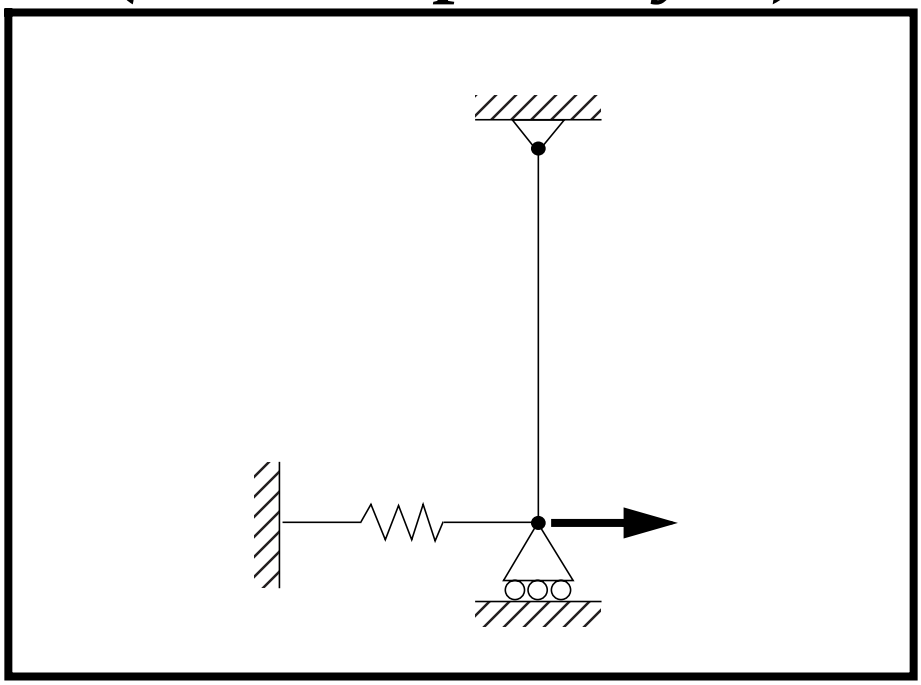
WORKSHOP PROBLEM 1c

Spring Element with Nonlinear Analysis Parameters (Multi-step Analysis)



Objectives:

- Import the model from the previous exercise.
- Apply incremental load through multiple subcases.
- Submit an MSC/NASTRAN nonlinear analysis.
- Review the multiple subcase results.

Model Description:

Below in Figure 1c.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. An incremental 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 1c.1

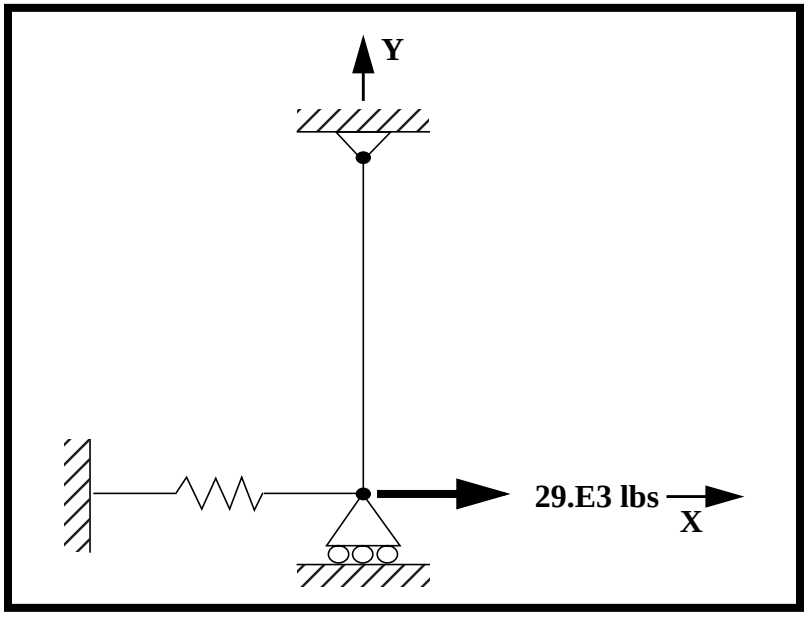


Table 1c.1 - Properties

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

Table 1c.2 - Load Cases

Subcase	Load	Load Increments	Note
1	16.E3 lbs	4	Do not use line search, quasi-Newton updates or bisection, and print output at every load step.
2	24.E3 lbs	8	Use the work criteria for convergence and print output at every load step.
3	29.E3 lbs	5	Request output at the end of the subcase.

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

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

View/Redraw

View/Autoscale

View/Magnify...

Down 10%

OK

3. Remove the model load set used in Workshop Problem 1a.

Delete/Model/Load-Set...

Select All

OK

When prompted “OK to Delete 1 Selected Load Set(s),” respond **Yes**.

Yes

4. Create the load set.

Model/Load/Set...

Title: **load_1**

OK

5. Define the nonlinear analysis parameters for the load set.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Number of Increments: **4**

Stiffness Updates/Method: **3..SEMI**

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

Convergence Tolerances: ☒ **Load**

OK

6. Next, apply the first step of the load history.

Model/Load/Nodal...

Select **Node 1**.

OK

Highlight **Force**.

FX ☒ **16.E3**

OK

Cancel

7. Repeat **Steps 4, 5, & 6** to create the remaining load steps. Use the following table to make the appropriate changes to the steps:

Load Set ID	**2**	**3**
Load Set Title	**load_2**	**load_3**
# of Increments	**8**	**5**
Stiffness Updates/Method	**1..AUTO**	**1..AUTO**
Output Control/Intermediate	**1..YES**	**2..NO**
Convergence Tolerances	☐ **Load**	☒ **Load**
FX @ Node 1	**24.E3**	**29.E3**

NOTE: Be certain to change the ID each time when creating a new load set!

After creating all the load sets, redraw the viewport by selecting:

View/Redraw

8. Submit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1c**

Write

☒ **Run Analysis**

Restarts...

Restart Control: ● **Save Databases for Restart**

OK

Advanced...

Problem ID: **Spring Element Problem, Multiple Load Cases**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Element Force**

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **1**

☒ **Loads =** **1..load_1**

Write Case...

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

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID: **2**

☒ **Loads =** **2..load_2**

Write Case...

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

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID:	**3**
☒ **Loads =**	**3..load_3**

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:	**prob1c**

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\prob1c.xdb," respond **Yes**.

Yes

9. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**4..Case 4 Step 1.000000**
Category:	**1..Displacement**

Entity: ● **Node**

ID: **1**

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

For each load set, what is the maximum T1 displacement at the guided end, **Node 1**?

Step 1 Max T1 @ Node 1 = _____

Step 2 Max T1 @ Node 1 = _____

Step 3 Max T1 @ Node 1 = _____

Save before existing the program.

File/Save

This concludes the exercise.

Spring Element with Nonlinear Parameters

Step 1 Disp X:	**6.30076**
Step 2 Disp X:	**7.75118**
Step 3 Disp X:	**8.54017**