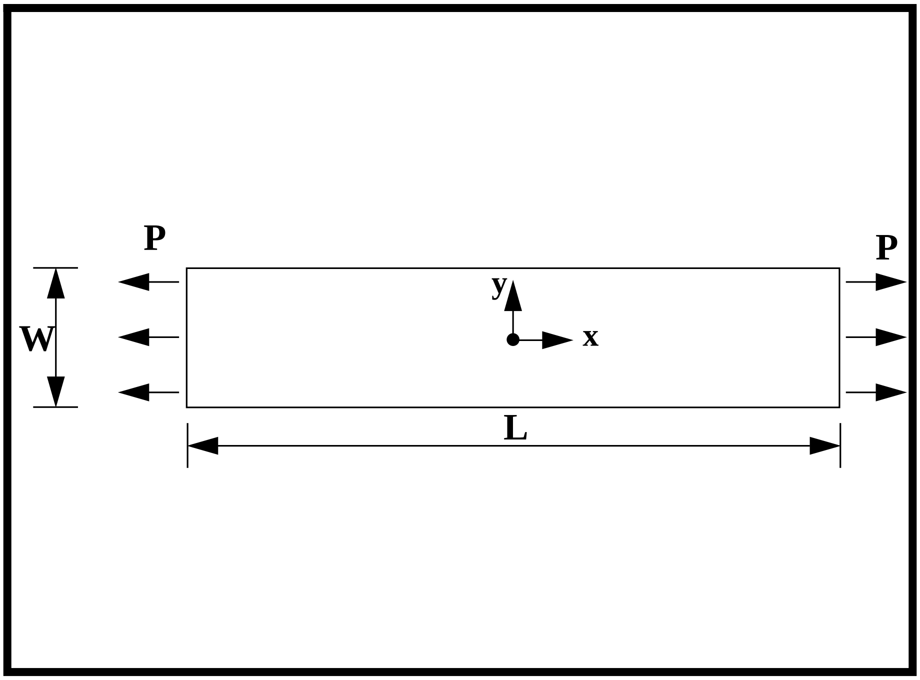
WORKSHOP PROBLEM 6

Elasto-Plastic Deformation of a Thin Plate



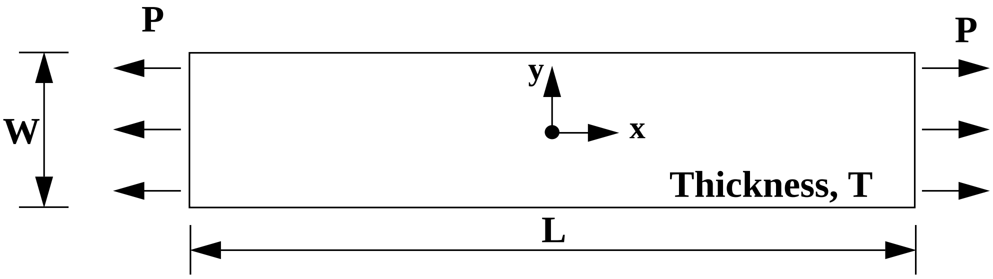
Objectives:

- 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



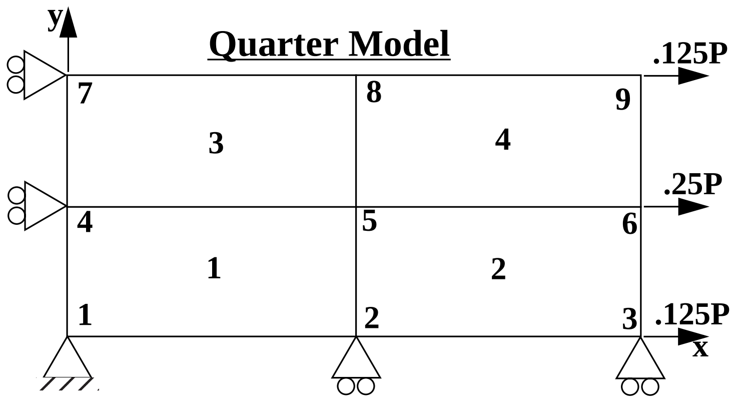


Table 6.1 - Material Properties

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

Table 6.2 - Load Cases

Subcase	Load Factor	Load Increments	Work Error
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 V3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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...

● **Tools and View Style**

Category: **Workplane and Rulers**

Draw Entity ☐

Apply

Cancel

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

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**25**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**25**	**5**	**0**

OK

	X:	*Y:*	*Z:*
Corner 4:	**0**	**5**	**0**

OK

To bring the model into the viewable area, use the Autoscale feature.

View/Autoscale...

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set must be created. Do so by performing the following:

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Constrain the bottom left corner node. Select **Node 1.**

OK

In the *DOF* box, check the following boxes:

☐ **TX** ☐ **TY** ☒ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

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.)

OK

☒ **TX** ☐ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

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

No

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.)

OK

☐ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

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

No

Cancel

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: **load_1**

OK

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...

Analysis 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 **Yes** when asked "OK to Read Nonlinear Stresses and Strains?"

Yes

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

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

☐ **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 Equivalent 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.

Step 1 Disp X:	**6.66667E-3**
Step 2 Disp X:	**1.33333E-1**
Step 3 Disp X:	**1.32920E-1**
Step 4 Disp X:	**1.25010E-1**