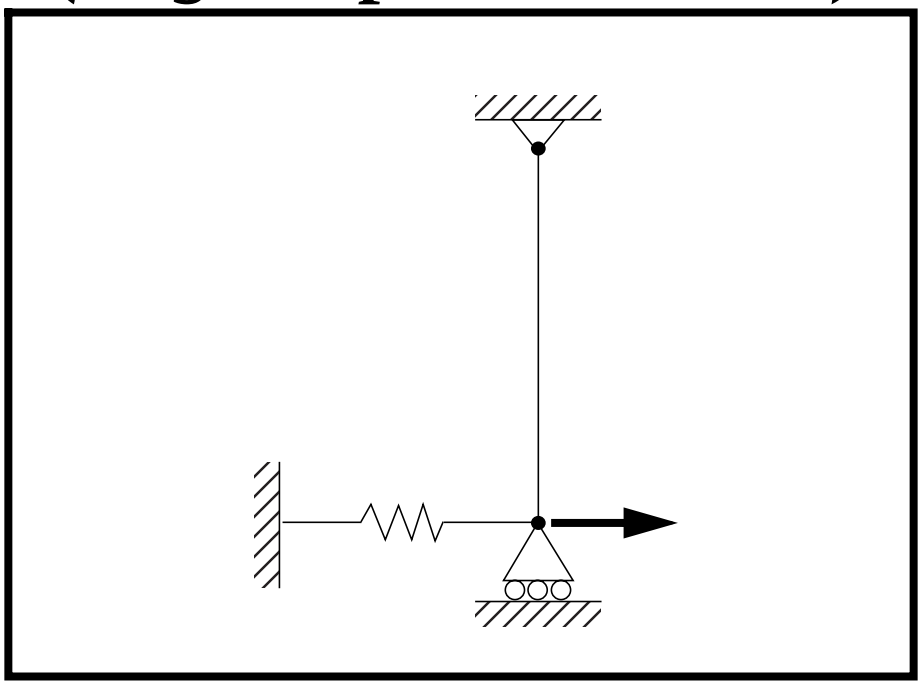
WORKSHOP PROBLEM 1b

Spring Element with Nonlinear Analysis Parameters (large displacements on)



Objectives:

- Import the model from the previous exercise.
- Submit an MSC/NASTRAN nonlinear analysis with large displacements option enabled.
- Review and compare the results with those computed in the previous exercise.

Model Description:

Below is a finite element representation of a rod connected to a grounded spring via a roller. The grounded spring will be modeled using a DOF spring element. A load is applied at the junction of these elements. A nonlinear analysis with the large displacements option enabled will be performed on the model.

Figure 1b.1

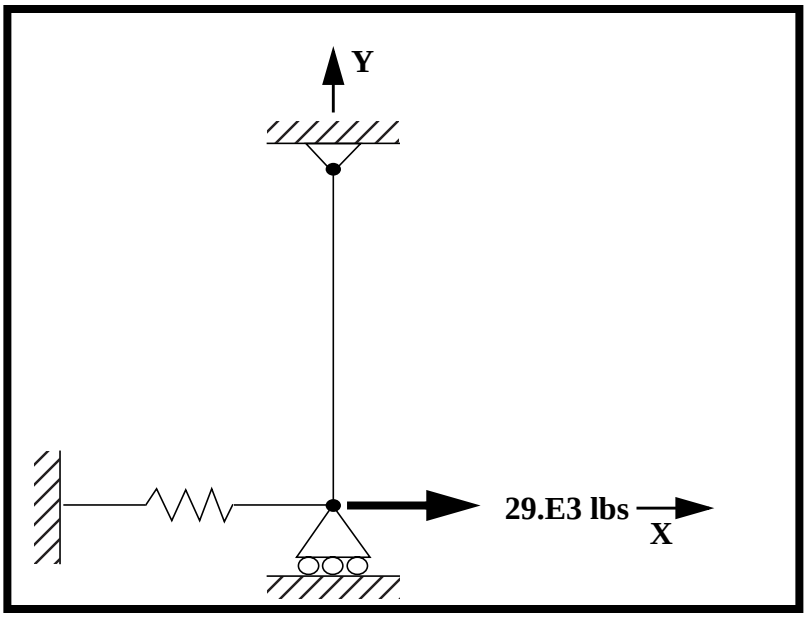


Table 1b.1 - Properties

Elastic Modulus:	**1.0E7 psi**
Length:	**10.0 in**
Bar Cross Sectional Area:	**0.01 in^2**
Spring Constant (K):	**1.0E3 lb/in**

