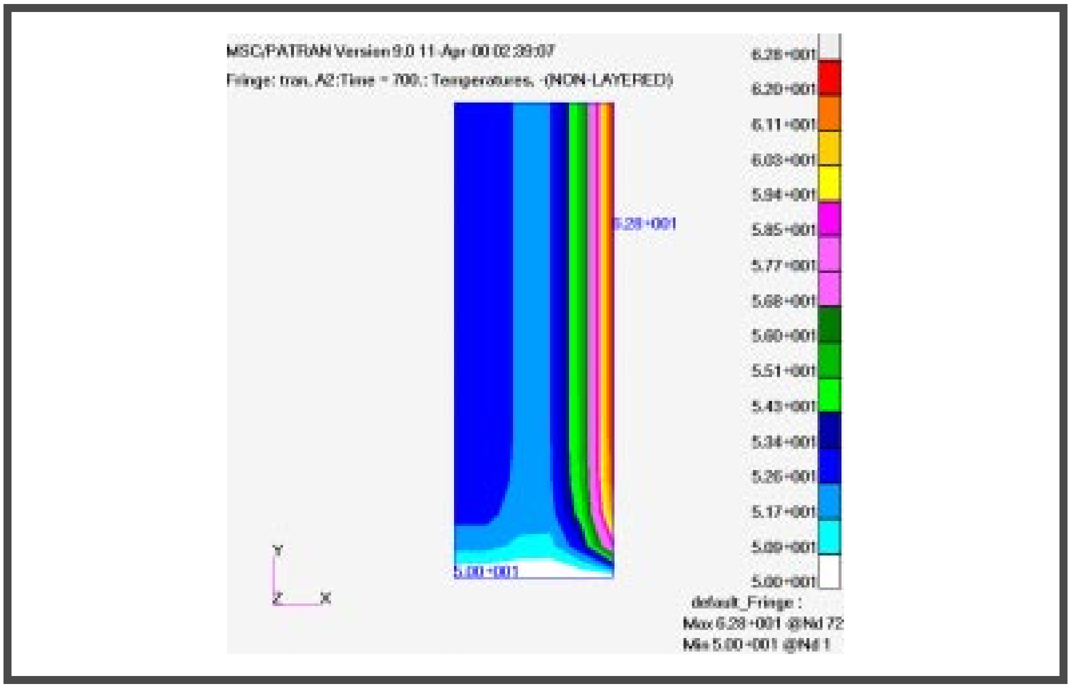
WORKSHOP 1b

Transient Thermal



Objectives:

