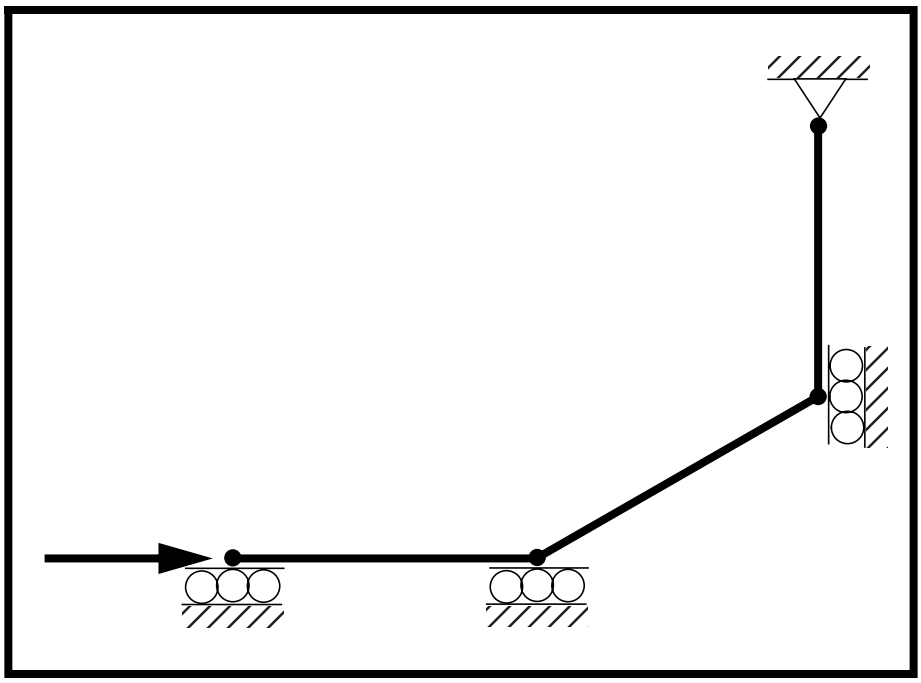
WORKSHOP PROBLEM 3

Load Deflection of a 3-Rod Structure



Objectives:

- Create a model of a 3-rod structure.
- Apply the appropriate constraints and load.
- Submit an MSC/NASTRAN nonlinear analysis.
- Create an animation of the deformation of the structure.

Model Description:

Below in Figure 3.1 is a finite element representation of a 3-rod structure under a load. A nonlinear analysis with load increments will be performed to obtain the load-deflection behavior of the structure.

