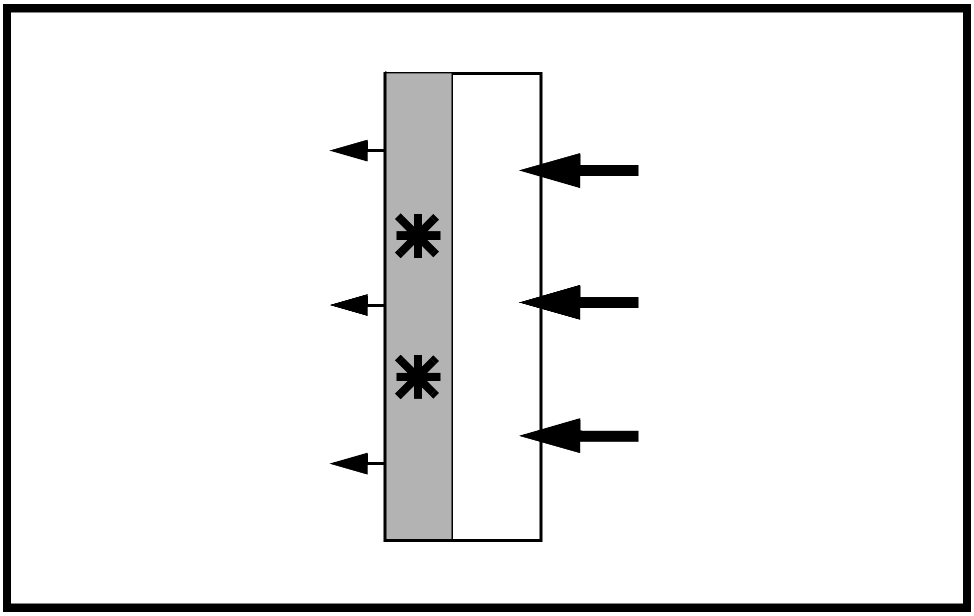
LESSON 1

Transient Thermal Analysis of a Heating Element



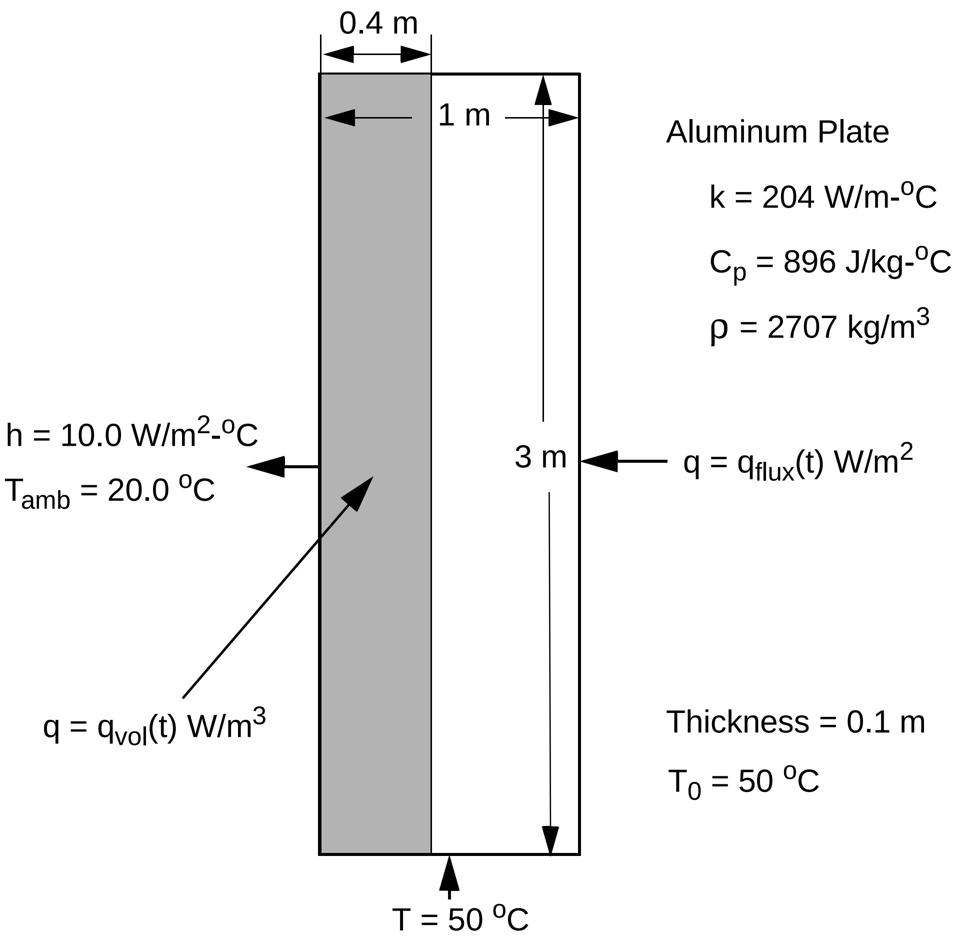
Objectives:

- Create a solid model of the heating element.
- Apply thermal load of convection, heat generation, heat flux, and fixed temperature to the model.
- Run a transient heat transfer analysis of the plate.

Model Description:

A rectangular heating element is subjuected to a heat flux along one of its long edges. The opposite edge is cooled by convection, and a temperature controlled fitting holds the bottom of the element at 50 degrees Celsius. It can be assumed that, despite the heat generation processes/materials within, the heating element's thermal properties are identical to those of aluminum. The aim of this exercise is to determine the transient thermal response of the heating element for a period of 1000 seconds from a 'cold start' (i.e. the unloaded condition)

Below is shown an aluminum plate which is subjected to several types of thermal loading. You will create this model, and analyze it to determine the transient behavior of the temperature for a period of 2 seconds.



Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0.2 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.2.

On the *Open Model File* form, select **New Model.**

Open Model File: **New Model**

2. Create the NASTRAN geometry for the plate.

Geometry/Surface/Plane...

Fill in the table as the following:

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Point 1:	**1**	**0**	**0**
Point 2:	**0**	**1**	**0**

OK

3. The following table will apear on your screen. Fill in the appropriate dimensions.

Width (along Plane X): **1**

Height (along Plane Y): **3**

OK

Cancel

4. To fit the display onto the screen, use the Autoscale feature.

View/Autoscale... **< CTRL A >**

5. Set the default size for the mesh.

Mesh/Mesh Control/Default Size...

Size:	**0.1**

OK

6. Create a material called **alum**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title:	**alum**
Mass Density:	**2707**
Conductivity, k:	**204**
Specific Heat, Cp:	**896**

OK

Cancel

7. Create a property called **plate** to apply to the members of the plate itself.

From the pulldown menu, select **Create/Property**.

Model/Property...

Title:	**plate**

To select the material, click on the list icon next to the databox and select **alum**.

Material:	**1..alum**
Thickness, Tavg or T1:	**0.1**

OK

8. Create a property called **solid**, to be applied to the model later.

Title: **solid**

Material: **1..alum**

Elem/Property Type...

Volume Elements: l **Solid**

OK

OK

Cancel

9. Create the mesh for the model.

Mesh/Geometry/Surface...