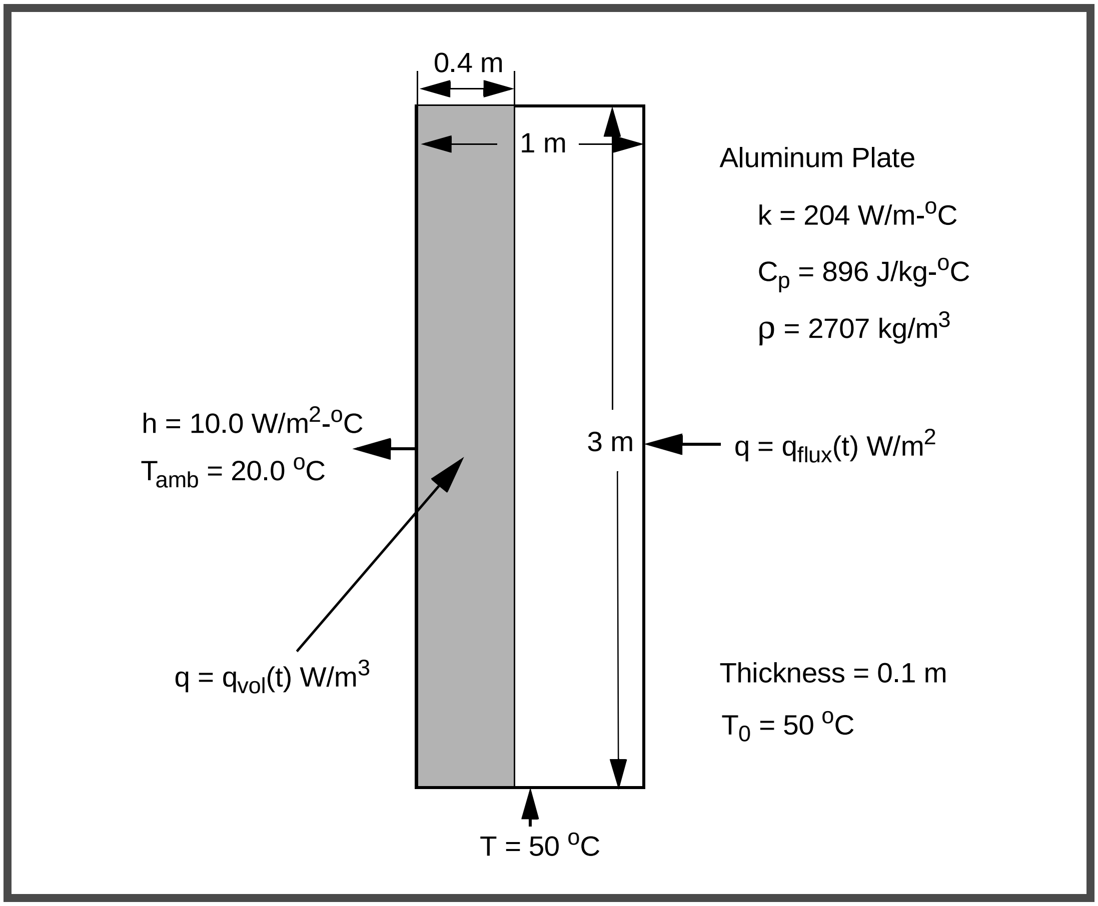
- Open the database created in Workshop 1a.
- Define time dependent funtions using the Field application.
- Create a trasient load case.

Model Description:

This exercise describes transient thermal analysis, it is an extension of the steady state modeling exercise given in Workshop 1a. This workshop contains step-by-step descriptions of the menu picks involved in the modeling process.

Shown below is a drawing of the model you will be building and suggested steps for its construction

Figure 1b.1



Suggested Exercise Steps:

- Open the database created in Workshop 1a.
- Define time dependent functions using the Field application.
- Create a transient load case. Add two existing load sets (temperature and convection boundary conditions) to this transient load case.
- Apply time varying heat flux to the right edge of the plate
- Apply a transient volumetric heat generation inside the shaded area of the plate
- Select solution type as transient analysis.
- Specify the default initial temperature.
- Define time steps.
- Select a transient load case.
- Perform a transient thermal analysis using MSC.NASTRAN within the MSC.PATRAN system
- Postprocess the transient results (Contour and XY plots).

Exercise Procedure:

1. Open the database created in workshop 1a.

File/Open...

Existing Database Name: **ex1a**

OK

2. Define Time Dependent Functions.

Before applying time varying loads and boundary conditions, we need to define time dependent functions using the Field application. In this model, two time fields are defined, one for applied heat flux and one for volumetric heat generation.

◆ **Fields**

Action: **Create**

Object: **Non Spatial**

Method: **Tabular Input**

Field Name: **flux_time**

Input Data...

Fill in the table with the following values using the RETURN or ENTER key.

	Time(t):	*Value:*
1 :	**0**	**1**
2 :	**10**	**1.25**
3 :	**30**	**1.75**
4 :	**50**	**2**
5 :	**100**	**2**

OK

Apply

Similarly, a time dependent function for volumetric heating is defined as follows.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name:	**qvol_time**

Input Data...

