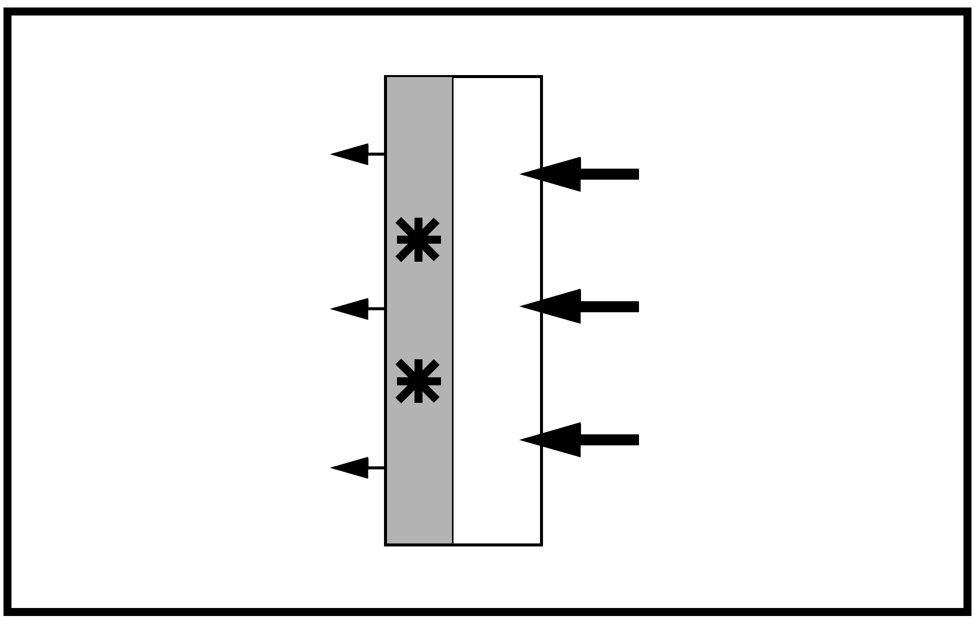
# WORKSHOP 4

# *Transient Thermal Analysis of a Heating Element*



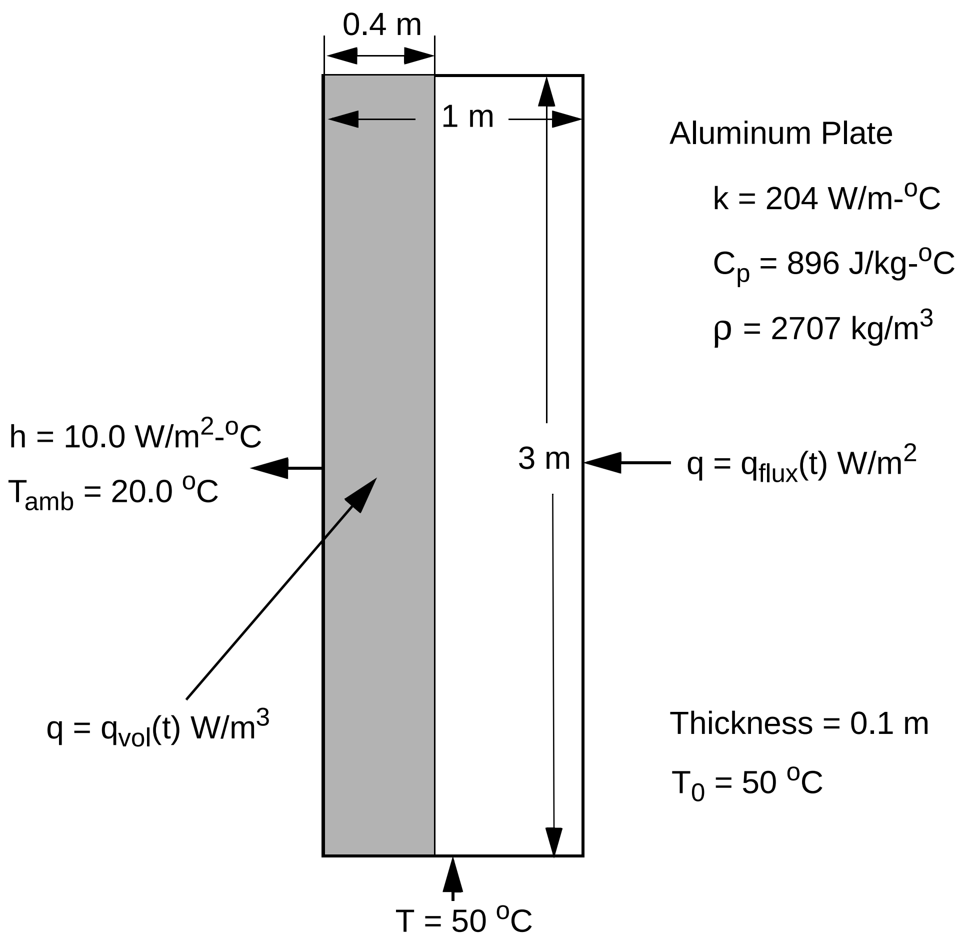
## Objectives:

