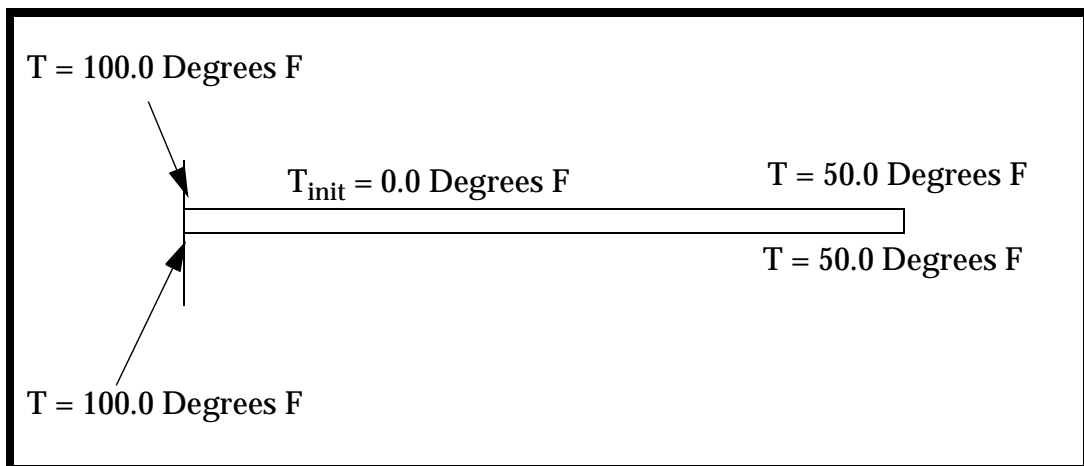


---

## LESSON 24

---

# *Heat Transfer Analysis of a Cantilever Beam*



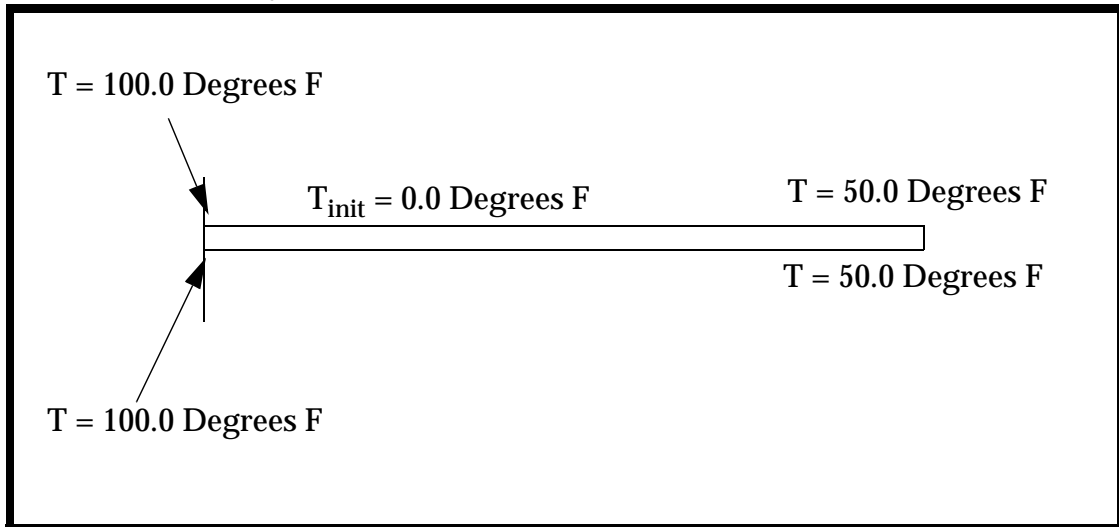
### **Objectives:**

- Subject a cantilever beam with thermal loading, set up in MSC.Marc.
- Accomplish both transient and steady state analyses.



**Model Description:**

In this exercise, you will subject the cantilever beam to thermal loading. The beam will have a temperature range between 50 and 100 degrees at opposing ends. The initial temperature of all nodes is zero. Both a transient and a steady state heat transfer analysis will be done. The transient analysis will be conducted over a period of approximately 1/2 hour to show that the steady state condition is not quite reached in this time.



---

## Exercise Procedure:

1. Open a new database called **beam\_heat\_transfer**.

### File/New ...

*New Database Name:*

**beam\_heat\_transfer**

**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.Marc**.

*Tolerance:*

**Based on Model**

*Analysis Code:*

**MSC.Marc**

*Analysis Type:*

**Thermal**

**OK**

2. Import the old database. Use the cantilever beam model from the first part of this exercise.

### File/Import ...

*Object:*

**Model**

*Source:*

**MSC.Patran DB**

*Import File:*

**cantilever\_beam.db**

This will be the old database just created.

**Apply**

Close the summary form by selecting "OK."