Figure 3.1

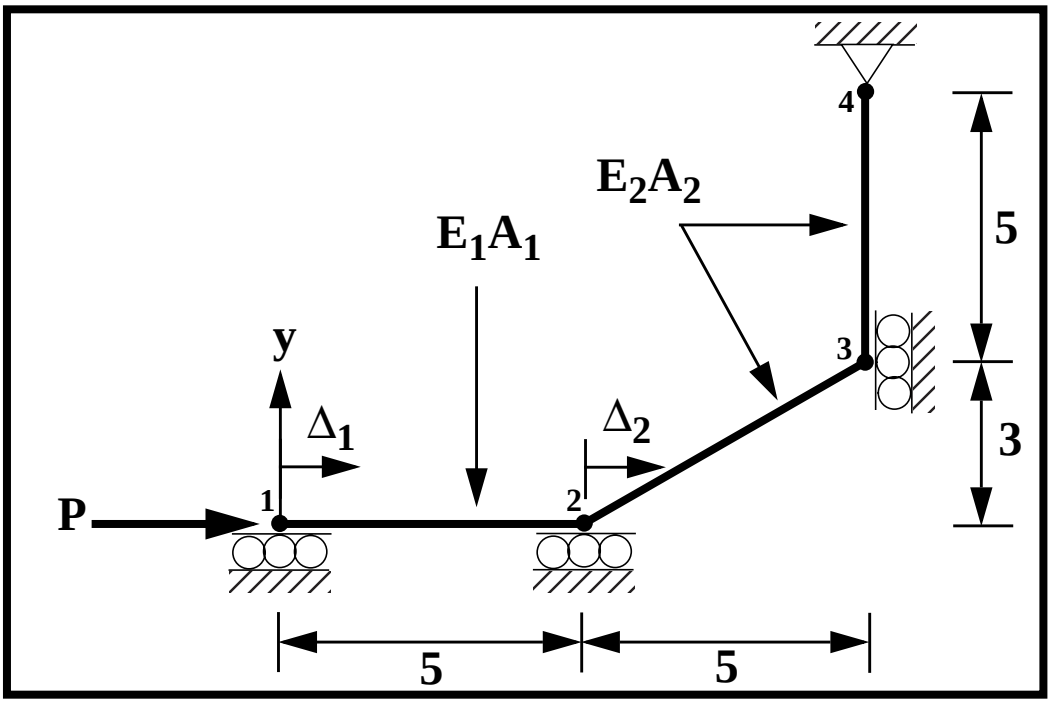


Table 3.1 - Properties

Elastic Modulus, E_1:	**1.0E6 psi**
Elastic Modulus, E_2:	**3.0E6 psi**
Bar Cross Sectional Area, A_1:	**0.5 in^2**
Bar Cross Sectional Area, A_2:	**1.0 in^2**
Load:	**4.0E5 lbs**

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: **1.0E6**

OK

Create a material called **mat_2**.

Model/Material...

Title: **mat_2**

Youngs Modulus: **3.0E6**

OK

Cancel

3. Create the properties that will define the two different rod elements.

Model/Property...

Elem/Property Type...

Line Elements: ● **Rod**

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**

Area: **0.5**

OK

Now create the second rod element property.

Title: **prop_2**

Material: **2..mat_2**

Area: **1.0**

OK

Cancel

4. Create the nodes for the structure.

Model/Node...

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

OK

Repeat the process for the other 3 nodes.

X:	*Y:*	*Z:*	
5	**0**	**0**	**OK**
10	**3**	**0**	**OK**
10	**8**	**0**	**OK**

Cancel

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

View/Autoscale

5. Create the elements of the structure.

Model/Element...

Type...

Line Elements: ● **Rod**

OK

Property: **1..prop_1**

Nodes: **1** **2**

OK

NOTE: When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the rod elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order. (Note that the node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!)

Element 1 has been created. Continue creating the remaining elements connecting the following nodes:

Property: **2..prop_2**

Nodes: **2** **3**

OK

Property: **2..prop_2**

Nodes: **3** **4**

OK

Cancel

6. 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...

Select **Node 1 & Node 2**.

OK

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

☐ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Select **Node 3**.

OK

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

☒ **TX** ☐ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Select **Node 4**.

OK

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

☒ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Cancel

7. 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: **40**

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

OK

Next, create the load.

Model/Load/Nodal...

Select **Node 1**.

OK

Highlight **Force**.

FX ☒ **4.0E5**

OK

Cancel

8. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob3**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Load Deflection of a 3-Rod Structure**

OK

Under *Output Requests*, change the output to:

2..Print and PostProcess

Also deselect all the boxes except the following:

☒ **Displacement**

OK

Now manually enter in the relevant NLPCI parameter.

Type Input...

Current Line: **NLPCI, 1**

OK

OK

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

Yes

File name: **prob3**

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

When asked if it is "OK to Begin Reading File C:\TEMP\prob3.xdb," respond **Yes**.

Yes

9. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

Select All

OK

Deselect **All Vectors** and instead select **T1 Translation** from the pull down menu.

☐ **All Vectors**

2..T1 Translation

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. The answers are listed at the end of the exercise.

What is the T1 displacement of **Node 1** at load step = 1.0?

T1 Translation @ Node 1 = _____

What is the T1 displacement of **Node 2** at load step = 1.0?

T1 Translation @ Node 2 = _____

10. 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 Vectors/Deformation: **2..T1 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

NOTE: You may need to decrease the magnification of the model in order to see deformation of the model.

View/Magnify...

11. Create an animation of the snap-through deformation.

View/Select...

Deformed Style: ● **Animate**

OK

Go into the **Animation Control** menu to decrease the frame rate of the animation.

View/Advanced Post/Animation...

Decrease the frame rate of the animation by clicking on the **Slower** button.

Slower

OK

You may also step through the animation frame by frame by using the **Next >** button.

Next >

When you wish to stop the animation, go to:

View/Select...

