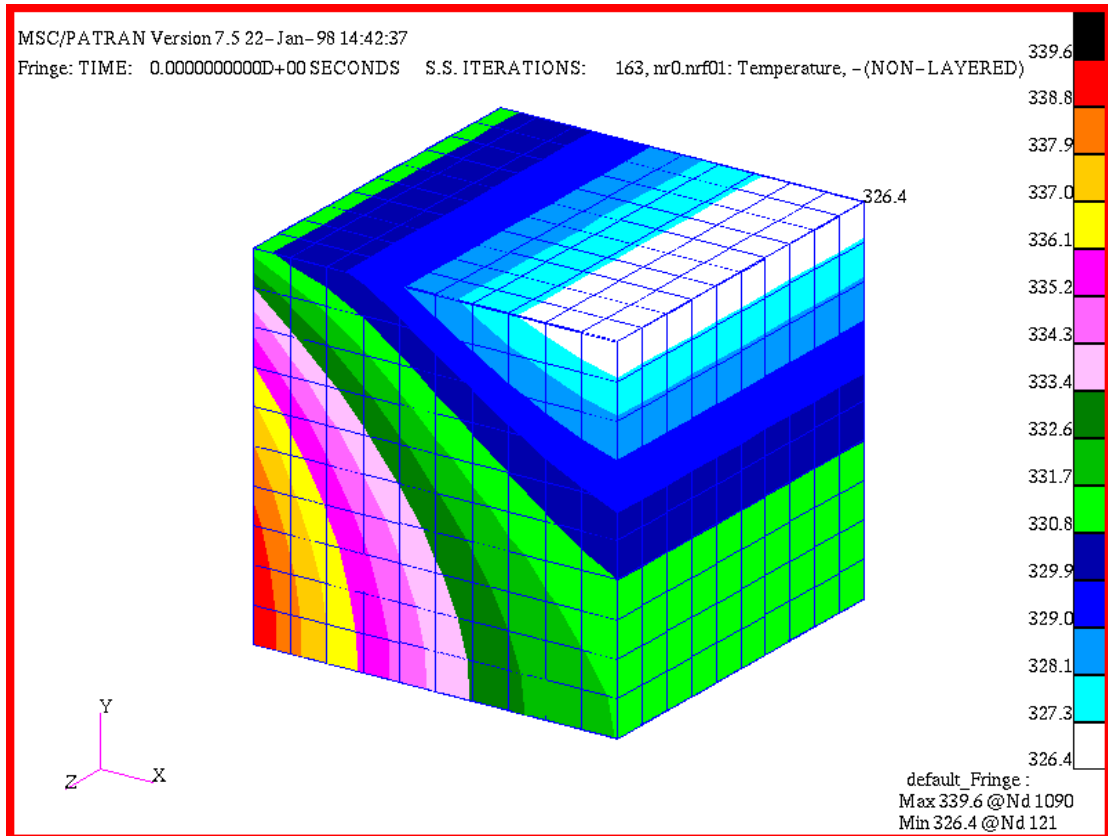


Exercise 11

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.