**OK**

3. Now graphically display only the cantilever beam.

### Group/Post...

*Selected Groups to Post:***cantilever\_beam****Apply****Cancel**

4. Create the thermal Properties for the *steel* material. This requires a new constitutive model.

**■ Material***Action:***Create***Object:***Isotropic***Method:***Manual Input***Existing Materials:***steel****Input Properties...***Constitutive Model:***Thermal***Conductivity:***0.000213***Specific Heat:***0.116***Density:***0.00074****OK****Apply**

5. Apply the Material Properties to the model.

**■ Properties***Action:***Create***Object:***2D***Method:***2D Solid***Property Set Name:***thermal\_beam****Input Properties...***Material Name:***m:steel***Thickness:***1.0**

---

**OK**

*Select Members:*

**Surface 1**

**Add**

**Apply**

When the warning that you are going to be overwriting the material properties for Surface 1 appears, enter:

**Yes for All**

#### 6. Apply temperature Loads

Apply the Loads and Boundary Conditions. Define a heat of 50 degrees at the right end of the beam.

#### ■ Loads/BCs

*Action:*

**Create**

*Object:*

**Temp (Thermal)**

*Type:*

**Nodal**

*New Set Name:*

**right\_temp**

**Input Data...**

*Temperature:*

**50**

**OK**

**Select Application Region...**

*Geometry Filter:*

**● Geometry**

*Select Geometric Entities:*

**point 3 4**

**Add**

**OK**

**Apply**

Your display should now show the fixed Boundary Condition on the left end, and the Thermal Temperature Boundary Condition on the right end of the beam, as shown in **Figure 24.1**.

**Figure 24.1:** Beam with fixed and Temperature Boundary Conditions.



Now apply the Loads and Boundary Conditions to the other side of the beam. Define a heat of 100 degrees at the left end of the beam.

New Set Name:

Temperature:

Geometry Filter:  Geometry

Select Geometric Entities:

In **Figure 24.2**, you should see the cantilever beam, including the left end Temperature Boundary Condition.

**Figure 24.2:** Cantilever Beam with given Heat and Displacement Boundary Conditions.



7. Apply Initial Conditions.

Now apply the initial Condition of zero degrees to the entire beam.

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Initial Temperature"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name:</i>	<input type="text" value="init_temp"/>
<input type="button" value="Input Data..."/>	
<i>Temperature:</i>	<input type="text" value="0.0"/>
<input type="button" value="OK"/>	
<input type="button" value="Select Application Region..."/>	
<i>Geometry Filter:</i>	<input checked="" type="radio"/> Geometry
<i>Select Geometric Entities:</i>	<input type="text" value="Surface 1"/>
<input type="button" value="Add"/>	

**OK****Apply**

8. Set up the model for Analysis.

Create the necessary **Analysis Load Steps** and submit the job for Analysis.

### ■ Analysis

<i>Action:</i>	<b>Analyze</b>
<i>Object:</i>	<b>Entire Model</b>
<i>Method:</i>	<b>Full Run</b>
<i>Job Name:</i>	<b>heat_transfer</b>

### **Load Step Creation...**

<i>Job Step Name:</i>	<b>Transient Heat Step</b>
<i>Solution Type:</i>	<b>Transient Heat Transfer</b>

### **Solution Parameters...**

<i>Increment Type:</i>	<b>Fixed</b>
<i>Time Step Size:</i>	<b>200.0</b>
<i>Total Time:</i>	<b>2000.0</b>

**OK**

### **Select Load Case...**

<i>Available Load Case:</i>	<b>Default</b>
-----------------------------	----------------

**OK****Apply**

<i>Job Step Name:</i>	<b>Steady State Step</b>
<i>Solution Type:</i>	<b>Steady State Heat Transfer</b>

### **Select Load Case...**

<i>Available Load Case:</i>	<b>Default</b>
-----------------------------	----------------

**OK**

---

**Apply**

**Cancel**

**Load Step Selection...**

*Existing Job Steps:*

**Transient Heat Step**  
**Steady State Step**

*Selected Job Steps:*

**Default Static Step**

**OK**

**Apply**

Monitor the analysis for job completion. The *heat\_transfer.log* is an ASCII file which contains Element, Loads & Boundary Conditions, Material Translation, Step Control parameters, Equilibrium and Error information. Another useful ASCII file is the *heat\_transfer.sts* file. This file contains a summary of job information; including step number, number of increments, number of iterations, total time of step, and time of a given increment. The *heat\_transfer.out* file contains a summary of any job errors. These files can be viewed during or after a job has completed. A more convenient method might be to use the **Analysis** application, **Monitor**.