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. Import the model created in Workshop Problem 1a.

File/Import/Analysis Model...

OK

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

File name: **prob1a**

Open

View/Redraw

3. Define the nonlinear analysis load set options.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Number of Increments: **4**

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

OK

4. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1b**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Spring Element Problem with Large Disp. on**

OK

Under *Output Requests*, change the output to:

2..Print and PostProcess

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Element Force**

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

OK

Click **OK** when you receive the confirmation that the subcase has been written.

OK

OK

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

Yes

File name: **prob1b**

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\prob1b.xdb," respond **Yes**.

Yes

5. 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 at the guided end, **Node 1**?

T1 Translation @ Node 1 = _____

You can make another list to find the answer to the second question.

List/Output/Unformatted...

Select All

OK

Deselect **All Vectors** and instead select **Spring Axial Force** from the pull down menu.

☐ **All Vectors**

3028..Spring Axial Force

OK

What is the force in the spring element?

Spring Axial Force = _____

Answer the remaining questions using similar procedure.

What is the force in the rod element?

Rod Axial Force = _____

6. 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**

☐ **Node - Perm Constraint**

Done

OK

Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Data Selection/Category: **1..Displacement**

Output Vectors/Deformation: **2..T1 Translation**

Output Vectors/Contour: **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**

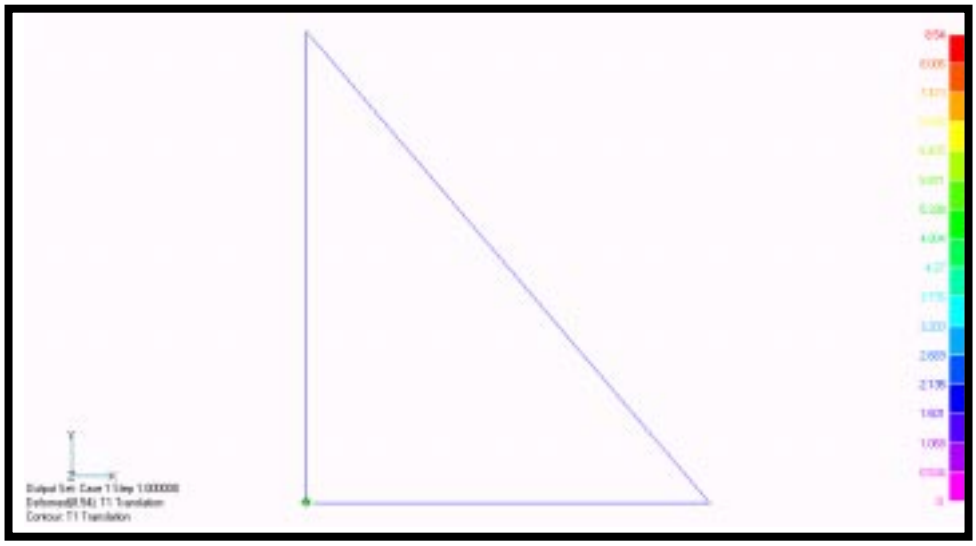
% of Model (Actual) ☐

OK

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

View/Magnify...

