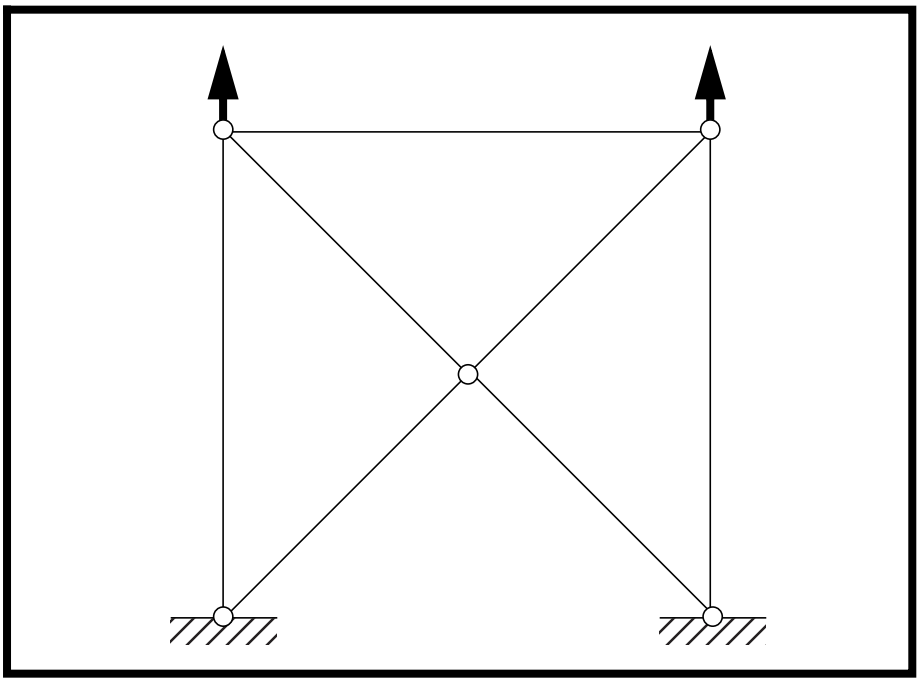
WORKSHOP PROBLEM 8

Elasto-Plastic Deformation of a Truss Structure

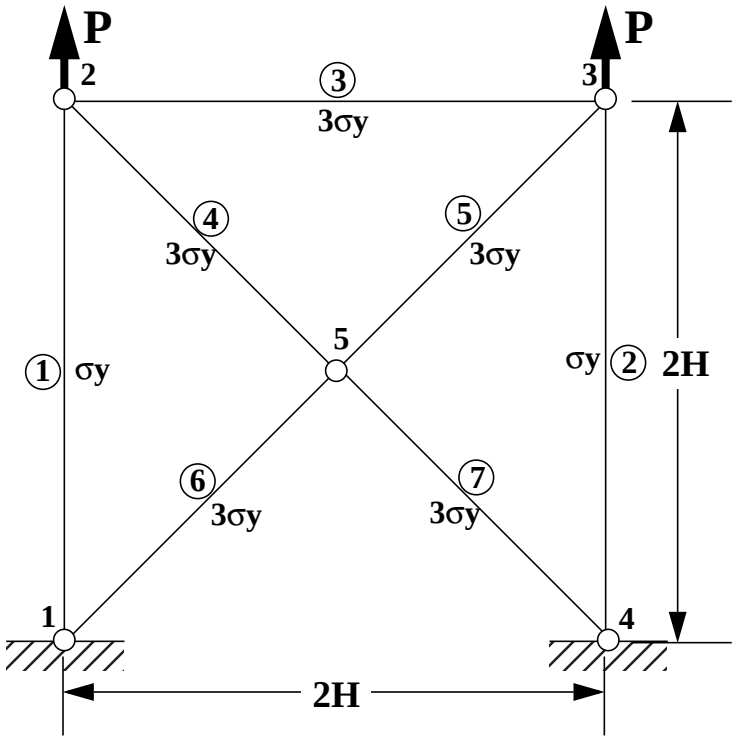


Objectives:

- Demonstrate the use of elastic-plastic material properties.
- Create an enforced displacement on the model.
- Run an MSC/NASTRAN nonlinear static analysis.
- Create an accurate deformation and fringe plot of the model.

Model Description:

Figure 8.1 - The Structure and Material Properties



2H = 10.
A = 1.
E = 2.e5
E_T = 0
σ_Y = 100
P = 0.05

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: **2e5**

Nonlinear >>

● **Elasto-Plastic (Bi-Linear)**

Initial Yield Stress: **100**

OK

OK

Title: **mat_2**

Youngs Modulus: **2e5**

Nonlinear >>

● **Elasto-Plastic (Bi-Linear)**

Initial Yield Stress: **300**

OK

OK

Cancel

3. Create a property called **prop_1** for the bar elements of the model.

Model/Property...

Title: **prop_1**

Material: **1..mat_1**

Elem/Property Type...

Change the property type from plate elements (default) to rod elements.

Line Element: ● **Rod**

OK

Area, A: **1.0**

OK

Title: **prop_2**

Material: **2..mat_2**

Area, A: **1.0**

OK

Cancel

4. Create the relevant NASTRAN geometry.

Create the first node of the model by doing the following:

Model/Node...

X:	*Y:*	*Z:*
0	**0**	**0**

OK

Repeat the process for the other 4 nodes.

X:	*Y:*	*Z:*	
0	**10**	**0**	**OK**
10	**10**	**0**	**OK**
10	**0**	**0**	**OK**
5	**5**	**0**	**OK**

Cancel

To fit the display onto the screen, use the autoscale feature.