	Time(t):	*Value:*
1:	**0**	**10000**
2:	**10**	**12000**
3:	**30**	**13000**
4:	**50**	**14000**
5:	**100**	**14000**

OK

Apply

3. Create a transient load case.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**transient**
Load Case Type:	**Time Dependent**

Since the temperature and convection boundary conditions are not changed from Workshop 1a, we can associate these two load sets with the new load case directly.

Assign/Prioritize Loads/BCs

Highlight **Conve_conv** and **Temp_tempbc** within the *Select Individual Loads/BCs* Sets listbox.

OK

Apply

At this point, we will impose a transient flux load on the plate's right edge. The magnitude of this flux load is 5000 W/m2 multiplied by the time dependent function **flux_time** defined earlier under the Fields application. Click on the Loads/BCs application.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Applied Heat**
Method:	**Element Uniform**
Option:	**Normal Fluxes**
Analysis Type:	**Thermal**
New Set Name:	**tran_flux**
Target Element Type:	**2D**

Input Data...

Surface Option:	**Edge**
Edge Heat Flux:	**5000**
Time Function:	**f:flux_time**

OK

Select Application Region...

Select 2D Elements:	**Surface 1.3**

Add

OK

Apply

4. Apply Transient Volumetric Heat Generation Inside the Plate.

The volumetric heating can be applied in a similar way, using the Loads and Boundary Conditions form as follows.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Applied Heat**
Method:	**Element Uniform**
Option:	**Volumetric Generation**
Analysis Type:	**Thermal**
New Set Name:	**tran_qvol**
Target Element Type:	**2D**

Input Data...

Time Function:	**f:qvol_time**

OK

Next, click on Select Application Region located on the Loads and Boundary Conditions form. We want to apply an internal heat generation inside a section of the plate from x=0.0 m to x=0.4 m. This application region will be selected by graphical cursor using the FEM geometry filter.

Select Application Region...

Geometry Filter: ◆ **FEM**

Use the mouse cursor to drag a rectangle covering the elements located between x=0.0 m and x=0.4 m. Release the mouse cursor. The first two columns of the elements will turn red indicating the selection. Also, a list of elements will appear in the Select 2D Elements databox.

Add

OK

Apply

Note: A square yellow marker will appear on the center of the selected element indicating that a volumetric heating has been applied on this element.

5. Now we are ready to set the analysis controls for transient thermal analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**ex1b**

Solution Type...

◆ **TRANSIENT ANALYSIS**

Solution Parameters...

For transient thermal analysis, we have to employ a starting temperature from which the solution evolves. If the initial temperature distribution is uniform, a default initial temperature is sufficient to specify the initial state. Otherwise, the Initial Temperature object in Loads and BCs application must be used to define initial nodal temperatures explicitly.

Default Init Temperature:	**50.0**

OK

OK

Subcase Create...

Available Subcase:	**transient**

Subcase Parameters...

Initial Time Step:	**10**
Number of Time Steps:	**100**

OK

Apply

Cancel

Subcase Select...

Subcases for Solution Sequence: **transient**

Subcases Selected: **Default**

OK

Apply

Submitting the Input File for Analysis:

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

    6a. To submit the MSC.PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran ex1b.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

7. 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.

8. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**

9. Proceed with the Reverse Translation process, that is, attaching the **ex1b.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**
Select Results File...	
Select Results File	**ex1b.xdb**
OK	
Apply	