*Action:*

**Monitor**

*Object:*

**Job**

**View Status File...**

After the job has finished, a successful completion will end with the line: Job ends with exit number: 3004

9. Read in the results from the two analyses.

### ■ Analysis

*Action:*

**Read Results**

*Object:*

**Result Entities**

*Method:*

**Attach**

*Available Jobs:*

**heat\_transfer**

**Select Results File...**

**heat\_transfer.t16**

**OK**

**Apply**

10. Plotting the temperature distribution.

Plot the temperature with respect to time for all nodes on the bottom of the beam.

## ■ Results

Action:

Create

Object:

Graph

Select the **Target Entity** icon



Target Entity:

Nodes

Nodes:

Node 1:9

Go to the **Select Results** form



Select **View Subcases** icon, highlight the **Harmonic Step** Case, then choose the **Select Subcases** icon.



Select Result Cases:

Transient Heat Step,  
...subcases



Filter

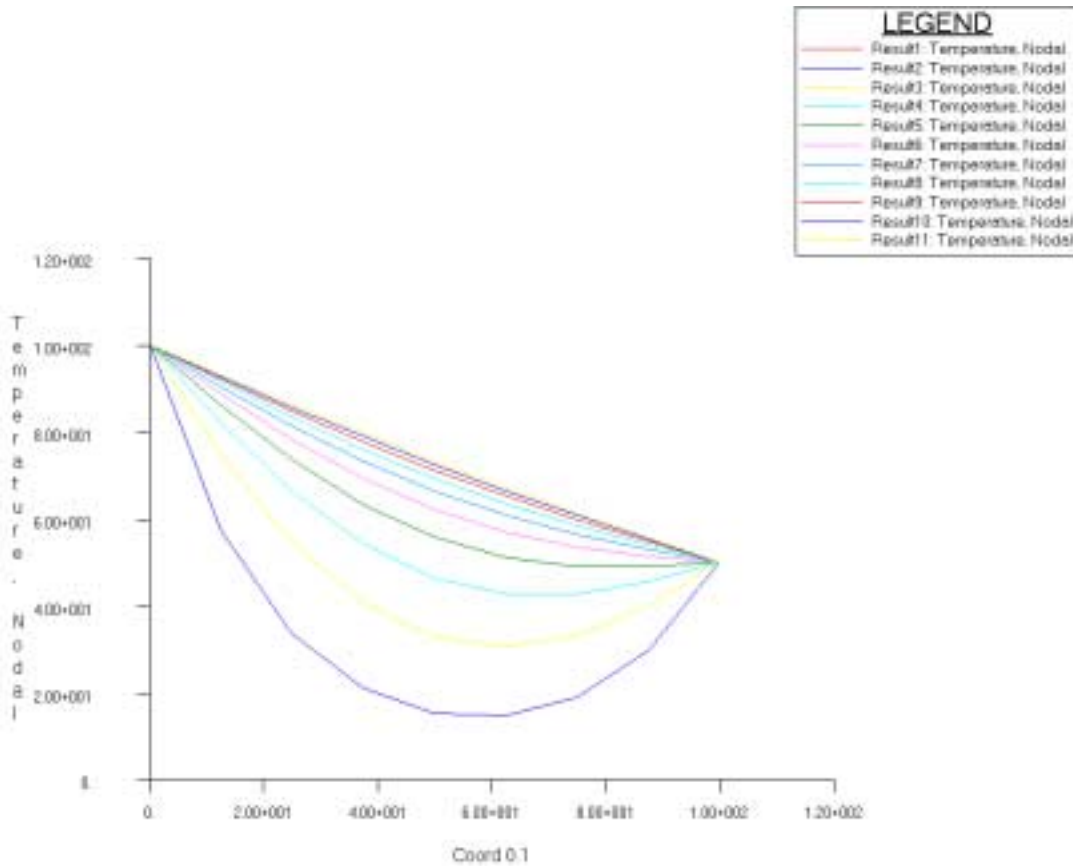
Apply

Close

Y:	Result
Select Y Result:	Temperature, Nodal
X:	Coordinate
Select Coordinate Axis:	Coord 0.1
<b>Apply</b>	

The results are shown below, in Figure 24.2. A separate curve for each load increment (time step) is plotted showing the temperature distribution over the length of the beam.

**Figure 24.2 - Result of the Analysis**



The first curve is at zero, coincident with the x-axis, corresponding to the initial conditions. The last plot is the steady state solution, being a linear temperature distribution between 100 and 50 degrees. The other curves are different states of the transient solution over time. Note that the time was not quite long enough to obtain the steady state solution. Also note that because of the coarseness of the mesh, the results are

not as accurate as they would be with a fine mesh. This would be apparent if you were to redo the transient step and do about 50 steps in the first 20 seconds. You will notice that an instability occurs where the temperatures in a couple of locations on the beam dip below zero which is physically impossible. A finer mesh would take care of that problem.

Close the database and quit PATRAN.

This concludes the exercise.

