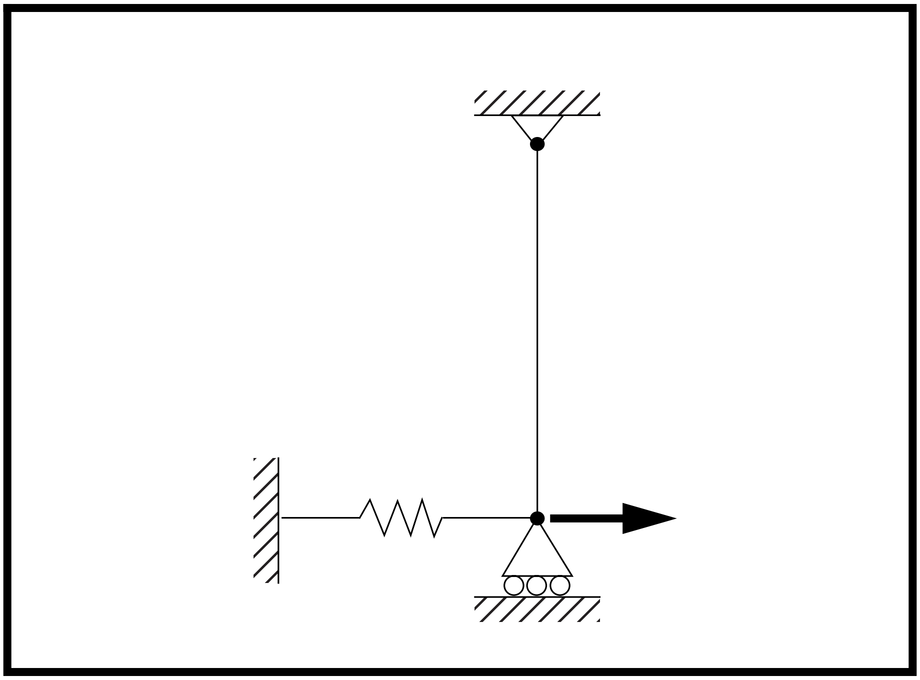
WORKSHOP PROBLEM 1a

Spring Element with Nonlinear Analysis Parameters (large displacements off)



Objectives:

- Create a model of a simple rod and grounded spring system.
- Apply the appropriate constraints and load.
- Submit an MSC/NASTRAN nonlinear analysis.
- Review the results and observe the difference between linear and nonlinear behaviors.

Model Description:

Below in Figure 1a.1 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 disabled will be performed on the model.

Figure 1a.1

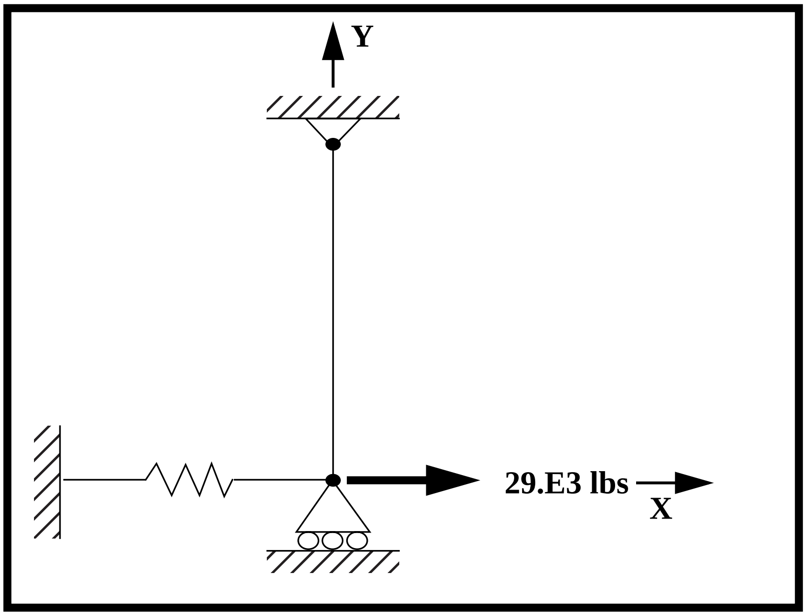


Table 1a.1 - Properties

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. Create a material called **mat_1**.