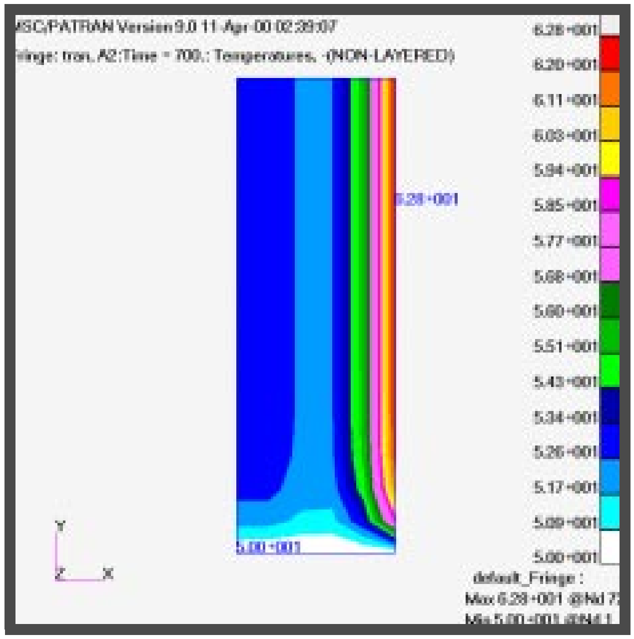
Note: The heartbeat will change to the color blue, indicating that reading process is underway. When the heartbeat turns green again, the results are ready for postprocess.

10. We will create a contour plot of temperature distributions at t*ime=700* sec using the **Results Display** form.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**
Select Results Cases:	**Transient, Time=700**
Select Fringe Result:	**Temperatures**
Apply	

Your model should look like the following figure.



Now we will apply XY plotting to visualize the temperature-time history of Nodes 49-54.

◆ **Results**

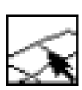
Action:	**Create**
Object:	**Graph**
Method:	**Y vs. X**

In the **Select Result Case(s)** listbox, click an drag mouse to select the time states from *transient, Time=0,* to *transient, time=1020.*

Within the **Select Y Result listbox,** highlight *Temperatures*.

Select Y Result:	**Temperatures**

Click on the **Target Entities** icon.

 **Target Entities**

Target Entity:	**Nodes**

Select Nodes: **Node 49:54**

Apply

At this point, we will modeify the Y scale of the XY plot and display grid lines in the Y directly by clicking on the **XY Plot** application.

◆ **XY Plot**

Action: **Modify**

Object: **Axis**

Select Current XY Window: **Results Graph**

Active Axis: ◆ **Y**

Scale...

Scale: ◆ **Linear**

Assignment Method: ◆ **Range**

Enter Lower and Upper Values: **45 70**

Number of Primary Tick Marks: **6**

Apply

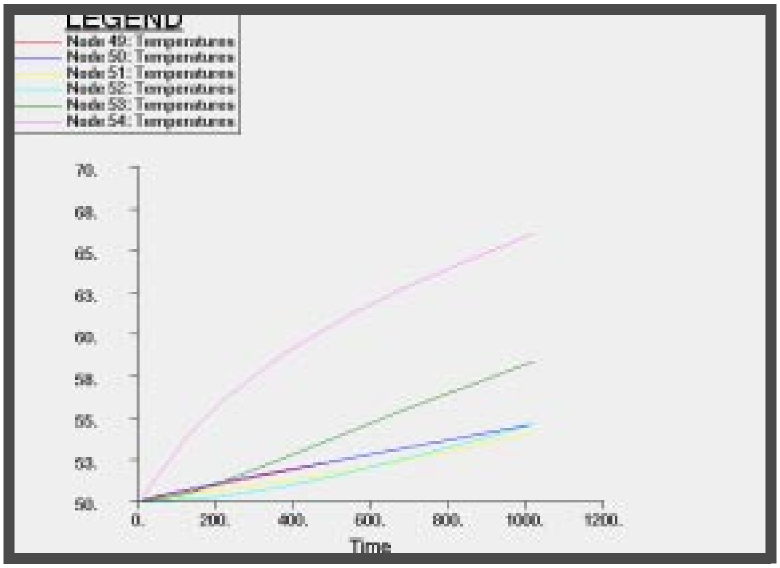
Cancel

Grid Lines...

Display: ■ **Primary**

Apply

Your model should look like the following figure.



11. Close the database and quit MSC.Patran when you have completed this exercise.

    File/Quit...