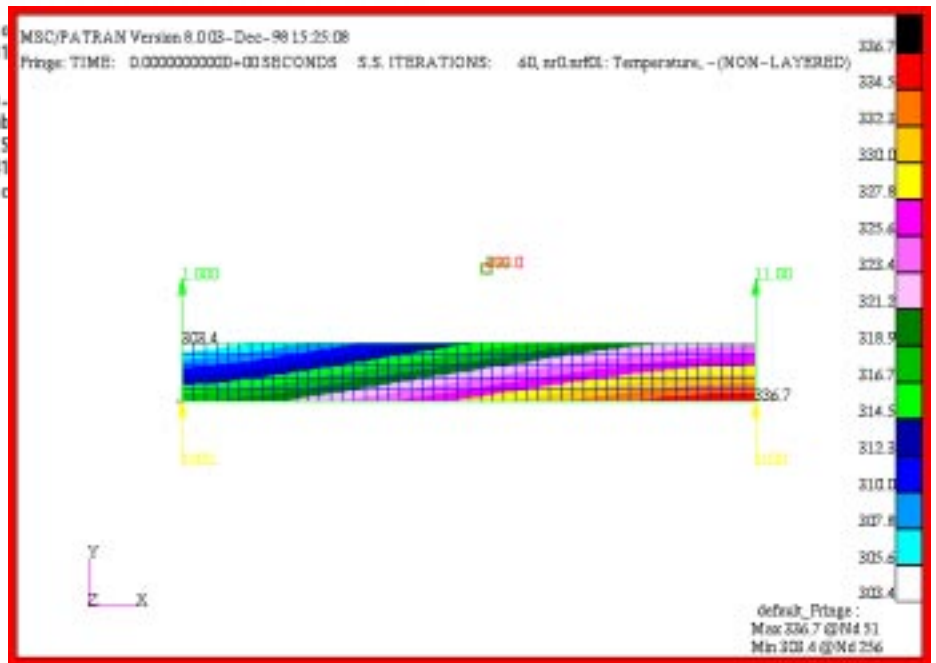


WORKSHOP 15

User Supplied Subroutines

```
=====
C
C   Set up the target statement label for configuration 1000.
C
C=C   1000 CONTINUE
      DOUBLE PRECISION RL, AREA
      RL = (GP(IRESIS,2)+GP(IRESIS,3))/2.0
      AREA = GP(IRESIS,1)
      H = (T1 + 100.0) / RL
      GVALH = H * AREA
      Q = GVALH * (T1-T2)
C
C   This configuration is going to
C   configuration number 31
C
C   Get the resistor surface area.
C   Geometric Property number
C   resistor (number IRESIS
C   of the GP array. GP #1
C   be in GP(IRESIS,2), etc
C
C=C   AREA = GP(IRESIS,1)
C
```



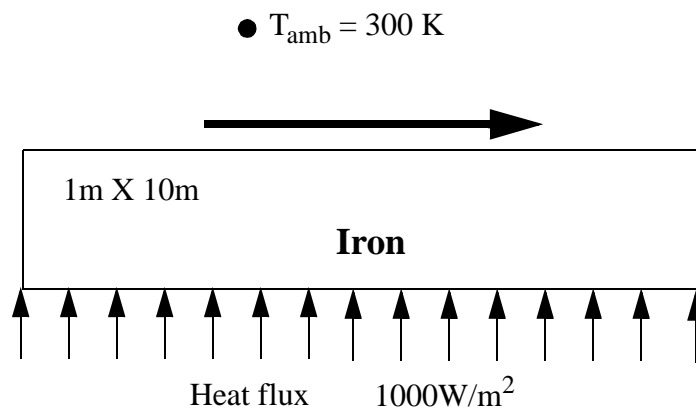
Objective:

- Create a user subroutine UHVAL that computes the values for the heat transfer coefficient.

Model Description:

In this exercise the Convection Correlation will be supplied by the user. You will write the necessary code to evaluate the Convection Correlation in the user subroutine UHVAL. This Qtran subroutine will compute the values for the heat transfer coefficient and return those values to the main program.

An iron slab is modeled in 2-dimensions. A heat flux of 1000W/m^2 is imposed on the bottom edge of the slab. The top surface convects heat to the ambient temperature at 300K with a heat transfer coefficient defined by a user supplied subroutine.