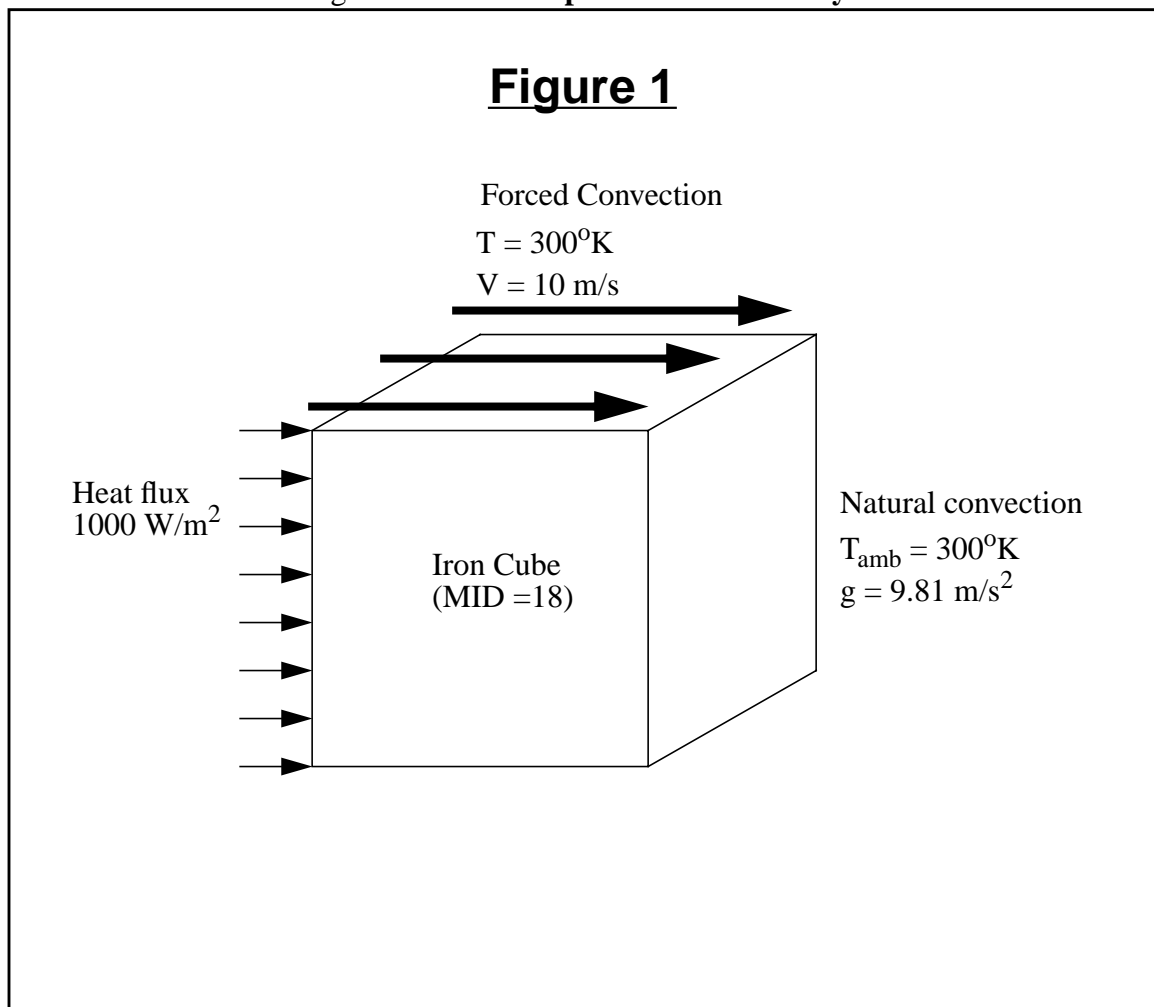


Model Description:

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 are edited into a **template.dat.apnd** file which you create in the same directory as your database. MPID's for air will be placed in a **mat.dat.apnd** which you also create.

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.
- Open a new window (shell) and in the directory which contains the database edit a file named **template.dat.apnd** creating the CONV definitions.
- In your xterm window (shell) and in the directory which contains the database, copy an existing file, <P3_HOME>/p3thermal_files/examples/qtran/prob4/**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**.

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

File	
New	
New Database Name	exercise_11
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 1m x 1m x 1m solid.

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

◆ Geometry	
Create/Solid/XYZ	
Vector Coordinate List	<1 1 1 >
Apply	

Open a new database

Create solid geometry

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

Viewing

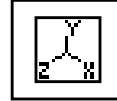
Named View Options...

