd** and **mat.dat.apnd** files are the only two files that may need to be created outside of the MSC.Patran in order to complete an analysis. As MSC.Patran evolves the creation of this files will be absorbed within the MSC.Patran interface.

11. 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.

**Prepare and
run analysis**

◆ Analysis

Analyze/Full Model/Full Run

Translation Parameters...

OK

Solution Parameters...

Run Control Parameters...

Initial Temperature =

300.0

Initial Temperature Scale

◆ Kelvin

OK

Units Scale for Output Temperatures Select Thermal Entries to Output

12. Read and plot the results.

From within MSC.Patran the only indication that the analysis has successfully finished is the existence of an **nr0.nrf.01** results file in a subdirectory one level below your working directory.

P3 was initiated from a working directory which contained the exercise_11.db database. Applying the analysis created a new subdirectory with the same name as the *Job Name*, exercise_11/. 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 a results file.

Directories

Available Files

Files

Read and plot results

There may be a warning message regarding Qmacro, select **OK**.

OK

Reduce the node size using *Node Size* icon.



After results are read in plot the results. To plot the results use the **Results Application radio button**. Select you results file.

◆ Results

Create/Quick Plot

Select Result Cases

Select Fringe Result

TIME: 0.000000000D+00 S...

Temperature,

Select the *Fringe Attributes* icon.



Display:

Element Edges

Label Style...

Label Format:

Fixed

Significant figures

4 <use slider bar>

OK

Apply

The model should now appear as shown on the front panel of this exercise. Feel free to plot the value of the heat transfer coefficient (and other quantities). Select the *Select Result* icon.



To plot the heat transfer coefficient data:

Select Result Case

Time: 0.000000000D+00 S...

Select Fringe Results

Average Convection Coefficient

Apply

The nodal averaged h's are displayed in the viewport.

To view detailed convection resistor data look in **qout.dat.01** file in the *Job Name* subdirectory. Search for string "CONVECTIVE RESISTOR DATA."

13. **Quit** MSC.Patran

To stop MSC.Patran select **F**ile on the *Menu Bar* and select **Q**uit from the drop-down menu.

Quit
MSC.Patran