Exercise Overview:

- Start MSC.Patran and create a new database named, **exercise_15.db**.
- Create the 2D model geometry.
- Mesh the geometry with Quad4 elements and a *Global Edge Length* of **0.2**.
- Create an ambient node for convection boundary conditions.
- Apply properties.
- Create values for distance from the leading edge using **Fields** and **Create/Spatial/PCL Function**.
- Define boundary conditions in Loads/BCs.
- Copy **ulib.f**.

- Modify and compile **ulib.f**.
- Prepare and submit the model for analysis.
- Read the results file and plot temperature and heat transfer coefficient results.
- **Quit** MSC.Patran.

Exercise Procedure:

Open a new database

1. Open a new database named **exercise_15.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 *Top Menu Bar* and select **New...** from the drop-down menu. Assign the name `exercise_15.db` to the new database by clicking in the *New Database Name* box and entering **exercise_15**.

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_15"/>
<input type="button" value="OK"/>	

MSC.Patran will open a Viewport and change various *Control Panel* 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"/>	

Create plate geometry

2. Create the 2D model geometry.

To create the Surface that will represent the geometry of the 2D-model click on **Geometry** in the *Main Window* and set the *Action*, *Object*, and *Method* respectively to **Create**, **Surface**, and **XYZ**. Change the *Vector Coordinates List* to **<10, 1, 0>** and click on the **Apply** button to create the Surface.

◆ Geometry

Create/Surface/XYZ

Vector Coordinate List

<10 1 0>

Apply

3. Mesh the surface with an IsoMesh of quad4 elements, global edge length of 0.2.

IsoMesh the surfaces

Select the **Finite Elements Applications Radio Button** and set the *Action*, *Object*, and *Type* respectively to **Create**, **Mesh**, and **Surface**. Enter, **0.2**, for the *Global Edge Length* of the **Quad4** elements you are now creating. Click in the *Surface List* box and then select **Surface 1** in the viewport. Click on **Apply** to create the element.

◆ Finite Elements

Create/Mesh/Surface

Global Edge Length

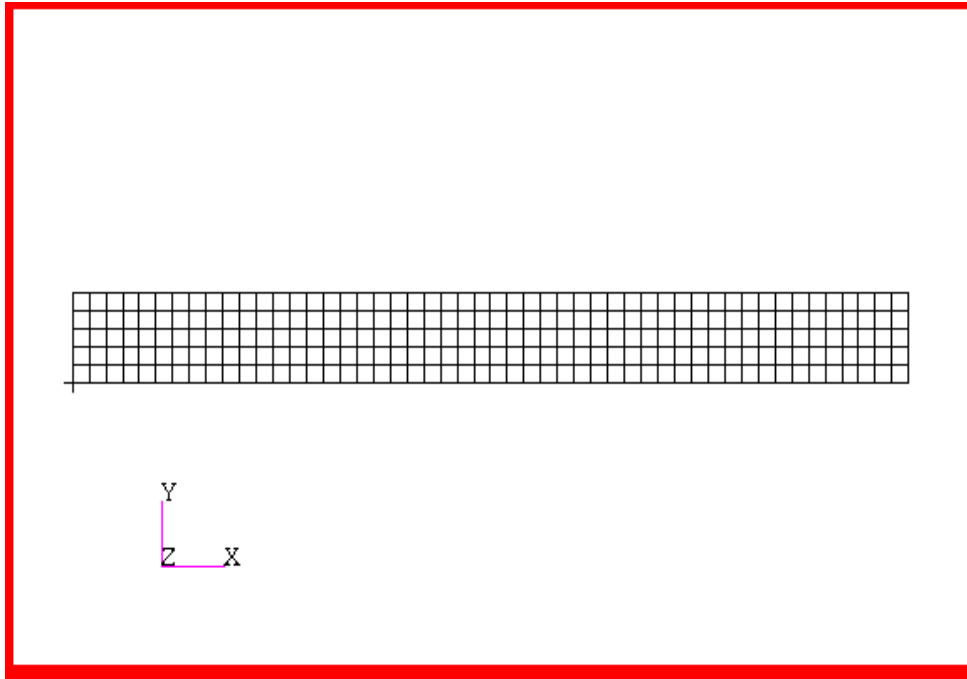
0.2

Surface List

<click on Surface 1 in the viewport>

Apply

The display should now appear as shown below.