Select All

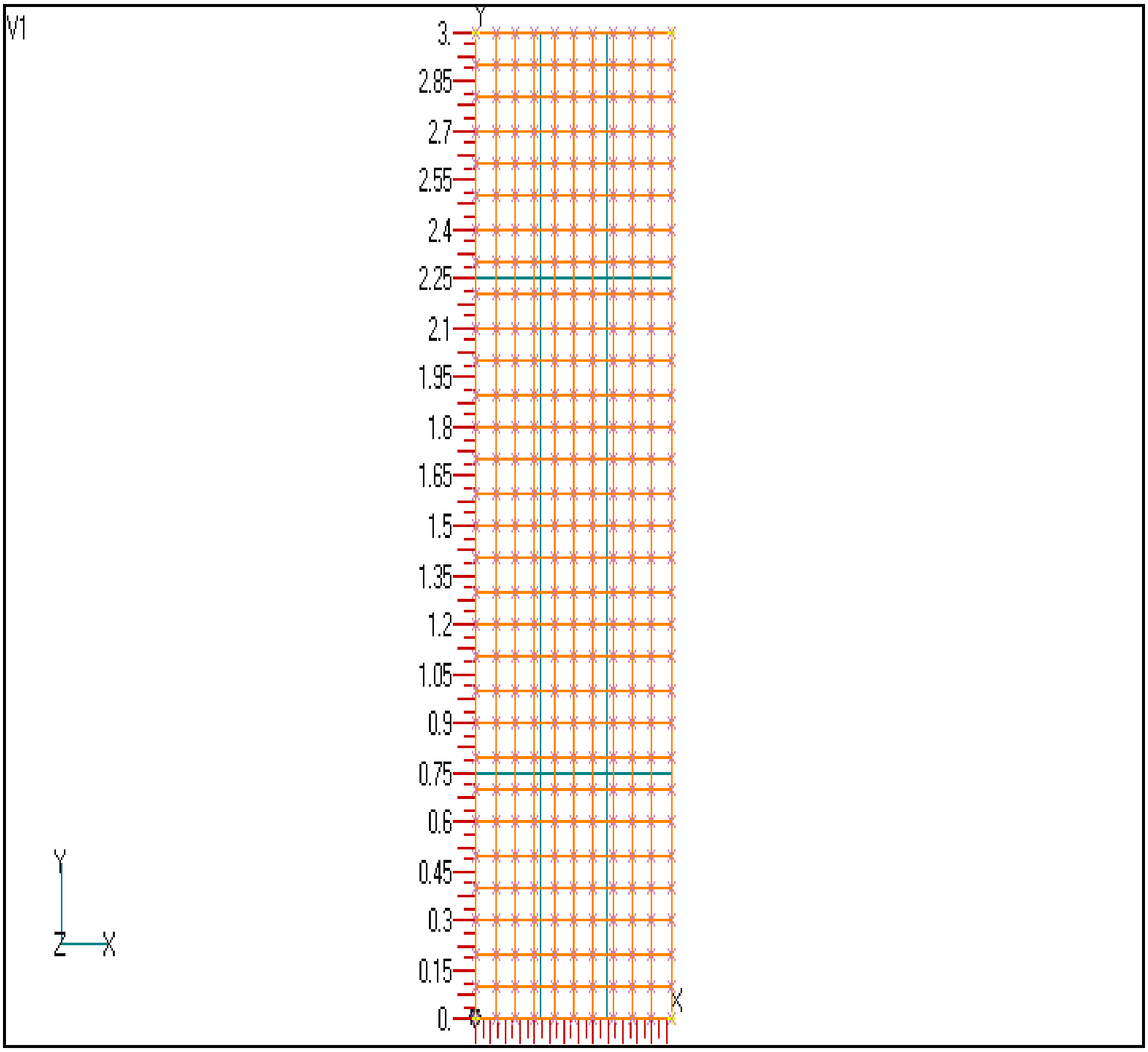
OK

Property: **1..plate**

OK

Your model should appear to be like the following:

Figure 1: Plate model with meshed surfaces



10. Remove the labels from the screen.

View/Options... < CTRL Q >

Quick Options...

Labels Off

Done

OK

11. Extrude the 2D shell elements into 3D solid elements.

Mesh/Extrude/Element...

Select All

OK

Property: **2..solid**

Elements Along Length: **1**

☒ **Delete Original Elements**

OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**0.1**

OK

When asked “OK to Delete 300 Select Element(s)?”, respond **Yes**.