Select Named View

`isometric_view`

Close

or,



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**

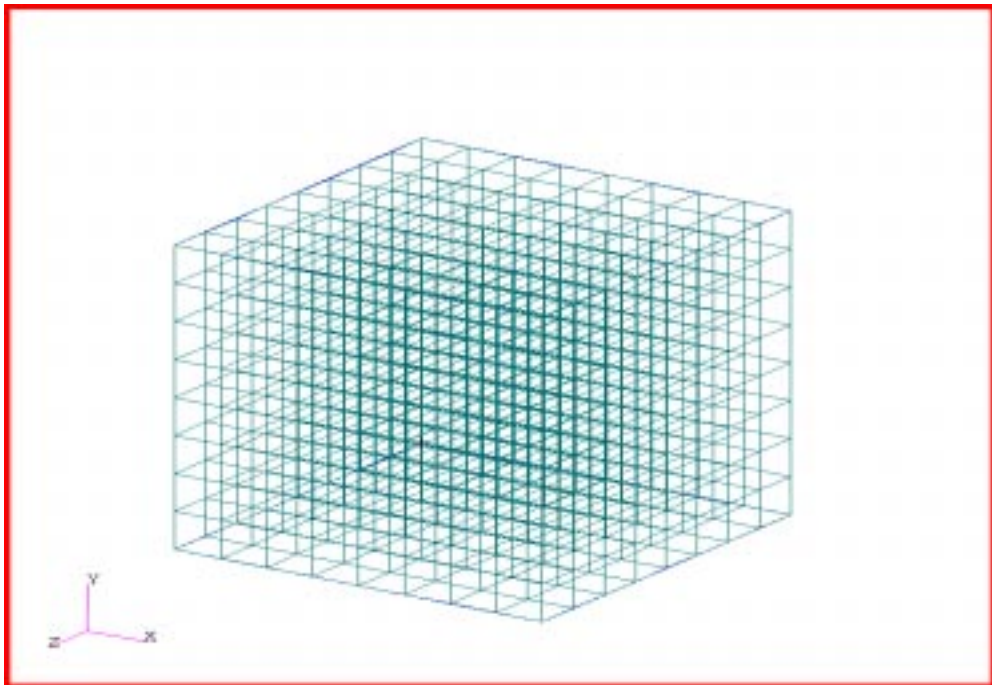
Create/Mesh/Solid

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

Associate with Geometry

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 Element

Node Size

6 <use slider bar>

Apply

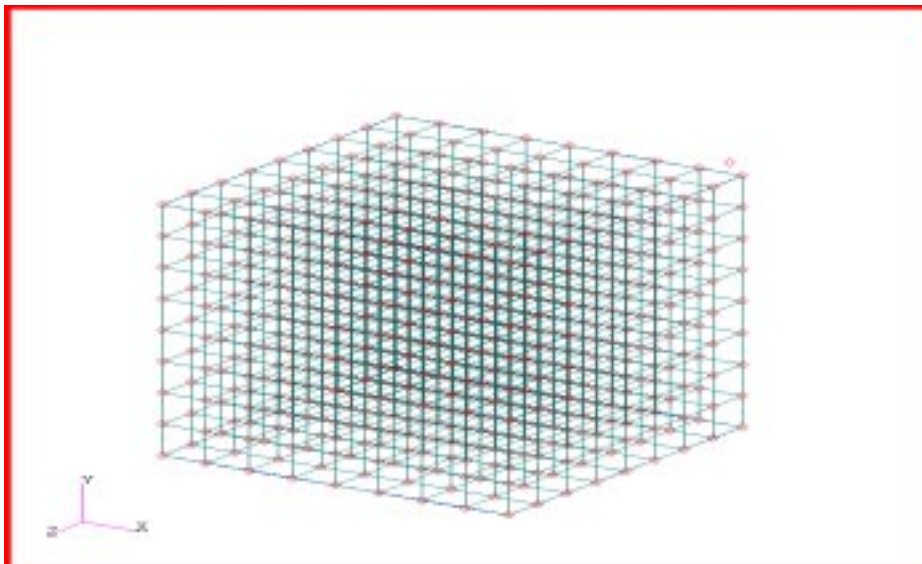
Cancel

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 **Load/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

◆ Load/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/m²**. Apply the boundary condition to the left facing (-X normal) surface of the solid, Solid 1.1, as shown in Figure1.

◆ Load/BCs

Create/Heating/Element Uniform	
Option:	Fluxes
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 Coefficient* data field to supply the leading edge distance to the correlation.

◆ **Load/BCs**

Create/Convection/Element Uniform	
Option:	Use Correlations
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 Coefficient* data box.

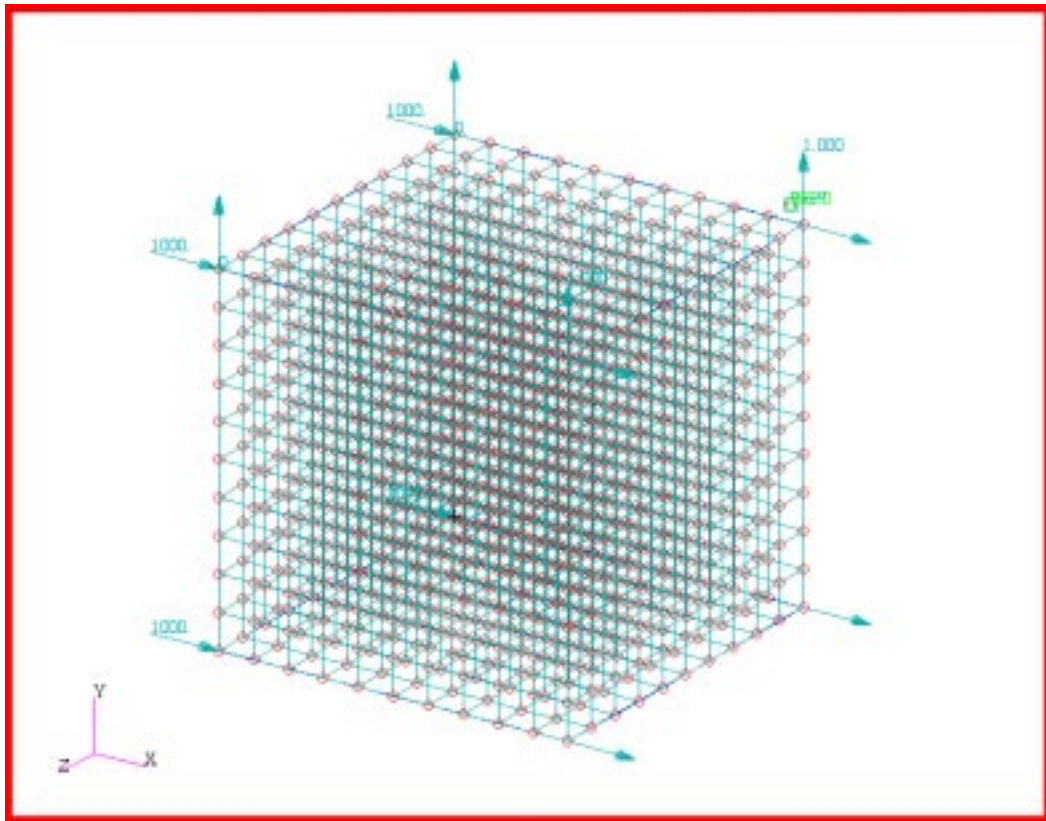
Convection Coefficient	<select X_dist from the <i>Spatial Fields</i> list box>
Convection Template ID	93
Fluid Node ID	9999
OK	
Select Application Region...	

