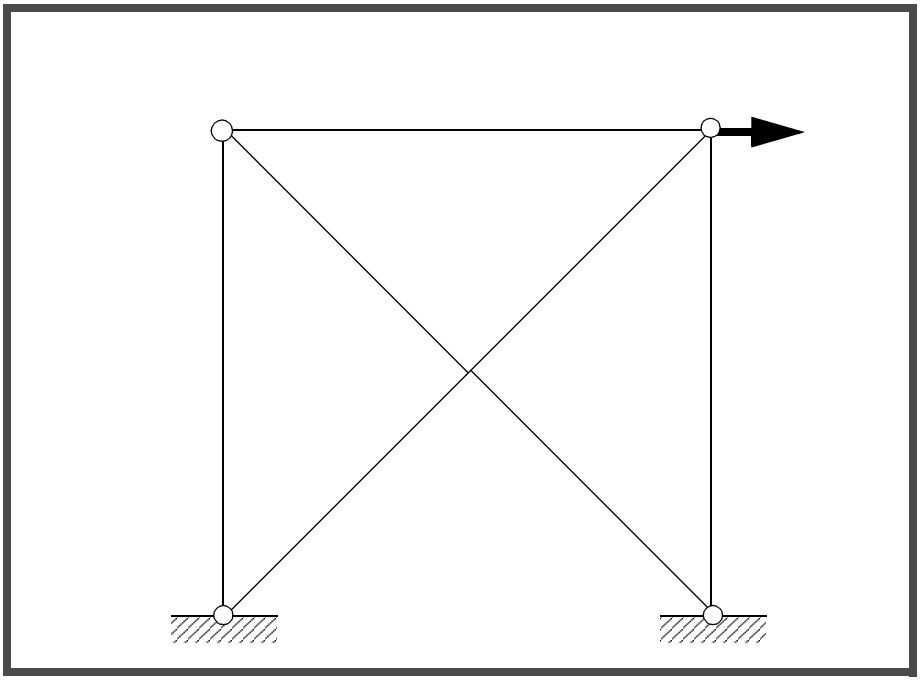
APPENDIX D

Cable Tension



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

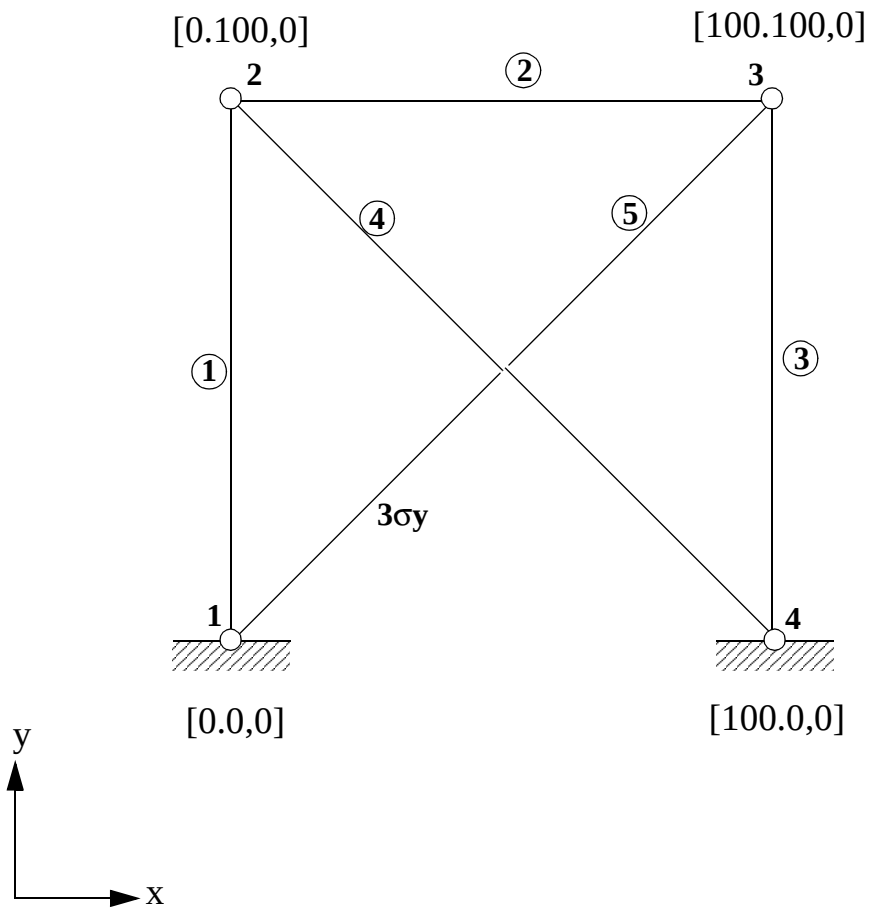


Table D.1 - Material Properties

Material:	**Steel**
Youngs Modulus:	**30e6**

Material:	**Wire**	
Youngs Modulus:	**30e6**	
Stress vs. Strain:	**X:**	**Y:**
	-0.01	**-1**
	0	**0**
	0.01	**300000**

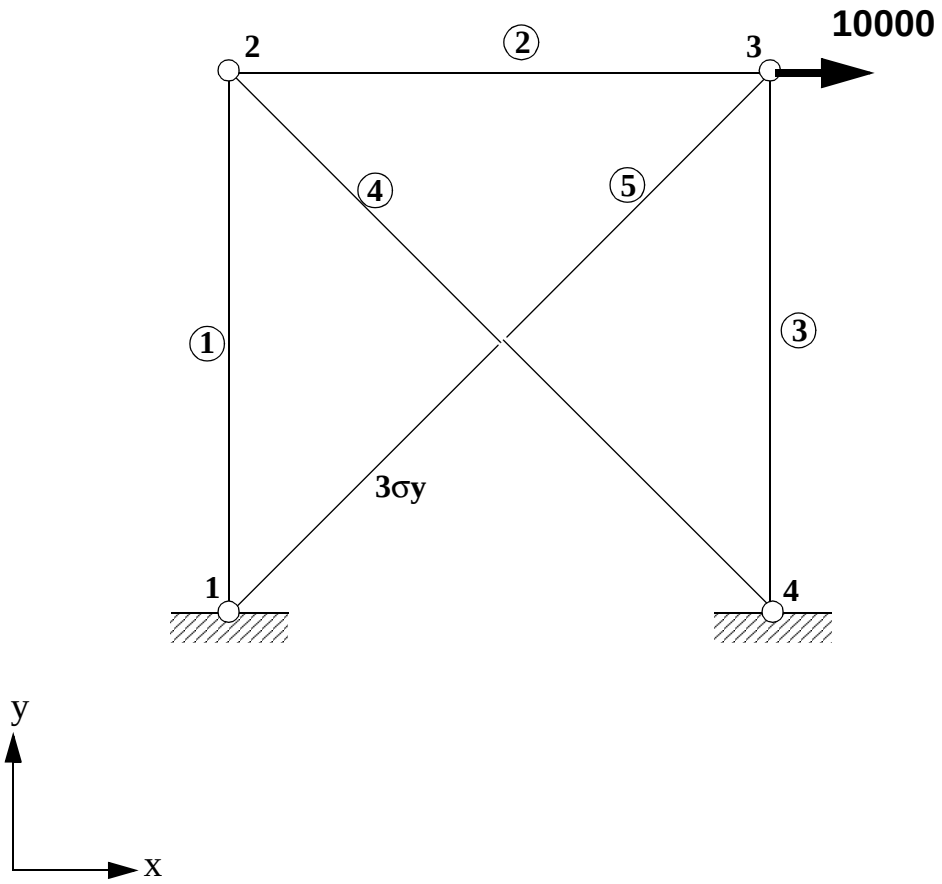
Table D.2a - Rod Element Properties

Material:	**Steel**
Line Element:	**Rod**
Area:	**10**

Table D.2b - Cable Element Properties

Material:	**Wire**
Line Element:	**Rod**
Area:	**1.0**

Cable Tension



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 function to define the nonlinear material properties.

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

Model/Function...

Title: **Stress vs. Strain**

Type: **13..Stress vs. Strain**

Data Entry

X:	Y:	
-0.01	-1	More
0	0	More
0.01	300000	OK

Cancel

3. Create a material called **steel**

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

Model/Material...

Title:	steel
Youngs Modulus:	30.e6

OK

Title:	Wire
Young Modulus:	30.e6

Nonlinear >>

● **Nonlinear Elastic**

Functin Dependence:	1..Stress vs. Strain

OK

OK

Cancel

4. Create a property called **rod** for the bar elements of the model.

Model/Property...