Yes

12. Create a uniform temperature loading for the model.

First, a load set must first be created before creating the appropriate model loading.

Model/Load/Set...

Title: **transient**

OK

Next, apply a uniform default temperature to the model.

Model/Load/Body...

(next to Thermal options) ☒ **Active**

Default Temperature: **50**

OK

13. Create time-dependent functions for the heat flux and volumetric heating.

Model/Create/Function...

Title:		**flux_time**
Type:		**1.. vs Time**
X:	*Y:*	
0	**1**	**More**
10	**1.25**	**More**
30	**1.75**	**More**
50	**2**	**More**
100	**2**	**More**

OK

Title:		**qvol_time**
Type:		**1.. vs Time**
X:	*Y:*	
0	**10000**	**More**
10	**12000**	**More**
30	**13000**	**More**
50	**14000**	**More**
100	**14000**	**More**

OK

Cancel

14. Apply a fixed temperature of 50 degrees to the bottom edge of the model.

Model/Load/Nodal...

Hold down the shift key and drag a box around the bottom edge nodes. (you might need to move the entity select menu)

OK

Type: l **Temperature**

Temperature: **50**

OK

Cancel

15. Create the heat flux for the model.

Model/Load/Elemental...

Hold shift and drag a box around the right edge of the model.

OK

Type: l **Heat Flux**

Value: **5000**

Function Dependence: **1..flux_time**

OK

Face: *(click on right edge of top right element)*

OK

16. Create the free convection for the model.

Hold shift and drag a box around the left edge of the model.

OK

Type: l **Convection**

Coefficient: **10**

Temperature: **20**

OK

Face: *(click on left edge of top left element)*

OK

17. Create the volumetric heat generation for the model.

Hold shift and drag a box to select the four left columns of elements in the model.