- 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 subjected 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 2 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 4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

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

*Open Model File:* **New Model**

2. Create the geometry for the plate.

**Geometry/Surface/Corners...**

Fill in the table as the following:

| *X:* | *Y:* | *Z:* | |
|---|---|---|---|
| **0** | **0** | **0** | **OK** |
| **1** | **0** | **0** | **OK** |
| **1** | **3** | **0** | **OK** |
| **0** | **3** | **0** | **OK** |

**Cancel**

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

**View/Autoscale...** *(or use* **< Ctrl>+A** *)*

4. Set the default size for the mesh.

**Mesh/Mesh Control/Default Size...**

*Size:* **0.1**

**OK**

5. 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**

6. Create a property called **plate** to apply to the elements of the plate.

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

7. Create a property called **solid**, to be applied to the model later (when the elements are extruded).

| | |
|---|---|
| *Title:* | **solid** |
| *Material:* | **1..alum** |

**Elem/Property Type...**

| | |
|---|---|
| *Volume Elements:* | ● **Solid** |

**OK**

**OK**

**Cancel**

8. Create the mesh for the model.

**Mesh/Geometry/Surface...**

**Select All**

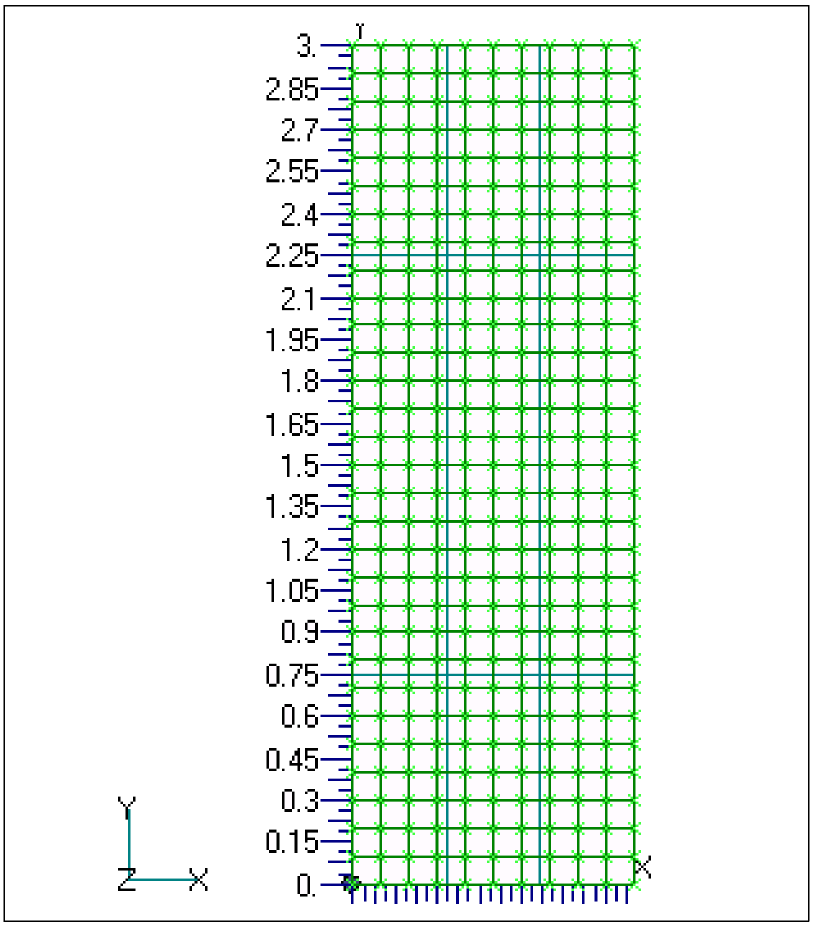
**OK**

*Property:* **1..plate**

**OK**

Your model should appear to be like the following:

Figure 4-1: Plate model with meshed surfaces



9. 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**

10. 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**

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

**Model/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**

12. 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:* | **Temperature** |
| *Temperature:* | **50** |

**OK**

**Cancel**

13. 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:* | **Heat Flux** |
| *Value:* | **5000** |
| *Function Dependence:* | **1..flux_time** |

**OK**

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

**OK**

The heat flux markers should all be pointing from the right.