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

Model/Material...

Title: **mat_1**

Youngs Modulus: **1.0E7**

OK

Cancel

3. Create the properties that will define the rod element and the grounded spring.

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: **0.01**

OK

Now create the grounded spring property.

Elem/Property Type...

Line Elements: ● **DOF Spring**

OK

Title: **prop_2**

Tie the element's x translational freedom to the DOF of its end nodes.

End A: ● **TX**

End B: ● **TX**

Stiffness: **1.0E3**

OK

Cancel

4. Create the NASTRAN finite element model.

The model consists of two elements, the rod element and the 0-D grounded spring element at the lower node of the rod element.

First, create the rod element.

Mesh/Between...

Property: **1..prop_1**

Mesh Size/#Nodes/Dir1: **2**

OK

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

OK

	X:	*Y:*	*Z:*
Corner 2:	**0**	**10**	**0**

OK

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

View/Autoscale

View/Magnify...

Down 10%

OK

Now create the ground node for the 0-D spring element.

Model/Node...

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

Parameters...

Permanent Constraints: ☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

OK

Cancel

Create the grounded spring element.

Model/Element...

Type...

Line Elements: ● **DOF Spring**

OK

Property: **2..prop_2**

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

OK

Cancel

5. Create the model constraints.

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

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select **Node 2**, the upper node of the rod. It will be marked with a white circle, a +2 will be added to the *Entity Selection* box, and you will not be able to highlight the node anymore. This is a way of checking which node you have selected already.

OK

Fixed

OK

Select **Node 1**, the lower node of the rod.

OK

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

☐ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

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: **4**

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

OK

Next, create the load.

Model/Load/Nodal...

Select **Node 1**.

OK

Highlight **Force**.

FX ☒ **29.E3**

OK

Cancel

7. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1a**

Write

☒ **Run Analysis**

Advanced...

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

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

To disable the large displacements option, deselect the **LGDISP** box under *PARAM*.

☐ **LGDISP**

OK

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

Yes

File name: **prob1a**

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\prob1a.xdb,” respond **Yes**.

Yes

8. 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 = ____________

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**

☐ **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**

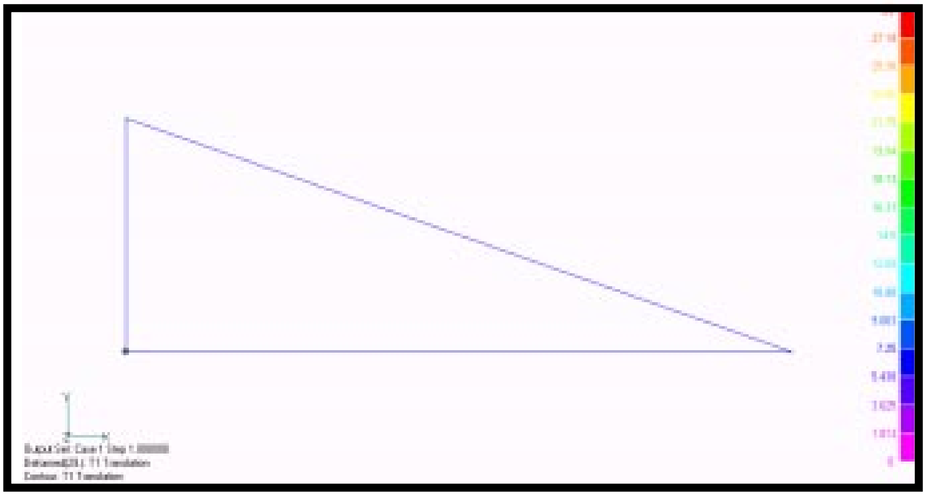
% of Model (Actual) ☐

OK

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

View/Magnify...