OK

Type: l **Heat Generation**

Value: **1**

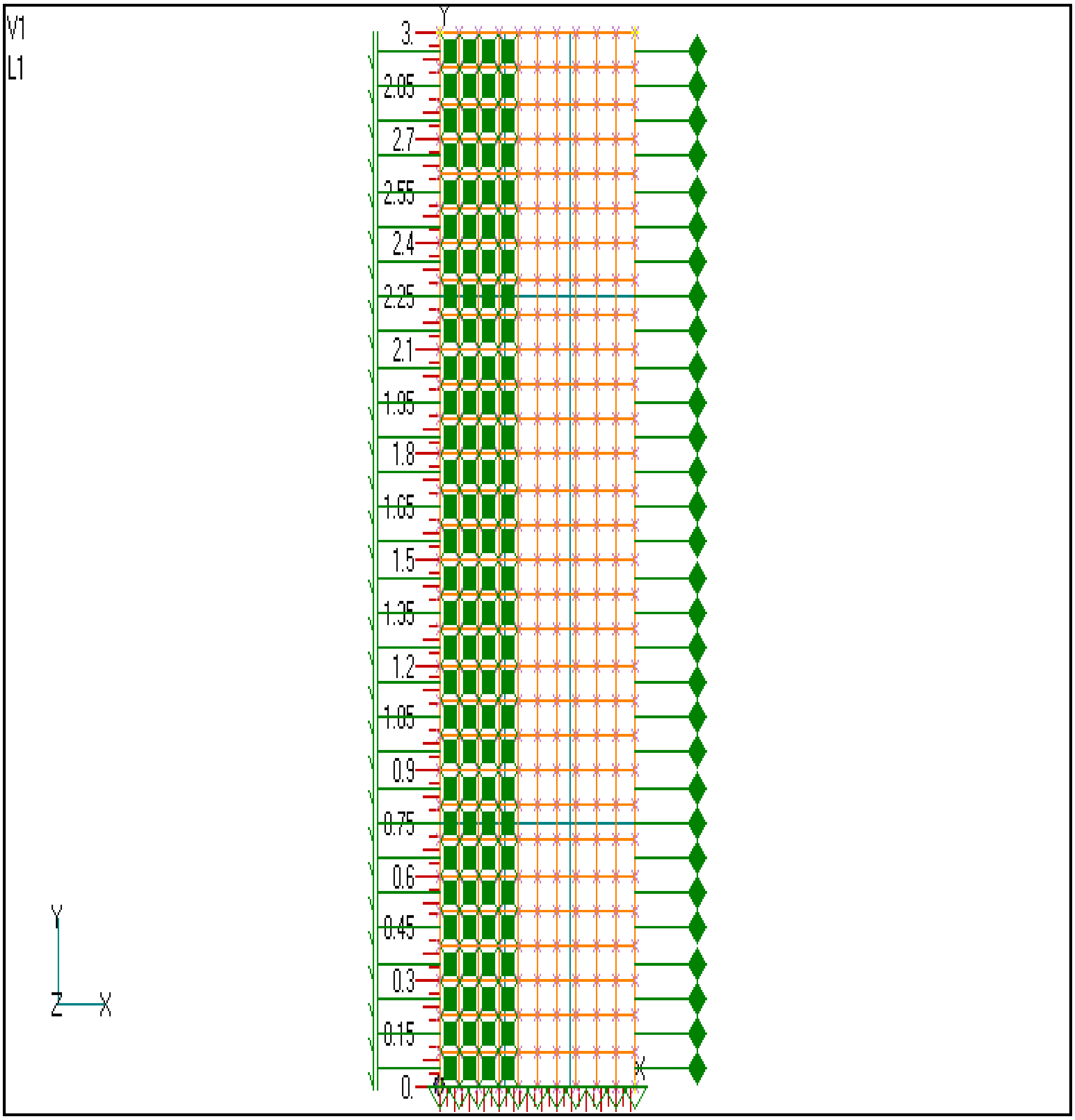
Function Dependence: **1..qvol_time**

OK

Cancel

Your model should look like the following:

Figure 2: Model with loads



18. Create the input file and run the analysis..

File/Analyze

Analysis Type:	**21..Transient Heat Transfer**
Number of Time Steps:	**100**
Initial Time Increment:	**10**

☒ **Run Analysis**

OK

19. When asked if you wish to save the model, respond **Yes**.

Yes

File Name: trans

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

.

20. Remove the thermal loading markers from the screen.

View/Options... < CTRL Q >

Quick Options...

☐ **Load - Thermal**

☐ **Load - Heat Generation**

☐ **Load - Heat Flux**

☐ **Load - Convection**

Done

OK

21. Create a final temperature distribution contour plot.

View/Select...

Model Style: l **Quick Hidden Line**

Contour Style: l **Contour**

Deformed and Contour Data...

Output Set: **5..Case 9 Time 190**

Contour: **31..Temperature**

OK

OK

22. Redfine the spectrm and ranges used to plot the temperature contours.

View/Options... **(or use <F6>)**

PostProcessing: l **PostProcessing**

Options: **Contour/Criteria Levels**

Level Mode: **2..Max Min**

Minimum: **50.0**

Maximum: **57.0**

Apply

OK

Notice the effects of the free convection, heat flux, fixed temperature, and heat generation on the temperature distribution. Compare your results against those depicted in Figure 3.

23. Create an XY plot of the temperature along the left edge as a function of Y.

First, you will need to create a group for the elements in the XY plot.

Group/Set...

Title: **temp along the left edge**

OK

Group/Node/Id...

