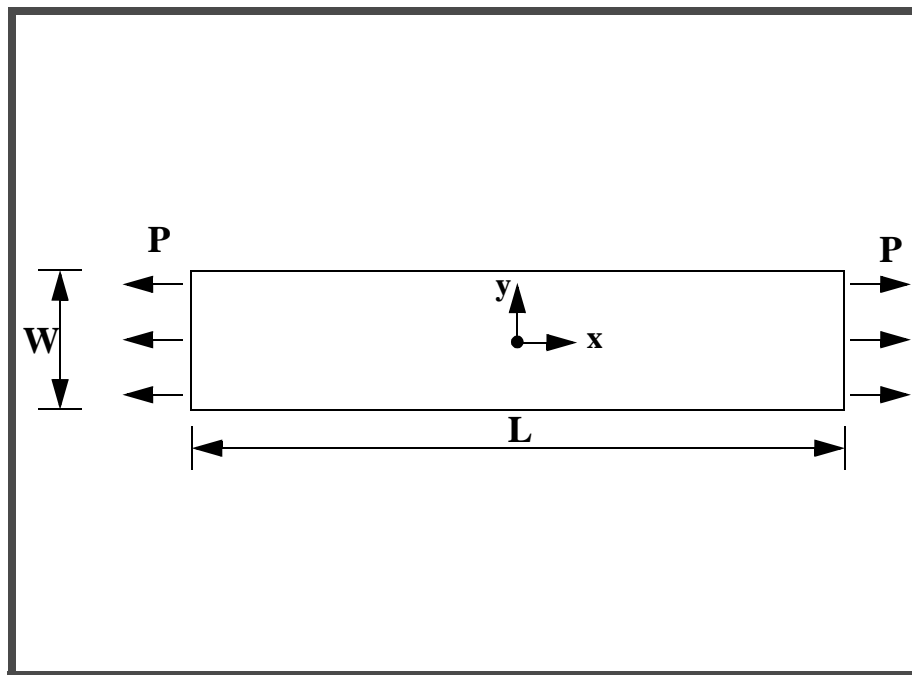


## WORKSHOP 6

# *Elasto-Plastic Deformation of a Thin Plate*



### Objectives:

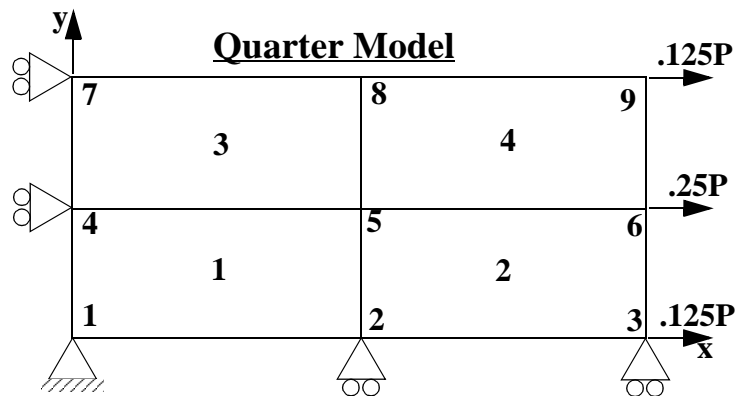
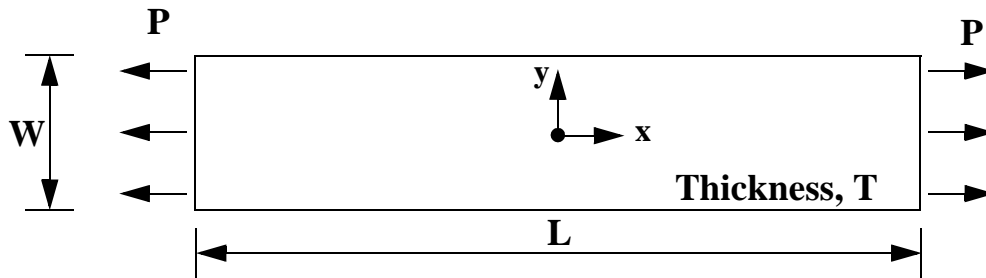
- Create a model with elasto-plastic material properties.
- Submit an MSC.Nastran nonlinear analysis.
- Generate an accurate deformation plot of the model.