Figure 1b.2



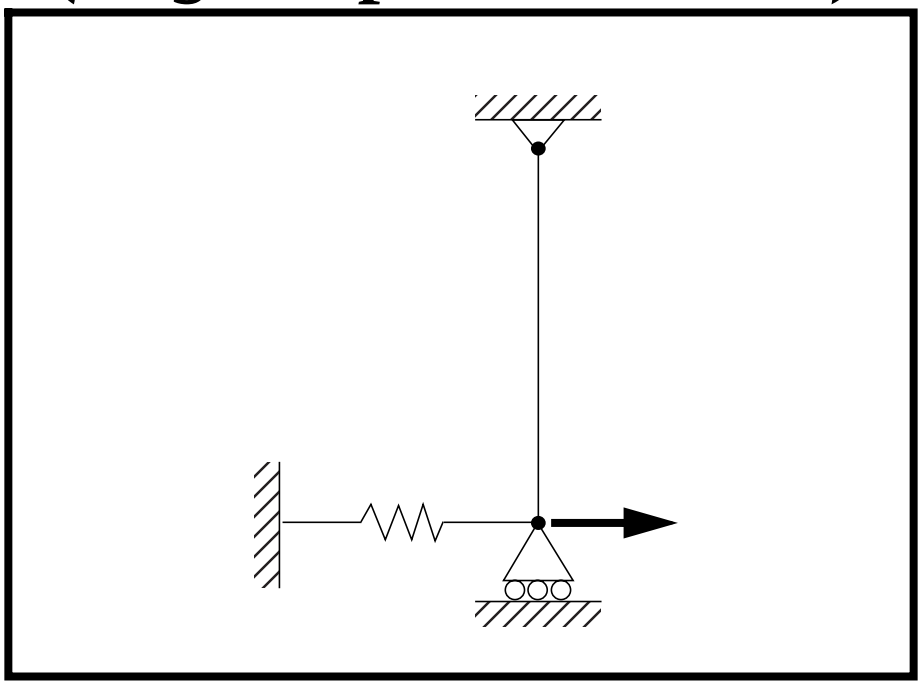
Notice the deflection computed in this exercise is much more reasonable than the result of the last exercise. The geometric nonlinearity of the problem has been better accounted for using the Large Displacements option.

This concludes the exercise.

Disp X:	**8.54019**
Spring Axial Force:	**8540.19**
Rod Axial Force:	**31504.7**

WORKSHOP PROBLEM 1b

Spring Element with Nonlinear Analysis Parameters (large displacements on)



Objectives:

- Import the model from the previous exercise.
- Submit an MSC/NASTRAN nonlinear analysis with large displacements option enabled.
- Review and compare the results with those computed in the previous exercise.

Model Description:

Below is a finite element representation of a rod connected to a grounded spring via a roller. The grounded spring will be modeled using a DOF spring element. A load is applied at the junction of these elements. A nonlinear analysis with the large displacements option enabled will be performed on the model.

Figure 1b.1

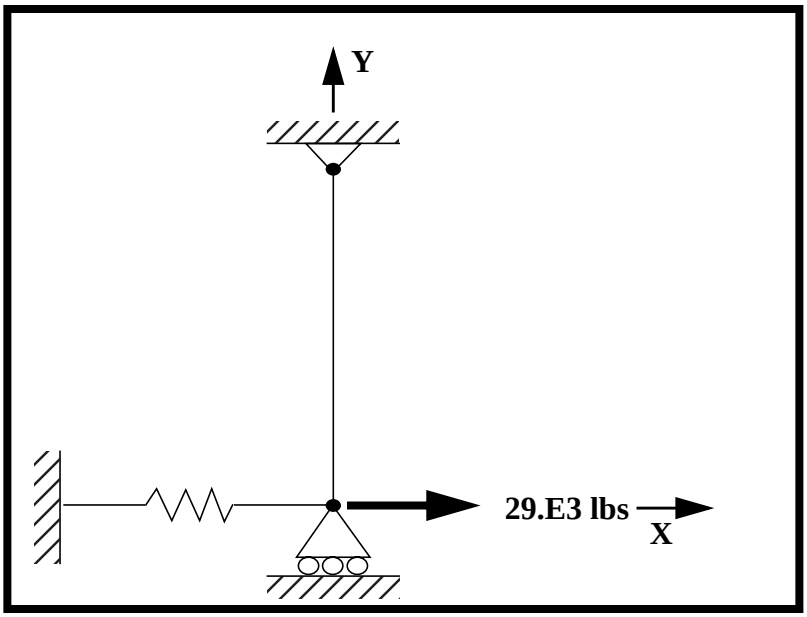


Table 1b.1 - Properties

Elastic Modulus:	**1.0E7 psi**
Length:	**10.0 in**
Bar Cross Sectional Area:	**0.01 in^2**
Spring Constant (K):	**1.0E3 lb/in**

