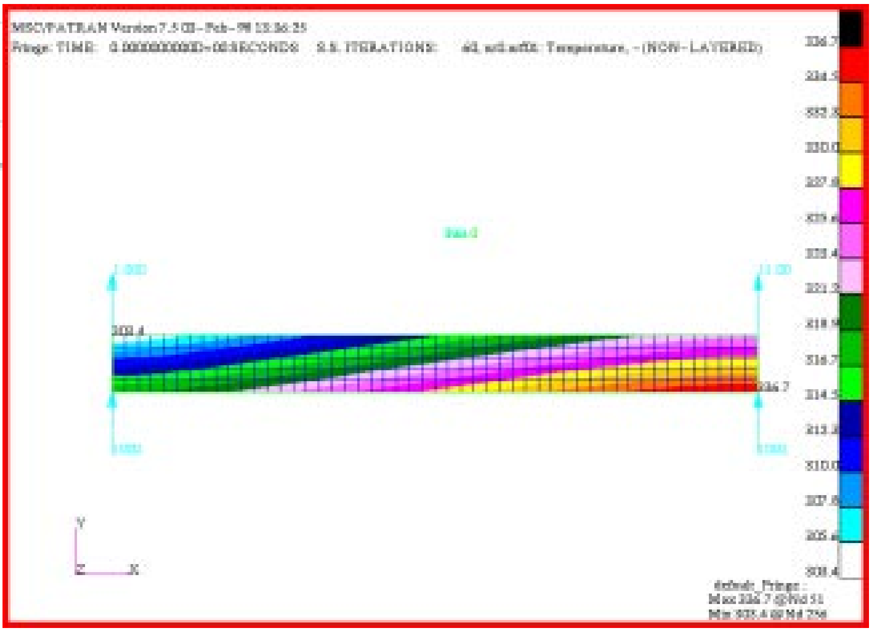
# Exercise 15

# *User Supplied Subroutines*

```
C#########################################################################
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 numbe
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
```



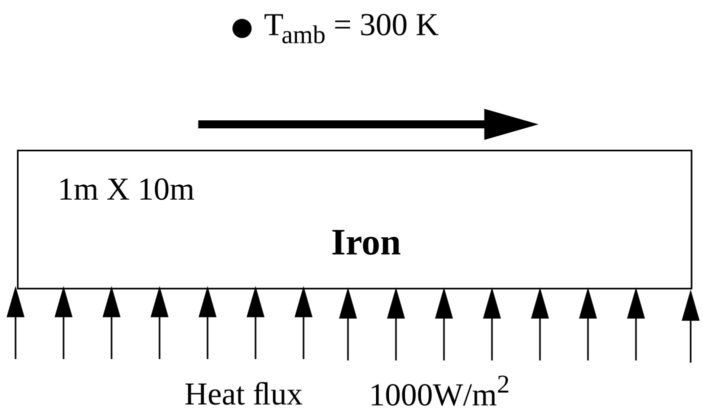
## Objective:

- 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 <u>Loads/BCs</u>.
- Copy **ulib.f**.

- Modify and compile **ulib.f**.
- Prepare and submit the model for analysis.
- Read the results file and plot temperature an 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