Title:	rod
Material:	1..steel

Elem/Property Type...

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

Line Element: ● **Rod**

OK

Area, A:	10

OK

Title:	cable

Material:	**2..wire**
Area, A:	**1.0**

OK

Cancel

5. 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	**100**	**0**	**OK**
100	**100**	**0**	**OK**
100	**0**	**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..rod**

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

Property: **2..cable**

Nodes: **2** **4** **OK**

Nodes: **1** **3** **OK**

Cancel

6. Create the model constraints.

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

Model/Constraint/Set...

Title: **constraint_1**

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**

OK

Cancel

7. 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

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

Model/Load/Nodal...

Select **Node 3**.

OK

Highlight **Force**

FX ☒ **10000**

OK

Cancel

8. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **1..Static**

OK

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

File name: **probD**

Write

☒ **Run Analysis**

OK

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

Yes

File name: **probD**

Save

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

9. Display the deformed plot on the screen.

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

View/Options...

Quick Options...

Labels Off

All Entities Off

☒ **Element**

Done

OK

Plot the deformation of the structure.

View/Select...

Deform Style: ● **Vector**

Contour Style: ● **Criteria**

Deformed and Contour Data...

Output Set: **1..MSC/NASTRAN Case 1**

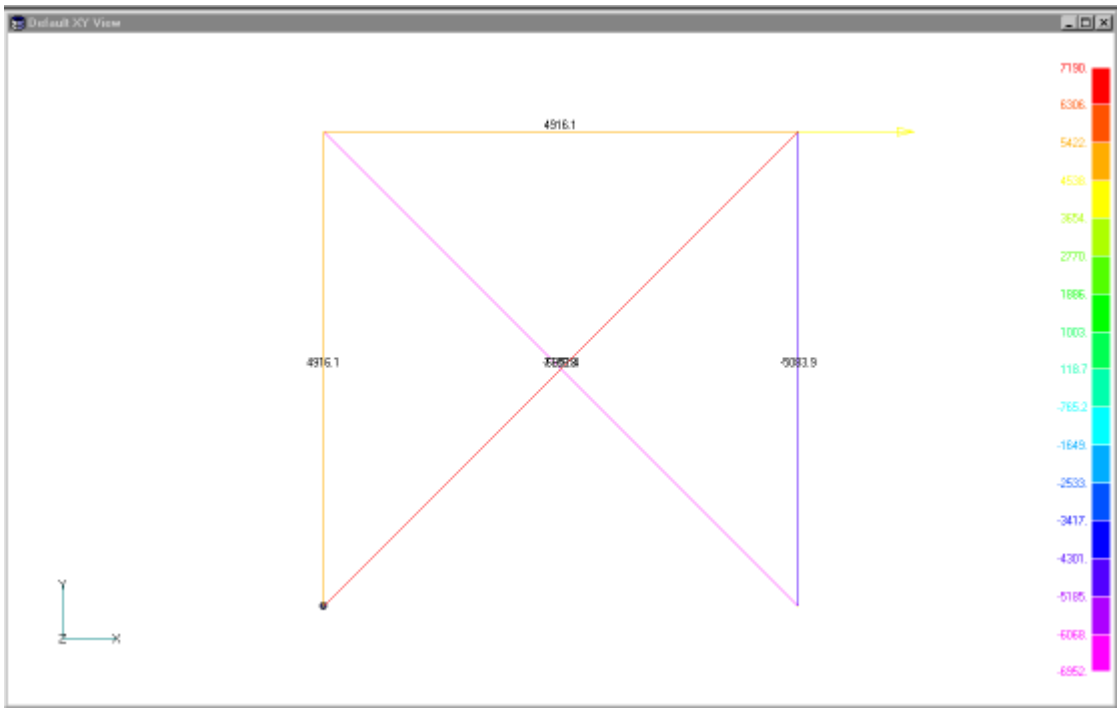
Output Vector/Deformation: **41..Total Applied Force**

Output Vectors/Contour: **3036..Rod Axial Force**

OK

OK

The XY view should appear as follows:



10. List the results of the analysis.

To list the results, select the following:

List/Output/Standard...