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. Import the model created in Workshop Problem 1a.

File/Import/Analysis Model...

OK

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

File name: **prob1a**

Open

View/Redraw

3. Define the nonlinear analysis load set options.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Number of Increments: **4**

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

OK

4. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1b**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Spring Element Problem with Large Disp. on**

OK

Under *Output Requests*, change the output to:

2..Print and PostProcess

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Element Force**

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

OK

Click **OK** when you receive the confirmation that the subcase has been written.

OK

OK

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

Yes

File name: **prob1b**

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\prob1b.xdb," respond **Yes**.

Yes

5. 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 at the guided end, **Node 1**?

T1 Translation @ Node 1 = _____

You can make another list to find the answer to the second question.

List/Output/Unformatted...

Select All

OK

Deselect **All Vectors** and instead select **Spring Axial Force** from the pull down menu.

☐ **All Vectors**

3028..Spring Axial Force

OK

What is the force in the spring element?

Spring Axial Force = _____

Answer the remaining questions using similar procedure.

What is the force in the rod element?

Rod Axial Force = _____

6. 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**

☐ **Node - Perm Constraint**

Done

OK

Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Data Selection/Category: **1..Displacement**

Output Vectors/Deformation: **2..T1 Translation**

Output Vectors/Contour: **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**

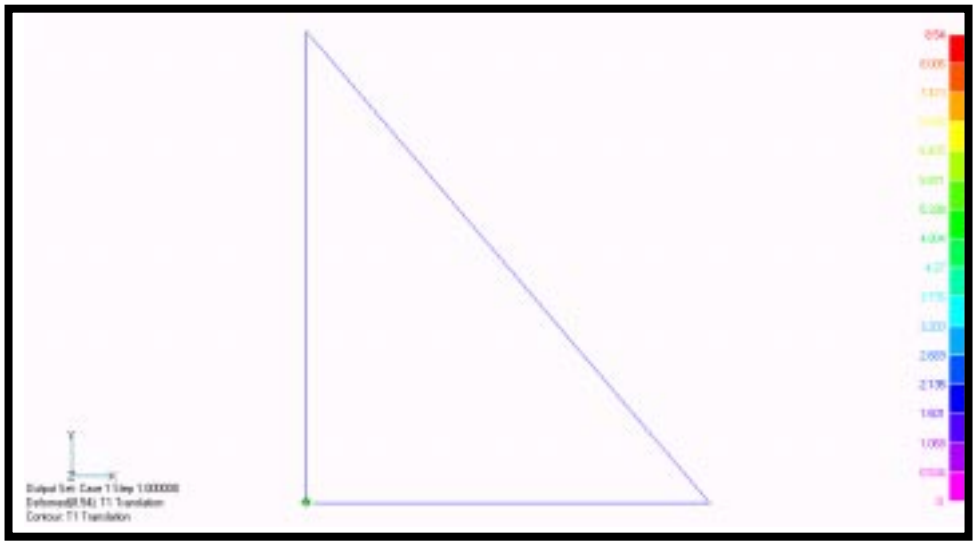
% of Model (Actual) ☐

OK

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

View/Magnify...

Figure 1b.2



Notice the deflection computed in this exercise is much more reasonable than the result of the last exercise. The geometric nonlinearity of the problem has been better accounted for using the Large Displacements option.

This concludes the exercise.

Disp X:	**8.54019**
Spring Axial Force:	**8540.19**
Rod Axial Force:	**31504.7**