**Model Description:**

The figure below is a finite element representation of an elasto-plastic material under a load. A nonlinear analysis with load increments will be performed on a quarter model to obtain the deformation of this element subjected to the following load history.

**Figure 6.1**



**Table 6.1 - Material Properties**

<b>Elastic Modulus:</b>	<b>3.0E6 psi</b>
<b>Poisson's Ratio:</b>	<b>0.25</b>
<b>Length:</b>	<b>50.0 in</b>
<b>Width:</b>	<b>10.0 in</b>
<b>Thickness:</b>	<b>0.1 in</b>
<b>Plasticity Modulus:</b>	<b>3.0E4 psi</b>
<b>Intial Yield Stress:</b>	<b>850 psi</b>
<b>Load:</b>	<b>800 lbs</b>

---

**Table 6.2 - Load Cases**

<b>Subcase</b>	<b>Load Factor</b>	<b>Load Increments</b>	<b>Work Error</b>
1	1.0	1	off
2	1.25	8	off
3	1.1875	5	off
4	0	2	off

**Exercise Procedure:**

1. Start up MSC.Nastran for Windows V4.0.2 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**

(Optional) For users who wish to remove the default rulers in the work plane model, please do the following:

**View/Options...**

*Category:*

**Tools and View Style**

*Options:*

**Workplane and Rulers**

**Draw Entity**

**OK**

2. Create a material called **mat\_1**.

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

**Model/Material...**

*Title:*

**mat\_1**

*Youngs Modulus:*

**3.0E6**

*Poisson's Ratio:*

**0.25**

**Nonlinear >>**

Define the elasto-plastic properties:

*Nonlinearity Type:*

**Elasto-Plastic (Bi-Linear)**

*Plasticity Modulus, H:*

**3.0E4**

*Initial Yield Stress:*

**850**

**OK**

**OK**

**Cancel**

3. Create the property that will define the elasto-plastic element.

### Model/Property...

**Elem/Property Type...**

*Plane Elements:*

**Plate**

**OK**

*Title:*

**prop\_1**

To select the material, click on the list icon next to the databox and select **mat\_1**.

*Material:*

**1..mat\_1**

*Thickness (T1):*

**0.1**

**OK**

**Cancel**

4. Create the NASTRAN finite element model.

### Mesh/Between...

*Property:*

**1..prop\_1**

*Mesh Size/#Nodes/Dir1:*

**3**

*Mesh Size/#Nodes/Dir2:*

**3**

**OK**

*Corner 1:*

X:	Y:	Z:
<b>0</b>	<b>0</b>	<b>0</b>

**OK**

*Corner 2:*

X:	Y:	Z:
<b>25</b>	<b>0</b>	<b>0</b>

**OK**

X:	Y:	Z:
----	----	----



---

Next, constrain the nodes on the left edge of the model. Select **Node 1, Node 4, and Node 7.** (**Hint:** Use the Shift key and the left mouse button for rectangular picking.)

TX    TY    TZ  
 RX    RY    RZ

Respond **No** when asked “OK to Overwrite (No=Combine)?”

Now constrain the nodes on the bottom edge of the model. Select **Node 1, Node 2, and Node 3.** (**Hint:** Use the Shift key and the left mouse button for rectangular picking.)

TX    TY    TZ  
 RX    RY    RZ

Respond **No** when asked “OK to Overwrite (No=Combine)?”

After creating all the constraints, redraw the viewport by selecting:

### View/Redraw

6. Create the model loading.

Like the constraints, a load set must first be generated before creating the appropriate model loading.

### Model/Load/Set...

*Title:*

Since this is a nonlinear analysis, the nonlinear analysis load set options must first be defined.

**Model/Load/Nonlinear Analysis...***Solution Type:* **Static****Defaults...***Number of Increments:***1***Stiffness Updates/Method:***1..AUTO***Convergence Tolerances:* **Work****OK**

Next, create the load.

