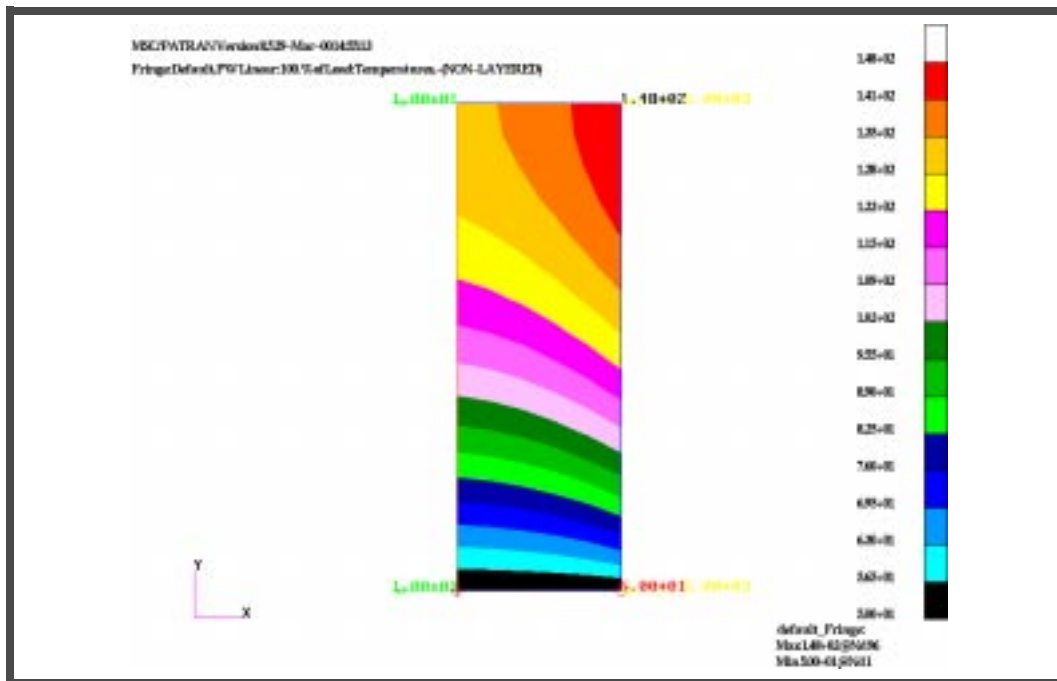


## WORKSHOP 1a

# *Getting Started Creating A Model*



### Objectives:

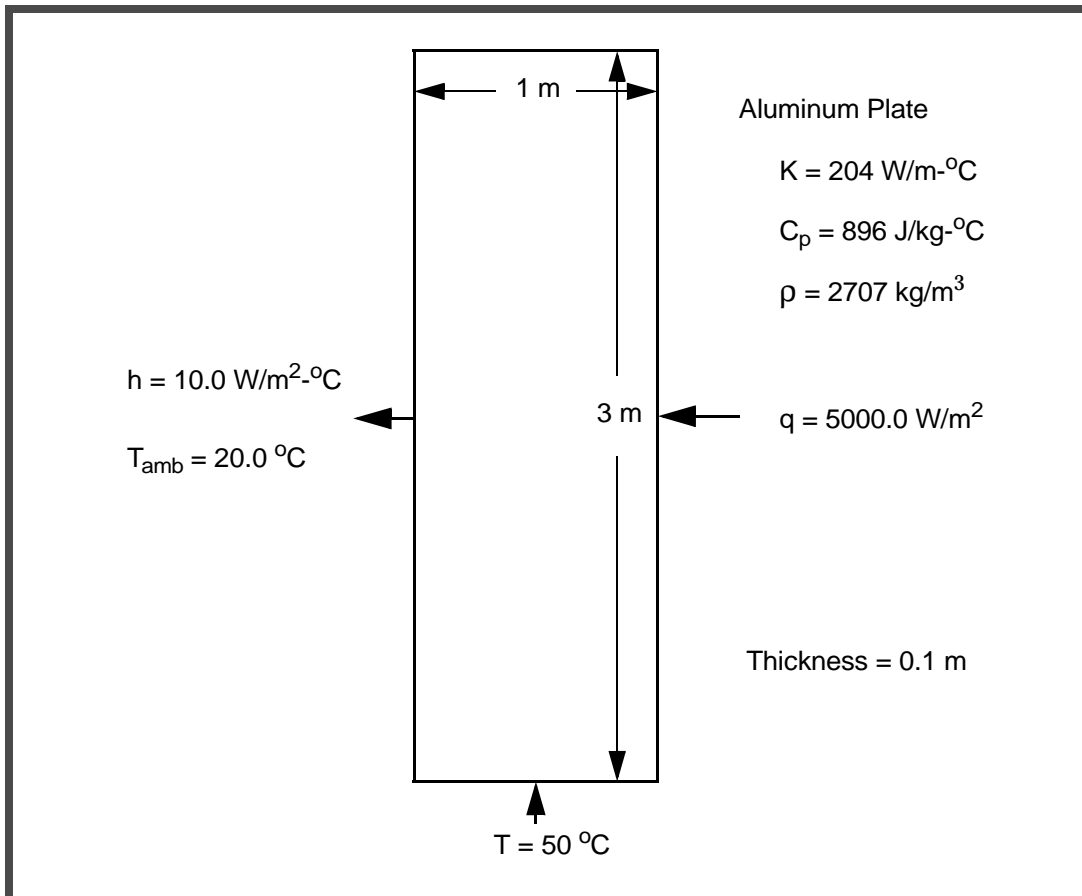
- Create a new database defined for MSC.Nastran thermal analysis.
- Define geometry for a rectangular plate.



## Model Description:

In this exercise you will first create an aluminum plate. Shown below is a drawing of the model you will be building and suggested steps for its construction.

Figure 1a.1





## Suggested Exercise Steps:

- Create a new database defined for MSC.NASTRAN thermal analysis.
- Define geometry for a rectangular plate.
- Mesh the structure with quadrilateral elements.
- Modify the mesh.
- Define the plate's material as aluminum. Specify a thermal conductivity of 204 W/m-oC, specific heat of 896 J/kg-oC, and a density of 2707 kg/m<sup>3</sup>.
- Define the plate's thickness to be 0.1 m.
- Clean up the display.
- Apply a temperature of 50 oC to the bottom edge of the plate.
- Apply heat flux of 5000 W/m<sup>2</sup> to the right edge of the plate.
- Apply to the left edge of the surface a convection boundary condition with heat transfer coefficient of 10.0 W/m<sup>2</sup>-oC and ambient temperature of 20 oC.
- Perform a steady-state thermal analysis using MSC.NASTRAN within the MSC.PATRAN system.
- Visualize the temperature distribution as a contour plot.



## Exercise Procedure:

1. Open a new database. Name it **ex1a**.

**File/New...**

*New Database Name:*

**ex1a**

**OK**

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC.PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC.Nastran**