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 Coefficient* data box. Leave the *Convection Coefficient* 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

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

Open a unix xterm window and change directories (cd) to the directory which contains your database.

If a **template.dat.apnd** already exists in this directory rename it to associate it with that previous analysis. For instance, in Exercise 10 you created a **template.dat.apnd** file. Use the following unix command to move it to a new name associated with that analysis:

```
> mv template.dat.apnd 10_template.dat.apnd
```

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.

Following is a detailed description of what you will be entering into the file. The actual entries are fairly brief and listed following the detailed description of the complete syntax.

Create two convection templates, one for the forced_convection and the other for the natural_convection boundary condition. The format for the CONV functions are as follows:

CONV TID# config# number_of_GP_values number_of_MPID_values

gp#2	gp#3	... gp#n
mpid#1	mpid#2	... mpid#n

Where, **TID:**

The template ID number (pointer) you entered in the Load/BCs form.

config#:

Convection type configuration number.

Identifies the type of convection class (e.g. 1=forced convection, smooth Isothermal tubes, 3=flat plate, forced convection, etc.).

number_of_GP_values:

Number of general properties needed for a specific convection configuration.

Example: **For config=3 (forced convection over a horizontal surface)**

GP(1) = element surface area (automatically supplied by MSC/PATRAN)

GP(2) = shortest distance from plate's leading edge to element.

GP(3) = longest distance from plate's leading edge to element.

GP(4) = free stream velocity

**In unix create
template.dat.
apnd file**

number_of_MPID_values:

Number of Material Property ID numbers used to point to material properties.

Example: **For config = 3 (mpid#'s point to)**

MPID(1) = fluid density, ρ

MPID(2) = fluid dynamic viscosity, μ

MPID(3) = fluid specific heat, C_p

MPID(4) = fluid thermal conductivity, κ

Note: **CONV** is a keyword and must be typed in uppercase.

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	#MPID	MPID's (order is important)
Forced	93	3	1	4	550100, 550101, 550103, 550105
Natural	913	13	4	5	550100, 550101, 550106, 550103, 550105

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. Start typing from the first column and do not enter any blank lines.

```
*=====
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 your xterm window (shell) and in the directory which contains the database, copy an existing file, <path>/**p3thermal_files/examples/qtran/prob4/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:

```
> which p3
```

In unix copy a
mat.dat.apnd
file

(response) > <path>/bin/p3

> cp <path>/p3thermal_files/examples/qtran/prob4/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 Scale	◆ Kelvin
Initial Temperature =	300.0
OK	
OK	
Output Requests...	
Units Scale for Output Temperatures	◆ Kelvin
Nodal Results File Format...	
Select Thermal Entries to Output	<select all 8 items listed>
OK	
Diagnostic Output	
OK	
	◆ Convection Resistors

OK
Apply

12. Read and plot the results.

From within MCS/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.

◆ Analysis

Read Results/Result Entities

Select Results File...

Directories

Filter

Available Files

OK

Select Rslt Template File...

Files

OK

Apply

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	TIME: 0.0000000000D+00 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 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) using *Select Fringe Result*.

To plot the heat transfer coefficient data:

◆ Results	
Form Type:	Basic
Select Result Case	2.1-Time: 0.0000000000D
Select Fringe Results	6.1-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

Quit MSC/ Patran

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