Figure 1a.2



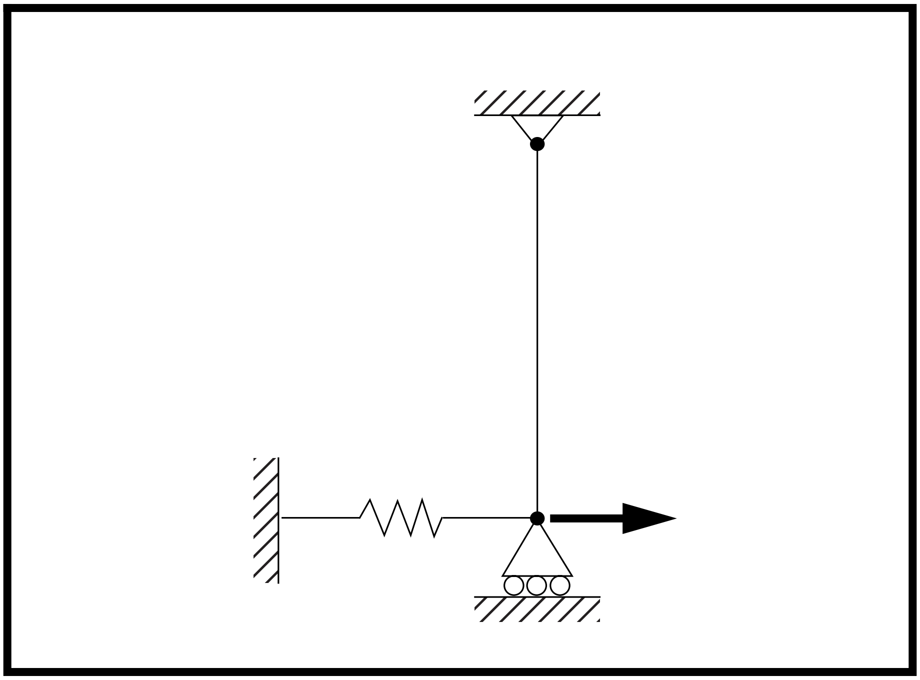
Notice that the deflection is almost three times the length of the beam! This suggests that a nonlinear analysis which accounts for large displacements is necessary to obtain a more accurate answer.

This concludes the exercise.

Disp X:	**29**
Spring Axial Force:	**29000**

WORKSHOP PROBLEM 1a

Spring Element with Nonlinear Analysis Parameters (large displacements off)



Objectives:

- Create a model of a simple rod and grounded spring system.
- Apply the appropriate constraints and load.
- Submit an MSC/NASTRAN nonlinear analysis.
- Review the results and observe the difference between linear and nonlinear behaviors.

Model Description:

Below in Figure 1a.1 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 disabled will be performed on the model.

Figure 1a.1

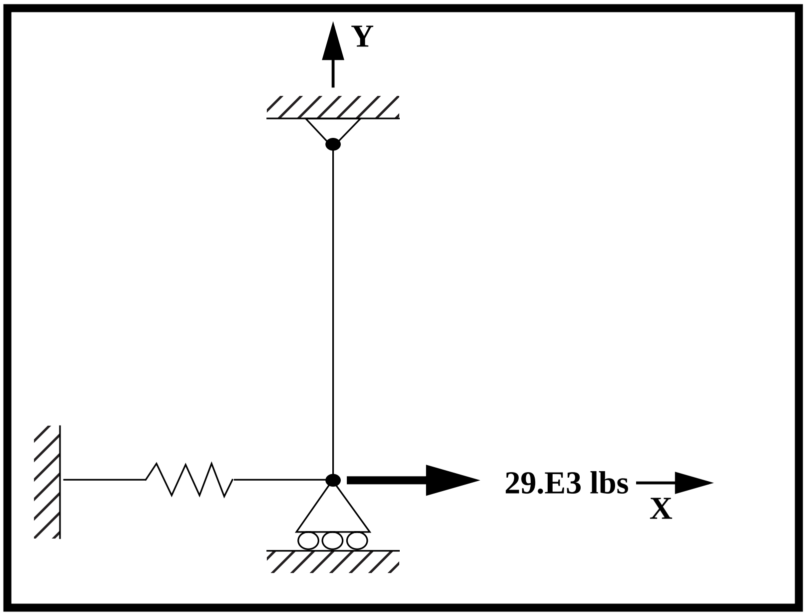


Table 1a.1 - Properties

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. Create a material called **mat_1**.