*Tolerance:*

◆ **Based on Model**

*Analysis Code:*

**MSC/NASTRAN**

*Approximate Maximum Model Dimension:*

**10.0**

*Analysis Type:*

**Thermal**

**OK**

2. Create the Model.

◆ **Geometry**

*Action:*

**Create**

*Object:*

**Surface**

*Method:*

**XYZ**

*Vector Coordinates List:*

**<1 3 0>**

*Origin Coordinates List:*

**[0 0 0]**

**Apply**

3. Mesh the surface with elements.

◆ **Finite Elements**

*Action:*

**Create**

*Object:*

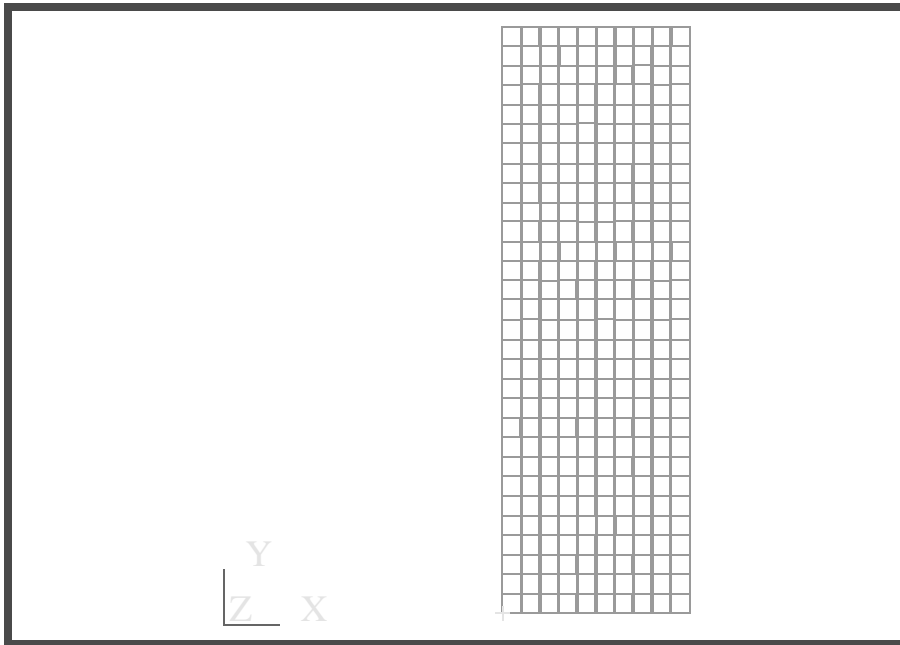
**Mesh**

---

Type:	Surface
Global Edge Length:	0.1
Mesher:	IsoMesh
Surface List:	Surface 1
<b>Apply</b>	

At this point, we will invoke MSC.PATRAN's undo feature so that we can make a coarser mesh. The mesh we have just created (300 elements) is excessive for our example.

Your model should look like the following figure.



At this point, we will invoke MSC.PATRAN's undo feature so we can make a coarser mesh. The mesh we have just created (300 elements) is excessive for our example.

Click on the **Undo** icon.



Click on the **Refresh Graphics** icon.



Note that the **Finite Elements** form is still visible. Change the Global Edge Length from 0.1 to 0.2. This will create elements of 0.2 units (meters) in length, which will result in a coarser mesh of 75 quadrilateral elements.

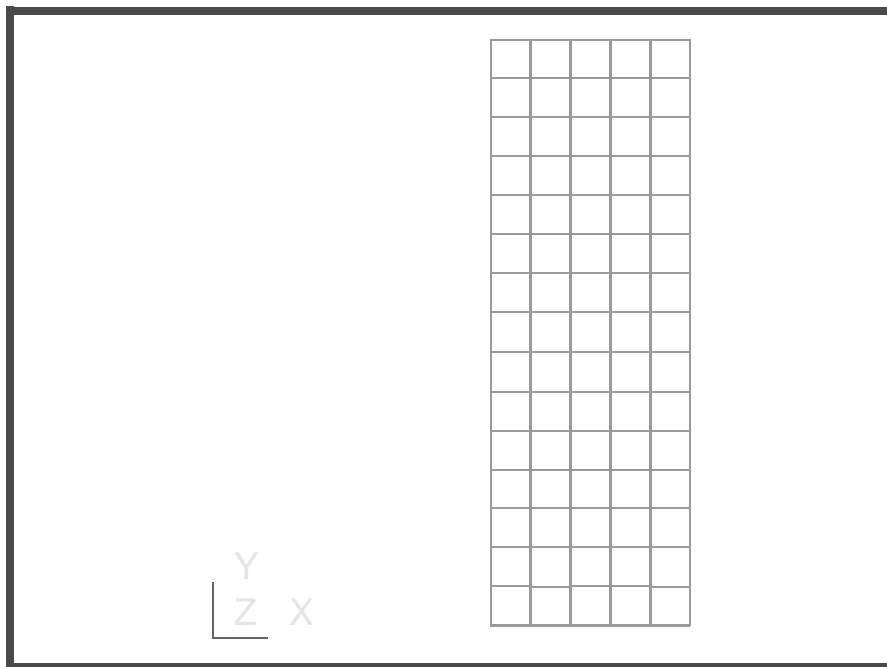
◆ **Finite Elements**

Global Edge Length:

0.2

Apply

Your model should look like the following figure.