**Model/Load/Nodal...**

Select **Node 3 & Node 9**.

**OK**

Highlight **Force**.

*FX* **100****OK**

Select **Node 6**.

**OK***FX* **200****OK****Cancel**

Create the second load set.

**Model/Load/Combine...***Scale Factor:***1.25**

---

*From Set:*

1..load\_1

**Last One**

Modify the nonlinear analysis options.

**Model/Load/Nonlinear Analysis...**

*Number of Increments:*

8

**OK**

Create the third load set.

**Model/Load/Combine...**

*Scale Factor:*

1.1875

*From Set:*

1..load\_1

**Last One**

Modify the nonlinear analysis options.

**Model/Load/Nonlinear Analysis...**

*Number of Increments:*

5

**OK**

Create the final load set.

**Model/Load/Combine...**

*Scale Factor:*

0

*From Set:*

1..load\_1

**Last One**

Modify the nonlinear analysis options.

**Model/Load/Nonlinear Analysis...**

*Number of Increments:*

2

**OK**

7. Submit the job for analysis.

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

Type:

10..Nonlinear Static

OK

Change the directory to C:\temp.

File name:

prob6

Write

Run Analysis

Loads...

Select All

OK

Output Requests/Output Types:

2..Displacements and Stresses

OK

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

Yes

File name:

prob6

Save

Respond **No** when asked “OK to Read Nonlinear Stresses and Strains?”

No

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

---

Respond **Yes** when asked “OK to Read Nonlinear Stresses and Strains?”

**Yes**

8. List the results of the analysis.

To list the results, select the following

**List/Output/Query...**

*Output Set:*

**All Sets**

*Category:*

**Displacement**

*Entity:*

**Node**

*ID:*

**3**

**OK**

**NOTE:** You may want to expand the message box in order to view the results. To do this, double click on the message box. Adjust the size of the box to your preference by dragging the top border downward.

Answer the following questions using similar procedure. The answers are listed at the end of the exercise.

What is the T1 displacement of **Node 3** at the end of each load step?

Step 1 T1 Disp. @ Node 3 = \_\_\_\_\_

Step 2 T1 Disp. @ Node 3 = \_\_\_\_\_

Step 3 T1 Disp. @ Node 3 = \_\_\_\_\_

Step 3 T1 Disp. @ Node 3 = \_\_\_\_\_

9. Display the deformed plot on the screen.

First, you may want to remove the labels and LBC markers in order to give a better view of the deformation.

**View/Options...**

**Quick Options...**

**Labels Off**

*Draw:*

**Load - Force**

Constraint

Done

OK

Plot the deformation of the structure.

**View/Select...**

*Deformed Style:*

Deform

Deformed and Contour Data...

*Data Selection/Category:*

1..Displacement

*Output Set:*

4..Case 4 Time 4

*Output Vectors/Deformation:*

1..Total Translation

OK

OK

In order to see the deformation results accurately, you will need to turn off the display scaling of the actual deformation.

**View/Options...**

*Category:*

PostProcessing

*Options:*

Deformed Style

% of Model (Actual)

OK

10. Display the Stress and Strain distribution of the plate

First, look at the Stress distribution.

**View/Select...**

*Contour Style:*

Contour

Deformed and Contour Data...

*Data Selection/Category:*

4..Stress

---

*Output Set:*

4..Case 4 Time 4

*Output Vectors/Contour:*

7033..Plate Top VonMises  
Stress

OK

OK

Next, look at the Von Mises Strain distribution.

**View/Select...**

*Contour Style:*

Contour

Deformed and Contour Data...

*Data Selection/Category:*

5..Strain

*Output Set:*

4..Case 4 Time 4

*Output Vectors/Contour:*

7077..Plate Top VonMises  
Strain

OK

OK

This concludes the exercise.

<i>Step 1 Disp X:</i>	<b>6.66667E-3</b>
<i>Step 2 Disp X:</i>	<b>1.33333E-1</b>
<i>Step 3 Disp X:</i>	<b>1.32920E-1</b>
<i>Step 4 Disp X:</i>	<b>1.25010E-1</b>