View/Autoscale

Now, connect the nodes to create the rod elements.

Model/Element...

Property: **1..prop_1**

Nodes:	**1**	**2**	**OK**
Nodes:	**3**	**4**	**OK**

Property: **2..prop_2**

Nodes:	**2**	**3**	**OK**
Nodes:	**2**	**5**	**OK**

Nodes:	**3**	**5**	**OK**
Nodes:	**1**	**5**	**OK**
Nodes:	**4**	**5**	**OK**

Cancel

5. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created by performing the following:

Model/Constraint/Set...

Title: **fixity**

OK

Now define the end constraints for the model.

Model/Constraint/Nodal...

Select **Node 1 and 4**.

OK

On the *DOF* box, select the following boxes.

☒ **TX** ☒ **TY** ☒ **TZ**

OK

Cancel

6. Create the model loading.

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

Model/Load/Set...

Title: **load_1**

OK

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

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Default...

Max Iterations/Step: **25**

Stiffness Update / Method: **1..AUTO**

Output Control / Intermediate: **1..YES**

☒ Displacement **0.001**

☐ Load

☒ Work **1e-7**

OK

Next, create the nodal displacement at the top edge of the model.

Model/Load/Nodal...

Select **Node 2 and 3**.

OK

Highlight **Displacement**.

TY ☒ **0.05**

OK

Cancel

Now, define the constraint necessary to keep the model static when modeling an enforced nodal displacement.

Model/Constraint/Nodal...

Select **Node 2 and 3**.

OK

☐ **TX** ☒ **TY** ☐ **TZ**

OK

Cancel

7. Submit the job for analysis.

In order for the solver to account for the preload, this job must be submitted as a nonlinear analysis.

File/Export/Analysis Model...

Analysis Type: **10..Nonlinear Static**

OK

Change the directory to **C:\temp**.

File name: **prob8**

Write

☒ **Run Analysis**

☒ **Large Disp**

Advanced...

Problem ID: **Elasto-Plastic Deformation of Truss**

OK

Under *Output Requests*, deselect everything except the following:

☒ **Displacement**

☒ **Element Stress**

Also, change output request to:

Output Request: **2..Print and PostProcess**

OK

OK

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

Yes

File name: **prob8**

Save

When asked if it is “OK to read Nonlinear Stresses and Strains”, respond **Yes**.

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

When asked if it is “OK to Begin Reading File C:\TEMP\prob8.xdb”, respond **Yes**.

Yes

8. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set: **40..Case 20 Step 1.000000**

Category: **1..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 the results. What are the x and y displacements of Node 3 at the end of the subcase?

T1= _____

T2= _____

List/Output/Query...

Output Set: **40..Case 20 Step 1.000000**

Category: **4..Stress**

Entity: ● **Elem**

ID: **1**

OK

What is the stress in Element 1 at the end of the subcase?

Stress = _____

What is the stress in Element 3 at the end of the subcase?

Stress = _____

What is the stress in Element 4 at the end of the subcase?

Stress = _____

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 - Displacement**

☐ **Constraint**

Done

OK

Next, reduce the magnification of the model.

View/Magnify...

Magnification Factor **0.8**

OK

Plot the deformation of the structure.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **40..Case 20 Step 1.000000**

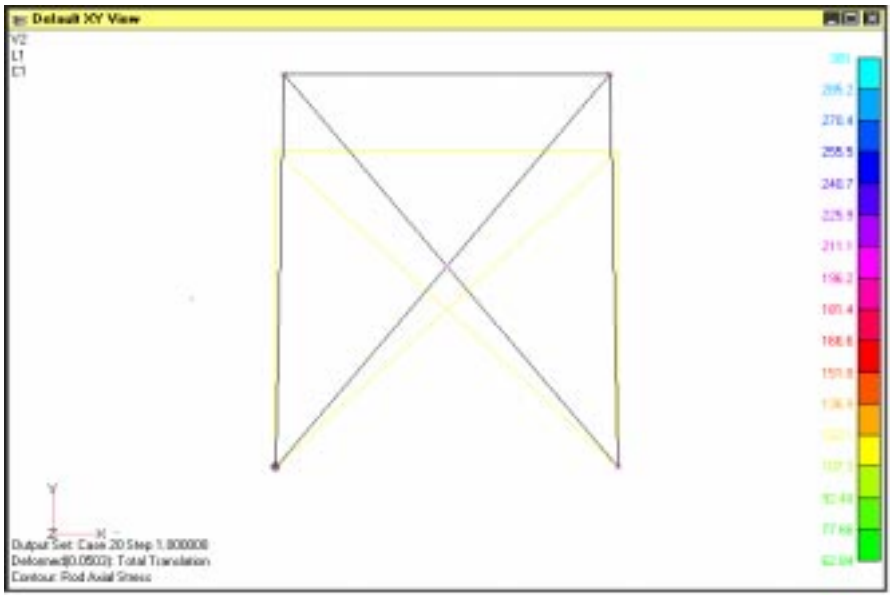
Output Vectors/Deformation: **1..Total Translation**

Output Vectors/Contour: **3183..Road Axial Stress**

OK

OK

The XY view should appear as follows:



This concludes the exercise.

Elasto-Plastic Deformation of Truss

Displacement	T1 (X Disp)	T2 (Y Disp)
Node 3	**-0.0052873**	**0.05**

Stress	Axial Stress
Elem 1	**100**
Elem 3	**-211.49**
Elem 4	**300**