14. Create the free convection for the model.

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

| | |
|---|---|
| **OK** | |
| *Type:* | **Convection** |
| *Coefficient:* | **10** |
| *Temperature:* | **20** |
| **OK** | |
| *Face:* | *(click on left edge of top left element)* |
| **OK** | |

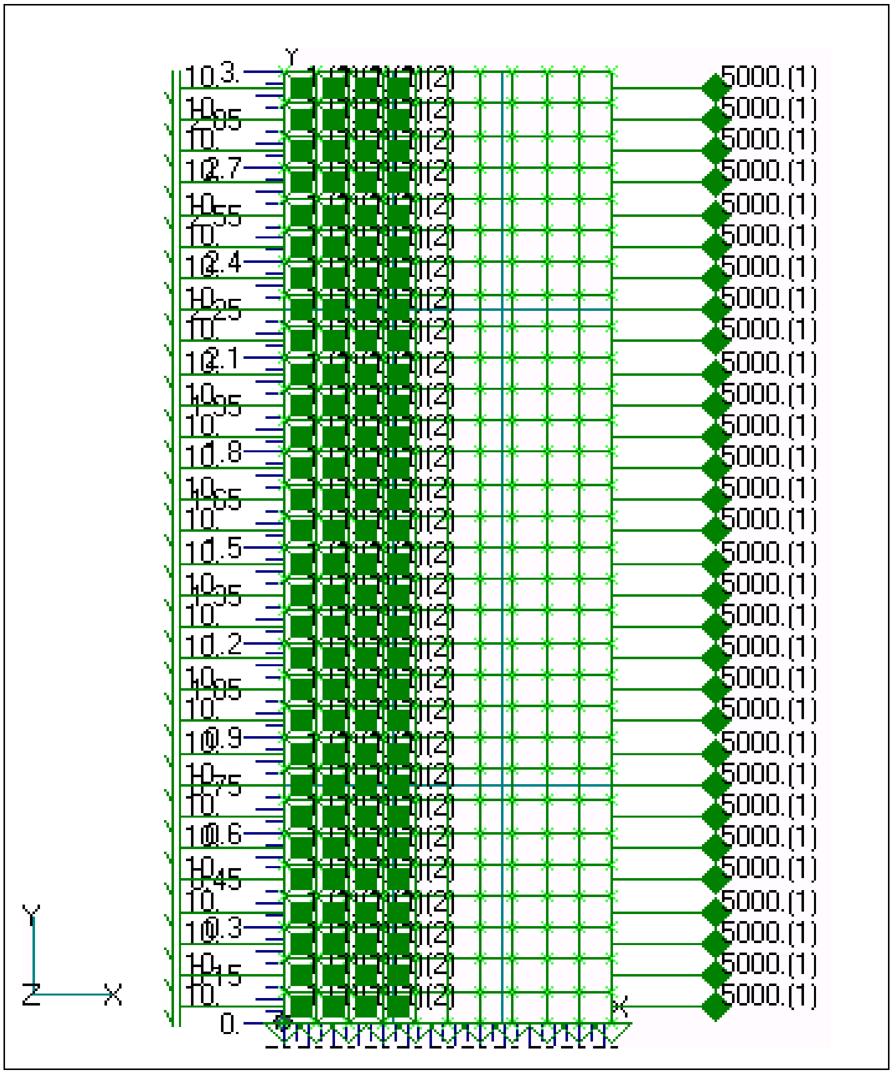
15. Create the volumetric heat generation for the model.

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

| | |
|---|---|
| **OK** | |
| *Type:* | **Heat Generation** |
| *Value:* | **1** |
| *Function Dependence:* | **2..qvol_time** |
| **OK** | |
| **Cancel** | |

Your model should look like the following:

Figure 4-2: Model with loads




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

**File/Export/Analysis Model...**

| | |
|---|---|
| *Type:* | **21..Transient Heat Transfer** |
| **OK** | |
| *File Name:* | **Transient** |
| **Write** | |

*Number of Time Steps:* **100**

*Initial Time Increment:* **10**

☒ **Run Analysis**

**OK**

17. 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 for Windows 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**