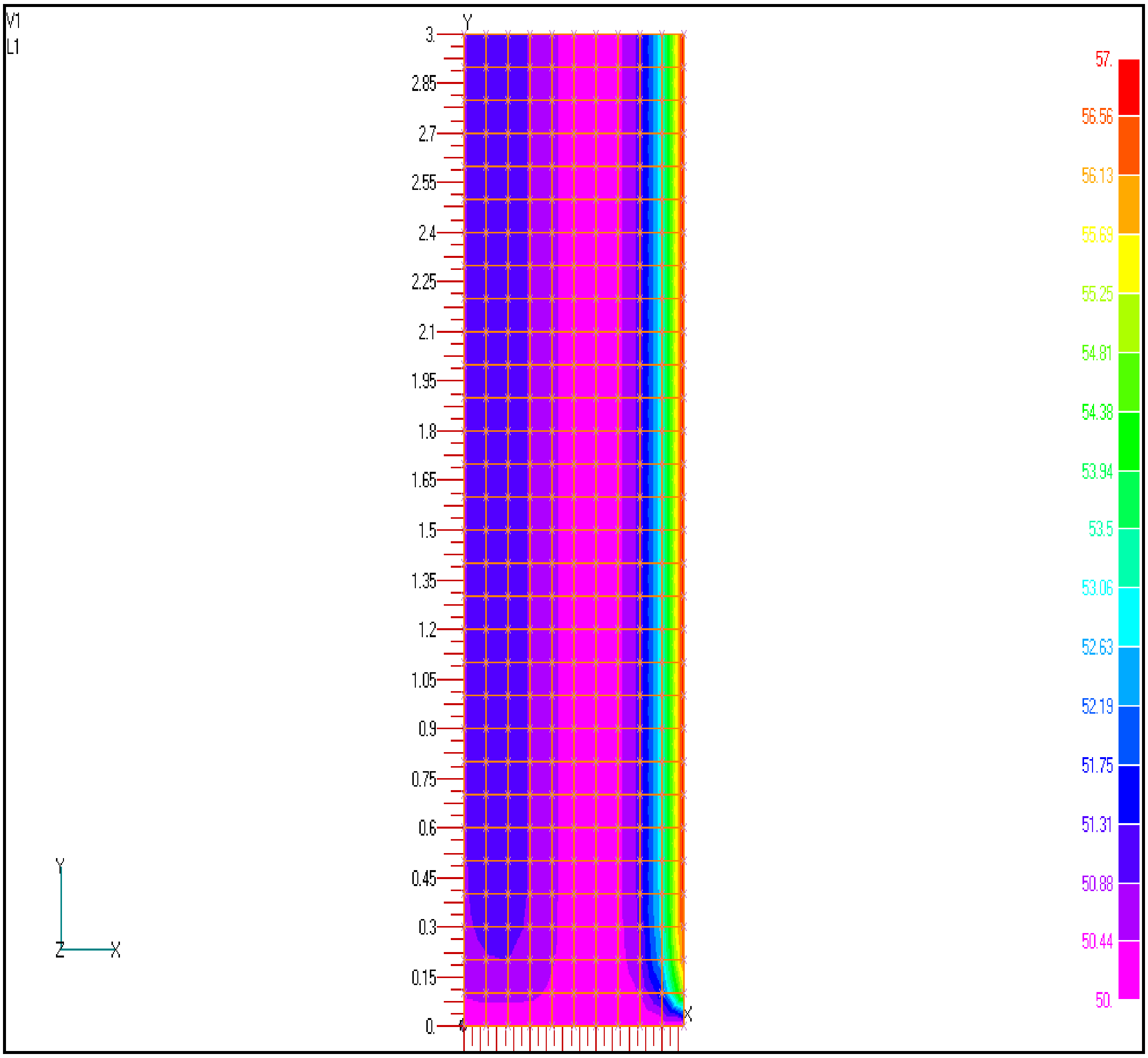
ID: **342**

to: **672**

by: **21**

More
OK

Figure 3: Transient Thermal Analysis



24. Now, create the XY plot.

View/Select...	**(or use <F5>)**
XY Style:	l **XY vs Position**
XY Data...	
Position:	l **Y**
Group:	l **Select**
	1..temp along the left edge
Output Set:	**5..Case 9 Time 190**
Output Vector:	**31..Temperature**
OK	
OK	

Compare your results with Figure 4.

When done, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Figure 4: Temperature along the left edge vs Vertical position