**File**

**New**

New Database Name **exercise_15**

**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 ◆ **Default**

Analysis Code **MSC/THERMAL**

**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**

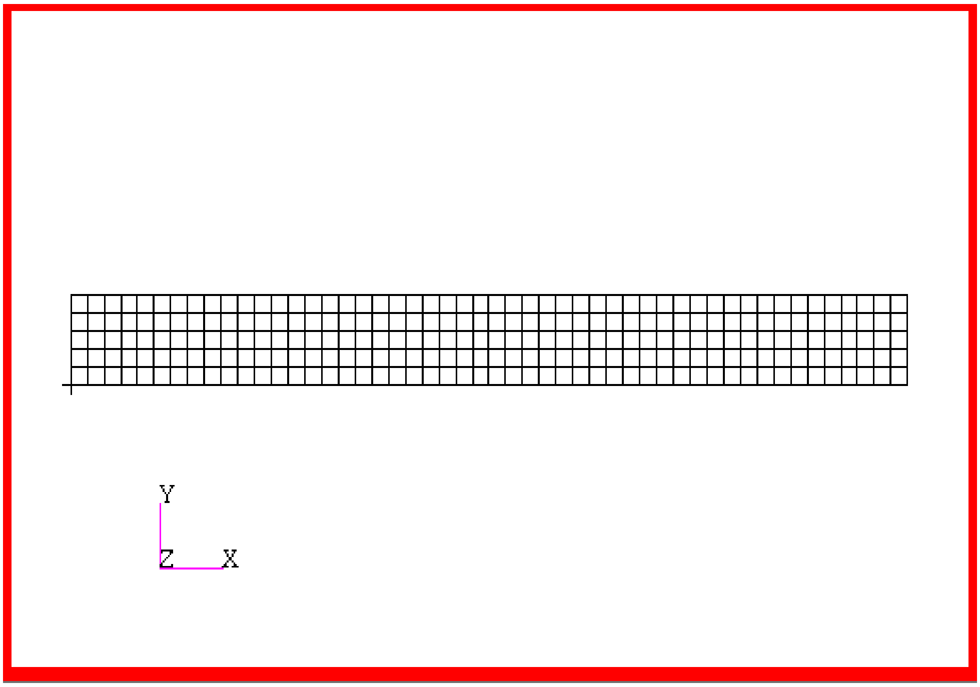
**Create/Mesh/Surface**

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**.

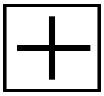
◆ **Finite Elements**

**Create/Node/Edit**

Node Id List **999**

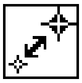
Associate with Geometry (toggle off)

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.



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

**Apply element properties**

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.

**◆ 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**

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

**Create micro-functions**

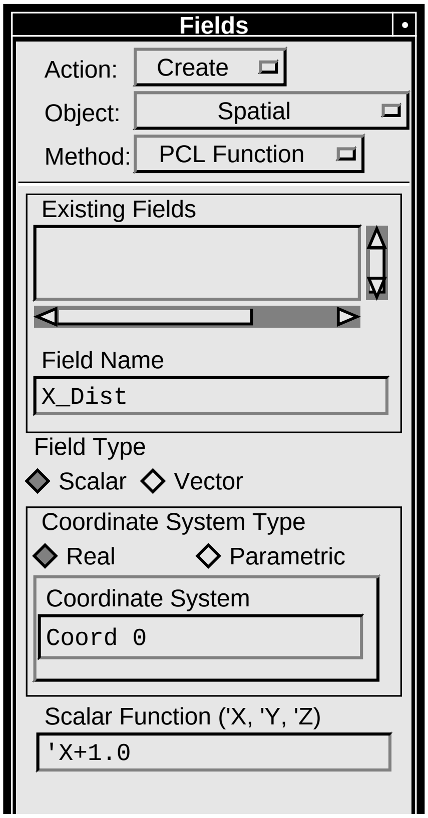
Spatial functions can be created in the Fields form using the *Action/Object/ Method* **Create/Spatial/PCL Function**.

**◆ Fields**

**Create/Spatial/ PCL Function**

| | |
|---|---|
| Field Name | **X_Dist** |
| Scalar Function ('X, 'Y, 'Z) | **'X+1.0** |

**Apply**




## 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 **Load/BCs** in the *Main Window*. Set the *Action*, *Object*, and *Type* respectively to **Create**/**Heating(P Thermal)**/**Element Uniform/Flux**.

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.

**◆ Load/BCs**

**Create/Heating/Element Uniform**

| | |
|---|---|
| Analysis Type: | **Thermal** |
| New Set Name | **Bott_Surf_Flux** |

Target Element Type: **2D**

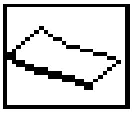
**Input Data...**

Heat Flux **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 **Load/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.