18. Remove the thermal loading markers from the screen.

**View/Options...** *(or use* **< Ctrl>+Q** *)*

**Quick Options...**

☐ **Load - Thermal**

☐ **Load - Heat Generation**

☐ **Load - Heat Flux**

☐ **Load - Convection**

**Done**

19. Create a final temperature distribution contour plot.

**View/Select...** *(or use* **<F5>***)*

*Model Style:* ● **Quick Hidden Line**

☒ **Render**

*Contour Style:* ● **Contour**

**Deformed and Contour Data...**

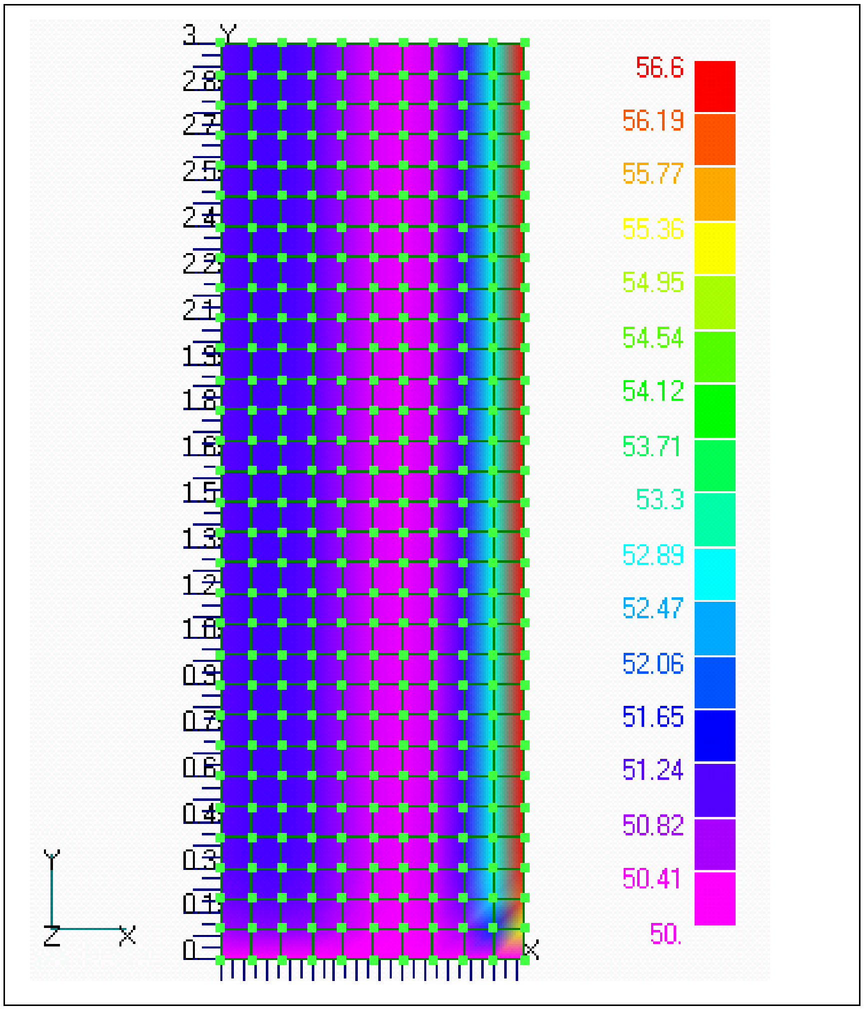
*Output Set:* **9..Case 9 Time 190**

*Contour:* **31..Temperature**

**OK**

**OK**

Figure 4-3: Transient Thermal Analysis



20. 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:* **left edge nodes**

**OK**

**Group/Node/ID**

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

**OK**

21. Now, create the XY plot.

**View/Select...** *(or use* **<F5>***)*

*XY Style:* ● **XY vs Position**

**XY Data...**

*Position:* ● **Y**

*Group:* ● **Select**

**1..left edge nodes**

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

*Output Vector:* **31..Temperature**

**OK**

**OK**

22. Redefine the ranges used by the XY plot.

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

● **Post processing**

*Options:* **XY Axes Style**

*X Tics* **10**

*Y Tics* **21**

**Apply**

| | |
|---|---|
| *Options:* | **XY Y Range/Grid** |
| *Axis Range* | **2..Max Min** |
| *Min* | **49** |
| *Max* | **51** |

**Apply**

**OK**

Compare your results with Figure 4-4.

When done, exit MSC.Nastran for Windows.

**File/Save...** (*or use* <**F4**>)

**File/Exit...**

This concludes this exercise.

Figure 4-4: Temperature along the left edge vs. vertical position