Deformed Style: ● **None - Model Only**

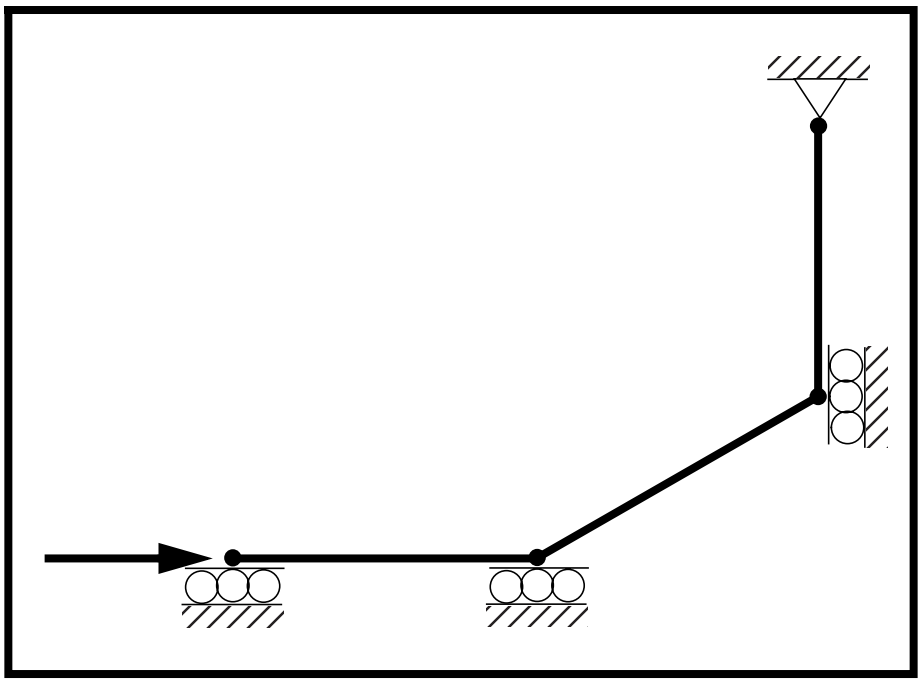
OK

This concludes the exercise.

Disp X @ Node 1:	**1.51092E1**
Disp X @ Node 2:	**1.11092E1**

WORKSHOP PROBLEM 3

Load Deflection of a 3-Rod Structure



Objectives:

- Create a model of a 3-rod structure.
- Apply the appropriate constraints and load.
- Submit an MSC/NASTRAN nonlinear analysis.
- Create an animation of the deformation of the structure.

Model Description:

Below in Figure 3.1 is a finite element representation of a 3-rod structure under a load. A nonlinear analysis with load increments will be performed to obtain the load-deflection behavior of the structure.