◆ **Load/BCs**

**Create/Temperature/Nodal**

Option: **Fixed**

New Set Name **Temp_amb**

**Input Data...**

Fixed Temperature **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 **Load/BCs** in the *Main Window* and set the *Action*, *Object*, and *Type* respectively to **Create/Convection(PThermal)**/**Element Uniform**.

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 and enter the field **X_Dist** for the *Convection Coefficient*, 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 S**elect 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.

◆ **Load/BCs**

**Create/Convection/Element Uniform**

| | |
|---|---|
| Option: | **Use Correlations** |
| New Set Name | **Conv_Coeff_Spatial** |
| Target Element Type: | **2D** |

**Input Data...**

| | |
|---|---|
| Convection Coefficient | **<select X-Dist> from Spatial Fields** |
| Convection Template ID | **1** |
| Fluid Node ID | **999** |

**OK**

**Select Application Region...**

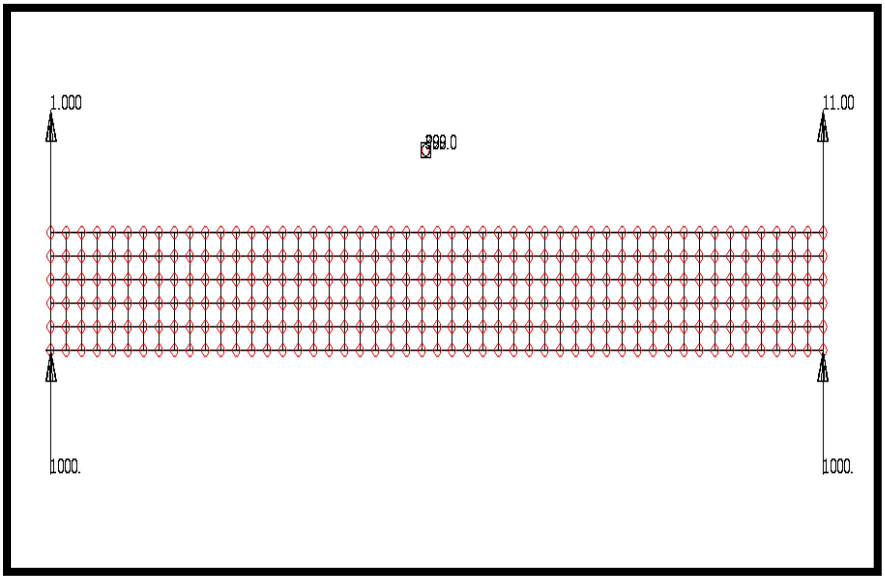
| | |
|---|---|
| **Geometry Filter** | ◆ **Geometry** |
| Select Surfaces or Edges | **<select top edge of Surface 1, Surface 1.2>** |

**Add**

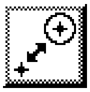
**OK**

**Apply**

The current model is shown below.




Reduce the node size with the *Node Size* icon.



8. Copy **ulib.f**.

**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**

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.

**Modify and Compile ulib.f**

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 are 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 Load/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 occurence of "UHVAL" in the file to locate the UHVAL subroutine. After the following line,

**C*C 1000 CONTINUE**

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. Move up one level of directories using **cd ..** and move any existing template.dat.apnd file out of the way and create a new one. Remember: the **template.dat.apnd** must exist at the same level as the **exercise_15.db**.

Use the same *Convection Template ID* that you specified in MSC/PATRAN when you applied the convection boundary condition. Create the Convection Template with TID=1 and Config 1000. Your convection template should look like he following:

**CONV 1 1000 0 0**

Using the system editor enter this line in the **template.dat.apnd** file.

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.

10. 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**

**Translational 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**

11. Read results file and plot results.

## Read Result

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**

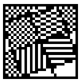
To plot the results to posted FEM use the **Results** *Application radio button*.

**◆ 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** | **Average Convection Coefficient** |

**Apply**

12. Quit MSC/PATRAN

## Quit MSC/ Patran

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