Create Ambient Node

4. Create an ambient node for convection.

To create the 'Ambient' Node, click on **Finite Elements** in the *Main Window*. Set the *Action*, *Object*, and *Type* respectively to **Create**, **Node**, and **Edit**. Change the *Node Id List* to **999**, set the *Associate with Geometry* option to off, and then click in the *Nodal Location List*. Select the *Screen Position* select icon in the *Select Menu* (the right-most icon) and then locate the Node somewhere above the model's center. Click on **Apply** to create **Node 999**.

◆ **Finite Elements**

Create/Node/Edit

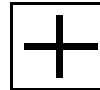
Node Id List

999

Associate with Geometry (toggle off)

Node Location List

<use Screen Position icon>



and <select somewhere above the model's center>

To better visualize the Node's location, set the *Node Radius* to **6 (Display/Finite Element)**. Or use the Tool Bar *Node Size* icon.



- Apply properties to the quad4's defining them as **2D/Thermal 2D** elements having a material MID of 18.

Select the **Properties Applications Radio Button**. Set the *Action*, *Dimension*, and *Method* to **Create/2D/Thermal 2D**. 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 Surface 1 in the viewport. Select **Add** then **Apply** in the *Element Properties* form to complete the element property definition.

Apply element properties

◆ Properties	
Create/2D/Thermal 2D	
Property Set Name	Prop1
Input Properties...	
Material Name	18
Ok	
Select Members	<select Surface 1 in the viewport>
Add	
Apply	

- Create values for distance from the leading edge using **Fields** and **Create/Spatial/PCL Function**.

Spatial functions can be created in the Fields form using the *Action/Object/Method* **Create/Spatial/PCL Function**.

Create micro-functions

◆ Fields	
Create/Spatial/PCL Function	
Field Name	X_Dist
Scalar Function	'X+1.0
Apply	

The screenshot shows the 'Fields' dialog box with the following settings:

- Action: Create
- Object: Spatial
- Method: PCL Function
- Existing Fields: (Empty list)
- Field Name: X_Dist
- Field Type: Scalar
- Coordinate System Type: Real
- Coordinate System: Coord 0
- Scalar Function ('X', 'Y', 'Z'): X+1.0

Apply boundary conditions

7. Define boundary condition in Loads/BCs.

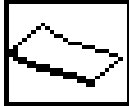
To assign a heat flux of **1000 W/m²** to the bottom of **Surface 1**, click on **Loads/BCs** in the *Main Window*. Set the *Action*, *Object*, and *Type* respectively to **Create/Heating(P Thermal)/Element Uniform**.

Enter, **Bott_Surf_Flux**, for the *New Set Name* and then set the *Target Element Type* to **2D**. Select **Input Data** and enter **1000** for *Heat Flux*. Click on **OK** to close the *Input Data* form. Next, click on the **Select Application Region** button and set the *Geometry Filter* to **Geometry** in the *Select Application Region* form. Click in the *Select Surfaces or Edges* box and then select the *edge* selection icon (second icon from left) in the *Select Menu*. Select the bottom edge of **Surface 1** in the viewport. Click on the **Add** and **OK** buttons to close the form.

Click on the **Apply** button to create the Heat Flux boundary condition.

◆ Loads/BCs
Create/Heating/Element Uniform
Option:
New Set Name

Flux, Fixed
Bott_Surf_Flux

Target Element Type:	<input type="text" value="2D"/>
Input Data...	
Fixed Heat Flux	<input type="text" value="1000"/>
OK	
Select Application Region...	
Select Menu	<Edge icon>
	
Select Surfaces or Edges	<select Surface 1.4>
Add	
OK	
Apply	

Next, Apply the **300K** ambient temperature to **Node 999** by clicking on **Loads/BCs** in the *Main Window*. Set the *Action*, *Object*, and *Type* respectively to **Create**, **Temp(P Thermal)**, and **Nodal**. Switch the *Option*: to **Fixed**. Enter, **Temp_amb**, for the *New Set Name*. Click on the **Input Data** button and enter **300** for the *Fixed Temperature*. Click on **OK** to close the *Input Data* form. Next, click on the *Select Application Region* button. Set the *Geometry Filter* to **FEM** and then click in the *Select Nodes* box. Select **Node 999** in the viewport. Click on the **Add** and **OK** buttons to close the form. In the *Load/Boundary Conditions* form click on the **Apply** button to assign the temperature to Node 999.