Figure 3.1

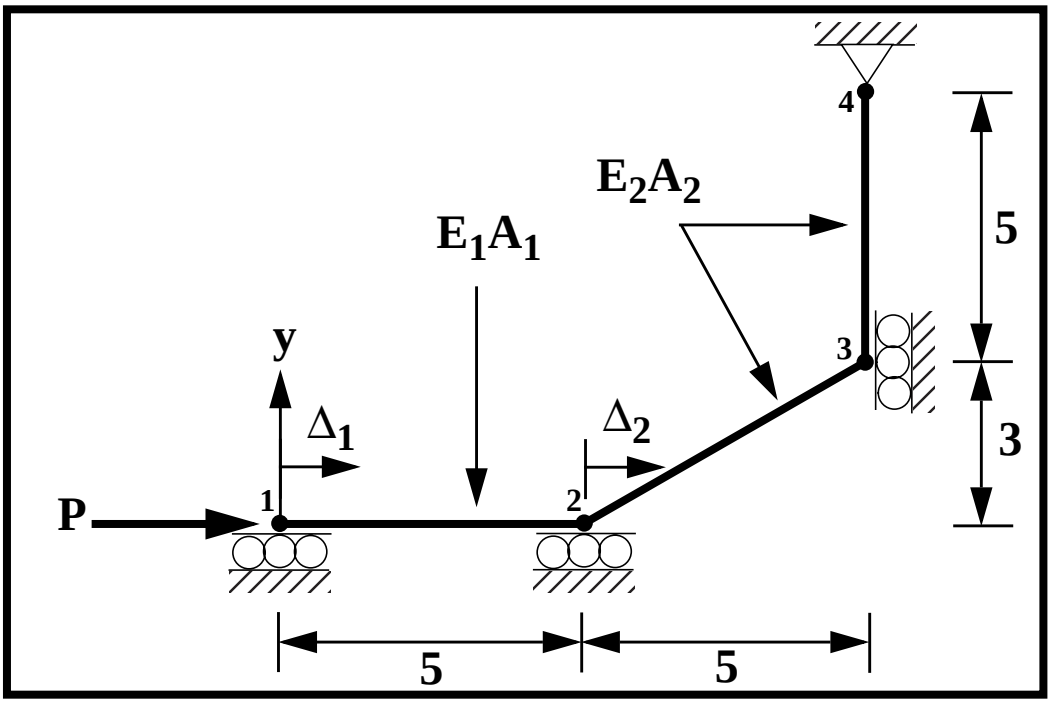


Table 3.1 - Properties

Elastic Modulus, E_1:	**1.0E6 psi**
Elastic Modulus, E_2:	**3.0E6 psi**
Bar Cross Sectional Area, A_1:	**0.5 in^2**
Bar Cross Sectional Area, A_2:	**1.0 in^2**
Load:	**4.0E5 lbs**

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: **1.0E6**

OK

Create a material called **mat_2**.

Model/Material...

Title: **mat_2**

Youngs Modulus: **3.0E6**

OK

Cancel

3. Create the properties that will define the two different rod elements.

Model/Property...

Elem/Property Type...

Line Elements: ● **Rod**

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**

Area: **0.5**

OK

Now create the second rod element property.

Title: **prop_2**

Material: **2..mat_2**

Area: **1.0**

OK

Cancel

4. Create the nodes for the structure.

Model/Node...

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

OK

Repeat the process for the other 3 nodes.

X:	*Y:*	*Z:*	
5	**0**	**0**	**OK**
10	**3**	**0**	**OK**
10	**8**	**0**	**OK**

Cancel

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

View/Autoscale

5. Create the elements of the structure.

Model/Element...

Type...

Line Elements: ● **Rod**

OK

Property: **1..prop_1**

Nodes: **1** **2**

OK

NOTE: When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the rod elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order. (Note that the node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!)

Element 1 has been created. Continue creating the remaining elements connecting the following nodes:

Property: **2..prop_2**

Nodes: **2** **3**

OK

Property: **2..prop_2**

Nodes: **3** **4**

OK

Cancel

6. 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...

Select **Node 1 & Node 2**.

OK

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

☐ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Select **Node 3**.

OK

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

☒ **TX** ☐ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Select **Node 4**.

OK

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

☒ **TX** ☒ **TY** ☐ **TZ**

☐ **RX** ☐ **RY** ☐ **RZ**

OK

Cancel

7. 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: **40**

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

OK

Next, create the load.

Model/Load/Nodal...

Select **Node 1**.

OK

Highlight **Force**.

FX ☒ **4.0E5**

OK

Cancel

8. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob3**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Load Deflection of a 3-Rod Structure**

OK

Under *Output Requests*, change the output to:

2..Print and PostProcess

Also deselect all the boxes except the following:

☒ **Displacement**

OK

Now manually enter in the relevant NLPCI parameter.

Type Input...

Current Line: **NLPCI, 1**

OK

OK

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

Yes

File name: **prob3**

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

When asked if it is "OK to Begin Reading File C:\TEMP\prob3.xdb," respond **Yes**.

Yes

9. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

Select All

OK

Deselect **All Vectors** and instead select **T1 Translation** from the pull down menu.

☐ **All Vectors**

2..T1 Translation

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. The answers are listed at the end of the exercise.

What is the T1 displacement of **Node 1** at load step = 1.0?

T1 Translation @ Node 1 = _____

What is the T1 displacement of **Node 2** at load step = 1.0?

T1 Translation @ Node 2 = _____

10. 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 Vectors/Deformation: **2..T1 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

NOTE: You may need to decrease the magnification of the model in order to see deformation of the model.

View/Magnify...

11. Create an animation of the snap-through deformation.

View/Select...

Deformed Style: ● **Animate**

OK

Go into the **Animation Control** menu to decrease the frame rate of the animation.

View/Advanced Post/Animation...

Decrease the frame rate of the animation by clicking on the **Slower** button.

Slower

OK

You may also step through the animation frame by frame by using the **Next >** button.

Next >

When you wish to stop the animation, go to:

View/Select...

Deformed Style: ● **None - Model Only**

OK

This concludes the exercise.

Disp X @ Node 1:	**1.51092E1**
Disp X @ Node 2:	**1.11092E1**