4. Specify Material Properties.

Our material for this exercise will be aluminum. Click on the Materials application. The **Material** form will appear with certain default options.

◆ **Materials**

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name:

alum

---

**Input Properties...**

*Thermal Conductivity:*

**204**

*Specific Heat:*

**896**

*Density:*

**2707**

**Apply**

**Cancel**

5. Our next task is to specify a thickness of 0.1 to our aluminum elements.

◆ **Properties**

*Action:*

**Create**

*Object:*

**2D**

*Type:*

**Shell**

*Property Set Name:*

**plate**

**Input Properties...**

*Material Name:*

**m:alum**

*Thickness:*

**0.1**

**OK**

From the **Element Properties** form, click on the Select Members databox. MSC.PATRAN will display two icons to the left of the **Element Properties** form. The first icon represents **surface or face**, the second represents **2D element**. The two options allow you to apply properties either on the geometric entity (in this case, the surface) or on the finite elements.

Click on the **Surface or Face** icon.



**Surface or Face**

Now click anywhere on the geometric surface. The surface will be highlighted in red. The Select Members databox will now appear as *Surface 1*.

**Add**

**Apply**

6. Apply the load and boundary conditions.

◆ **Load/BCs**

Action:	<b>Create</b>
Object:	<b>Temp(Thermal)</b>
Type:	<b>Nodal</b>
New Set Name:	<b>tempbc</b>

**Input Data...**

Boundary Temperature:	<b>50</b>
-----------------------	-----------

**OK**

**Select Application Region...**

Geometry Filter:	◆ <b>Geometry</b>
------------------	-------------------

Click on the **Curve or Edge** icon.



With your mouse, position the cursor on the bottom edge of the surface. Click on the edge. You will see *Surface 1.4* appear in the **Select Geometry Entities** databox. This means we have selected Edge number 4 in *Surface number 1*.

**Add**

**OK**

**Apply**

7. We will now apply heat flux to the model using the Loads/Boundary Conditions form.

◆ **Load/BCs**

Action:	<b>Create</b>
Object:	<b>Applied Heat</b>
Type:	<b>Element Uniform</b>

<i>Option:</i>	<b>Normal Fluxes</b>
<i>Analysis Type:</i>	<b>Thermal</b>
<i>New Set Name:</i>	<b>flux</b>
<i>Target Element Type:</i>	<b>2D</b>

Because the problem is a 2D one, we need to toggle that **Target Element Type** setting to 2D. Even though we are applying heat flux along an edge, which we normally think of as 1D, our finite element problem is 2D; i.e., we are modeling heat conduction in two dimensions.

<b>Input Data...</b>	
<i>Form Type:</i>	<b>Basic</b>
<i>Surface Option:</i>	<b>Edge</b>
<i>Edge Heat Flux:</i>	<b>5000</b>

**OK**

**Select Application Region...**

*Geometry Filter:*      **◆ Geometry**

Click on the **Edge** icon.