◆ **Loads/BCs**

Create/Temperature/Nodal	
Option:	<input type="text" value="Fixed"/>
New Set Name	<input type="text" value="Temp_amb"/>
Input Data...	
Fixed Temperature	<input type="text" value="300"/>
OK	
Select Application Region...	

Geometry Filter	◆ FEM
Select Nodes	<select node 999>
Add	
OK	
Apply	

To apply the (yet to be defined) Convection Coefficient click on **Loads/BCs** in the *Main Window* and set the *Action*, *Object*, and *Type* respectively to **Create/Convection/Element Uniform**, *option: Template, Convection*.

Enter, **Conv_Coeff_Spatial**, in the *New Set Name* box and then change the *Target Element Type* to **2D**. Next, click on the **Input Data** button, deselect "Fixed", and enter the field **X_Dist** for the *Conv GP2/GP3*, a *Convection Template ID* of **1**, and **999** for the *Fluid Node ID*. Click on **OK** to close the *Input Data* form.

Click on the **Select Application Region** button in the *Load/Boundary Condition* form. Using the *Select Application Region* form select the top edge of Surface 1. Click on the **Add** and **OK** buttons to close that form. Finally click on **Apply** to assign the boundary condition.

◆ Loads/BCs

Create/Convection/Element Uniform	
Option:	Template, Convection
New Set Name	Conv_Coeff_Spatial
Target Element Type:	2D
Input Data...	
<input type="checkbox"/> Fixed	(deselect)
Select Spatial Field...	<select X_Dist>
Close	
CONV GP2/GP3	f:X_Dist
Convection Template ID	1
Fluid Node ID	<Select Node 999>
OK	
Select Application Region...	
Geometry Filter	◆ Geometry

Select Surfaces or Edges

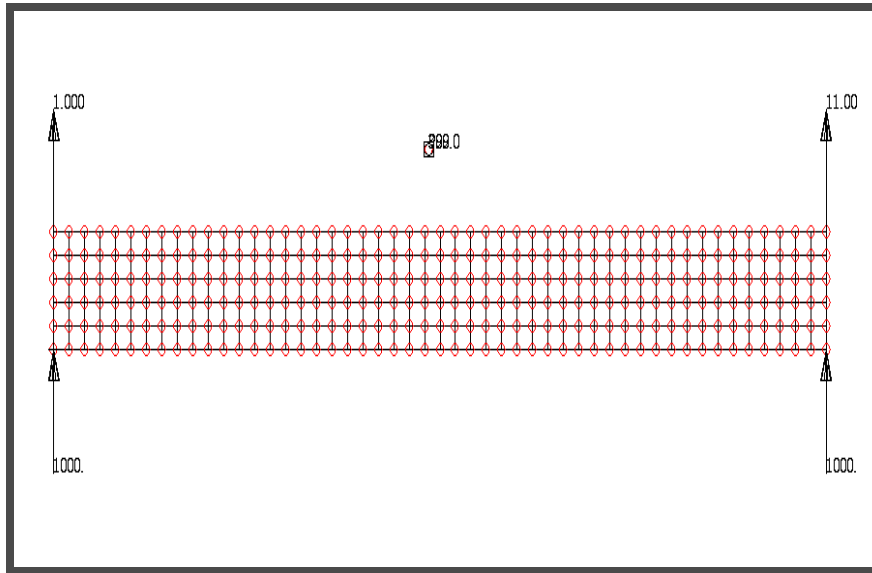
<select top edge of Surface 1, Surface 1.2>

Add

OK

Apply

The current model is shown below.

Reduce the node size with the *Node Size* icon.8. Copy **ulib.f**.

Before you start programming the correlation, you will need to create a job name subdirectory. In the directory you are running MSC.Patran create a subdirectory named, **exercise_15**. Change to that subdirectory.