Select All

OK

Format ID: **10..NASTRAN CROD Forces**

OK

Select **Element 4 and 5.**

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.

11. define load set options for nonlinear analysis

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Default

OK

12. Create an equivalence load in the opposite direction of load_1

Model/Load/Combine...

Scale Factor: **-1**

Last One

13. 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 Analysis**

OK

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

File name: **probD_nle**

Write

☒ **Run Analysis**

Advance

OK

Under *Output Requests*, deselect **Element Stress**

☒ **Element Stress**

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

☒ **Loads=** **1..load_1**

Write Case

When you get confirmation that the subcase was written, click **OK.**

OK

SUBCASE ID: **2**

☒ **Loads=** **2..Combined Set**

OK

OK

OK

When asked if you wish to save the model, 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

14. Display the deformed plot on the screen.

First, you may want to remove the labels to give a better view of the deformation.

View/Options...

Quick Options...

Labels Off

All Entities Off

☒ **Element**

Done

OK

Plot the deformation of the structure.

View/Select...

Deform Style: ● **Vector**

Contour Style: ● **Criteria**

Deformed and Contour Data...

Output Set: **2..Case 1 Time 1.**

Output Vector/Deformation: **41..Total Applied Force**

Output Vectors/Contour: **3036..Rod Axial Force**

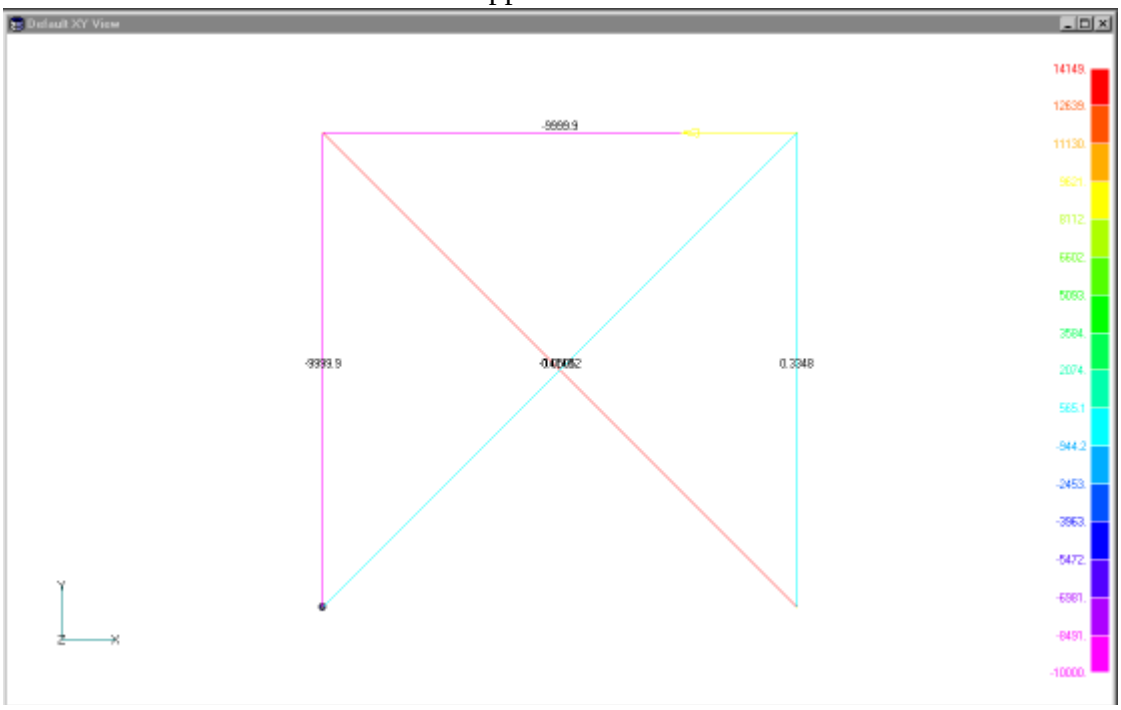
OK

OK

Repeat Step 14 and select the following for *Output Set:*

Output Set: **3..Case 2 Time 2.**

The XY view should appear as follows:



15. List the results of the analysis.

To list the results, select the following:

List/Output/Standard...

Select All

OK

Format ID: **10..NASTRAN CROD Forces**

OK

Select **Element 4 and 5.**

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.