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

Model/Material...

Title:	**mat_1**
Youngs Modulus:	**1.0E7**

OK

Cancel

3. Create the properties that will define the rod element and the grounded spring.

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: **0.01**

OK

Now create the grounded spring property.

Elem/Property Type...

Line Elements: ● **DOF Spring**

OK

Title: **prop_2**

Tie the element's x translational freedom to the DOF of its end nodes.

End A: ● **TX**

End B: ● **TX**

Stiffness: **1.0E3**

OK

Cancel

4. Create the NASTRAN finite element model.

The model consists of two elements, the rod element and the 0-D grounded spring element at the lower node of the rod element.

First, create the rod element.

Mesh/Between...

Property:	**1..prop_1**
Mesh Size/#Nodes/Dir1:	**2**

OK

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

OK

	X:	*Y:*	*Z:*
Corner 2:	**0**	**10**	**0**

OK

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

View/Autoscale

View/Magnify...

Down 10%

OK

Now create the ground node for the 0-D spring element.

Model/Node...

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

Parameters...

Permanent Constraints: ☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

OK

Cancel

Create the grounded spring element.

Model/Element...

Type...

Line Elements: ● **DOF Spring**

OK

Property: **2..prop_2**

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

OK

Cancel

5. Create the model constraints.

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

Model/Constraint/Set...

Title: **constraint_1**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select **Node 2**, the upper node of the rod. It will be marked with a white circle, a +2 will be added to the *Entity Selection* box, and you will not be able to highlight the node anymore. This is a way of checking which node you have selected already.

OK

Fixed

OK

Select **Node 1**, the lower node of the rod.

OK

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

☐ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

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: **4**

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

OK

Next, create the load.

Model/Load/Nodal...

Select **Node 1**.

OK

Highlight **Force**.

FX ☒ **29.E3**

OK

Cancel

7. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1a**

Write

☒ **Run Analysis**

Advanced...

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

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

To disable the large displacements option, deselect the **LGDISP** box under *PARAM*.

☐ **LGDISP**

OK

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

Yes

File name: **prob1a**

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\prob1a.xdb,” respond **Yes**.

Yes

8. 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 = ____________

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**

☐ **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**

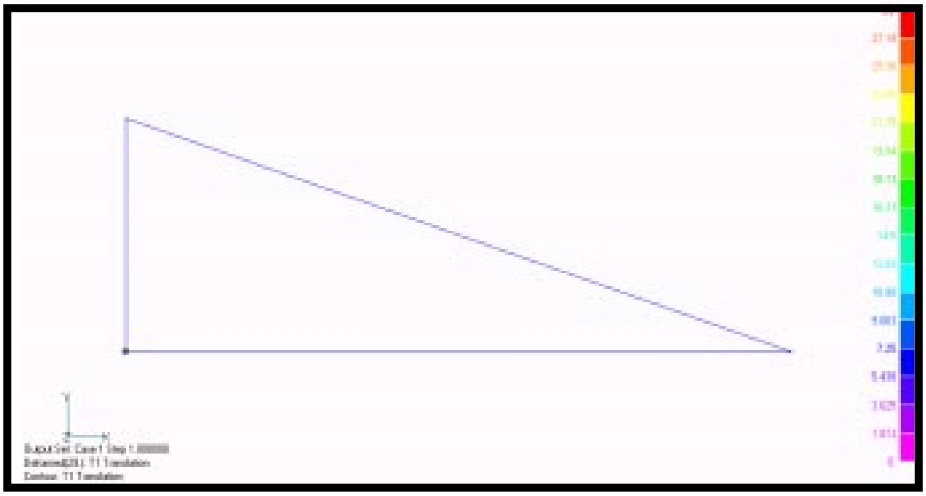
% of Model (Actual) ☐

OK

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

View/Magnify...

Figure 1a.2



Notice that the deflection is almost three times the length of the beam! This suggests that a nonlinear analysis which accounts for large displacements is necessary to obtain a more accurate answer.

This concludes the exercise.

Disp X:	**29**
Spring Axial Force:	**29000**