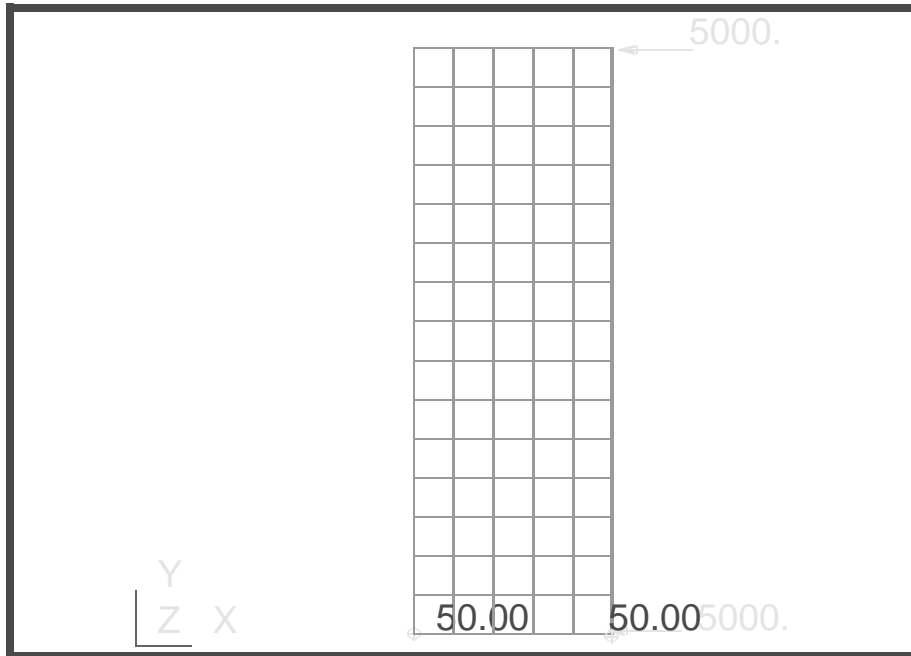


Position the cursor over the right edge of the surface and click on this edge with the mouse. MSC.PATRAN will insert Surface 1.3 in the databox under the heading Select Surfaces or Edges.

<b>Add</b>
<b>OK</b>
<b>Apply</b>

A yellow flag will appear on the right edge of your surface indicating that a heat flux of 5000 W/m<sup>2</sup> has been applied along the right edge.

Your model should look like the following figure.



8. We will now apply a convection boundary condition to the left edge of the plate-- again, using the **Loads/BCs** form.

◆ **Load/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Convention"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="To Ambient"/>
<i>Analysis Type:</i>	<input type="text" value="Thermal"/>
<i>New Set Name:</i>	<input type="text" value="conv"/>
<i>Target Element Type:</i>	<input type="text" value="2D"/>
<input type="text" value="Input Data..."/>	
<i>Surface Option:</i>	<input type="text" value="Edge"/>
<i>Edge Convection Coef:</i>	<input type="text" value="10"/>
<i>Ambient Temperature:</i>	<input type="text" value="20"/>
<input type="text" value="OK"/>	

---

**Select Application Region...**

*Geometry Filter:*

◆ **Geometry**

Click on the **Edge** icon.



Position the cursor over the left edge of the surface and click on the edge with the mouse. MSC.PATRAN will insert *Surface 1.1* in the databox under **Select Surfaces** or **Edges**

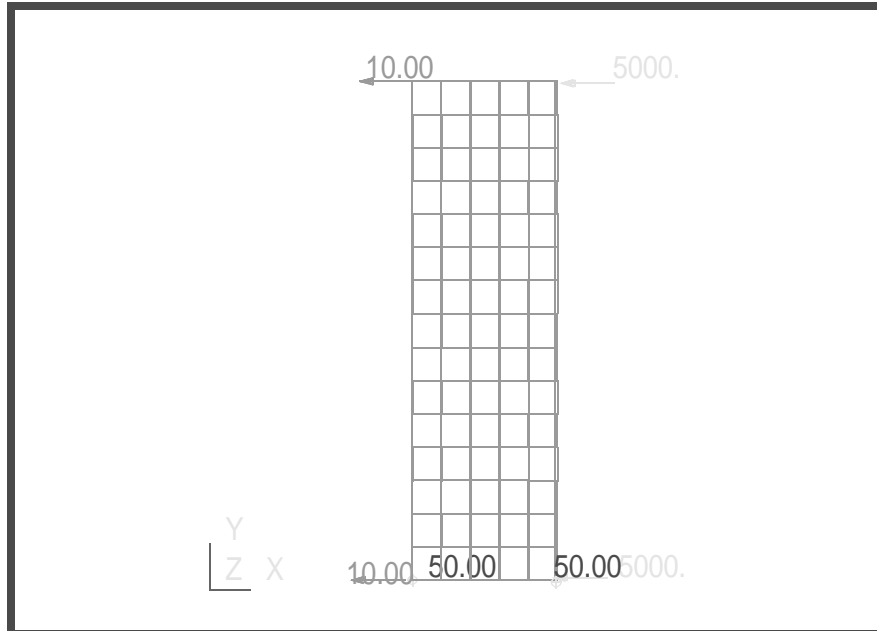
**Add**

**OK**

**Apply**

A green label will appear confirming that you have applied a convection coefficient of  $10.0\text{W/m}^2\text{-oC}$  at this location of your model.