You will now copy the **ulib.f** file from the MCS.Thermal library. To do so type **get_qtran** in the x-term window. Select **utility** and then **ulib.f**.

```
%mkdir exercise_15
```

```
%cd exercise_15
```

```
%get_qtran
```

Copy ulib.f

Enter a problem directory name or <cr> to exit: **utility**

Enter a filename or <cr> to exit: **ulib.f**

%ls

The **ulib.f** file contains, among others, the **UHVAL** subroutine and several other skeleton or sample subroutines. You will modify the **UHVAL** subroutine to create your own convection correlation.

9. Modify and compile **ulib.f**.

Using the system editor open **ulib.f** and find the *UHVAL* subroutine. Three example configurations, with *CFID*'s of 1000, 1001, 1002, are already programmed but commented out.

Note: The *CFID*'s will be defined in your Convection Template and are passed to the subroutine *UHVAL* with the three configurations. The following two values are always returned from the subroutine:

- i) Conductance: $GVALH = h * Area$
- ii) Heat transferred: $Q = GVALH * (T_1 - T_2))$

In any Convection Configuration you prescribe, these two variables must be calculated and returned to the solver.

To compute the h value ($h=(T_{surf} + 100)/L$) the following two inputs are needed, *RL*, the distance from the leading edge to a particular element, and T_{surf} the temperature of the element edge (surface).

The distance from the leading edge will be passed from the field (*X_Dist*) input in the Convection Coefficient data box in the Loads/BCs form. The average distance from the slab's leading edge to each element will be calculated from:

$$RL = (GP2 + GP3)/2$$

GP2 and GP3 are the distance from the model's leading edge to the leading and trailing edges of each element.

GP1 is automatically passed from MSC.Patran as the cross sectional or surface area of each element.

You will now write the **FORTRAN** code to calculate *GVALH*, *Q*, and *H*. With the systems editor open the **ulib.f** file. Scroll down the file or search for the second occurrence of "UHVAL" in the file to locate the *UHVAL* subroutine. After the following line,

```
C*C          1000  CONTINUE
```

**Modify and
Compile
ulib.f**

type the following lines of code while taking care to place all your code beyond column 7. Remember, this is FORTRAN.

DOUBLE PRECISION RL, AREA

RL = (GP(IRESIS,2) + GP(IRESIS,3))/2.0

AREA = GP(IRESIS,1)

H = (T1+100.0)/RL

GVALH = H*AREA

Q = GVALH*(T1-T2)

Save the file and quit your editor.

As you scrolled down through UHVAL you may have noticed that Q, GVALH, H, T1, and T2 are already declared variables; hence, you only needed to declare RL and AREA.

You will now compile your user routine. Delete any existing **ulib.a** you may have previously created, if any. Type in the command:

%ulib ulib.f

After successful compilation a new **ulib.a** will be present in your subdirectory. If any syntax errors scroll through the window during compilation then re-edit the file and repeat the above compilation step.

To call the Convection Configuration you just created in UHVAL, you will need to create an appropriate convection template in the **template.dat.apnd** file. Use the new **Analysis/Build Template** function to accomplish this.

◆ Analysis	
Action:	Build Template
Create Template File...	
Create/CONV/Data Entry	
CONV ID	1
CFIG ID	1000
Apply	
Write File...	
OK	

Cancel

Cancel

Note: Since all of the **GP** values are supplied through the MSC.Patran interface and the calculation in **UHVAL** uses no material properties, no **GP** values and **MPID**'s have to be specified in the convection template.

Prepare and run analysis

10. 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

◆ **Analysis**

Analyze/Full Model/Full Run

Translation Parameters...

2D Plane Geometry, X Y Co-ordinates (Unit Thickness in Z)

OK

Output Requests...

Nodal Results File Format...

Select Thermal Entries to Output

<select all 8 items by selecting them>

OK

OK

Apply

Read Result

11. Read results file and plot results.

After completion of the analysis read and post-process the results. The resulting figure should look like the one in the title page of this exercise.

◆ **Analysis**

Read Results/Result Entities

Select Results File...

Directories

<path>/exercise_15

Filter

Available Files

nr0.nrf.01

OK

Select Rslt Template File...

Files

pthermal_nod_T.res_tmpl

OK

Apply

Warning

OK

To plot the results to posted FEM use the **Results Application radio button**.

◆ 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

Select the *Select Result* icon.



Select Fringe Result

Apply

Average Convection Coefficient

Quit MSC.Patran

12. Quit MSC.Patran

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