Your model should look like the following figure.



9. We are now ready to submit the model for MSC.NASTRAN steady-state thermal analysis. Click on the **Analysis** application located on the MSC.PATRAN main form.

◆ **Analysis**

*Action:*

**Analyze**

*Object:*

**Entire Model**

*Method:*

**Analysis Deck**

*Job Name:*

**ex1a**

**Apply**

---

## Submitting the Input File for Analysis:

10. Submit the input file to MSC.NASTRAN for analysis.

To submit the MSC.PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran ex1a.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

11. When the run is completed, edit the **ex1a.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing **WARNING** messages indicate modeling errors.

- 12. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**
- 13. Proceed with the Reverse Translation process, that is, attaching the ex1a.xdb results file into MSC.Patran. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	<input type="text" value="Attach XDB"/>
<i>Object:</i>	<input type="text" value="Result Entities"/>
<i>Method:</i>	<input type="text" value="Local"/>
<input type="text" value="Select Results File..."/>	
<i>Select Results File</i>	<input type="text" value="ex1a.xdb"/>
<input type="text" value="OK"/>	
<input type="text" value="Apply"/>	

- 14. Display the Results.

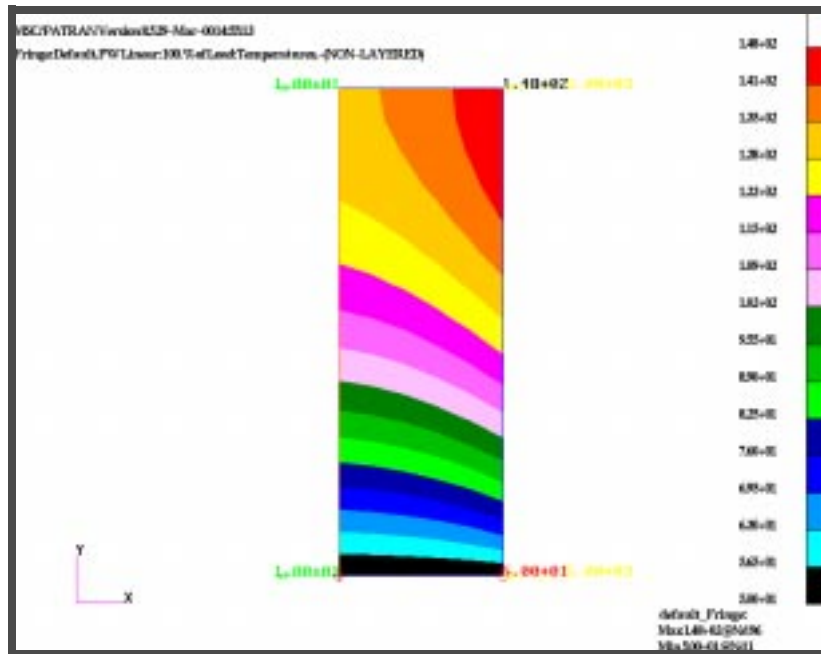
◆ **Results**

<i>Select Results Cases:</i>	<input type="text" value="Default, PW Linear: 100. % of Load"/>
<i>Select Fringe Result:</i>	<input type="text" value="Temperatures"/>
<input type="text" value="Apply"/>	

A contour plot displaying temperature distributions will appear.

---

Your model should look like the following figure.



Select the Save and Close operations from the File menu to save your plate.db file. We will perform a transient thermal analysis on this model in the next workshop.

15. Close database and quit MSC.Patran to complete this exercise.

**File/Quit...**