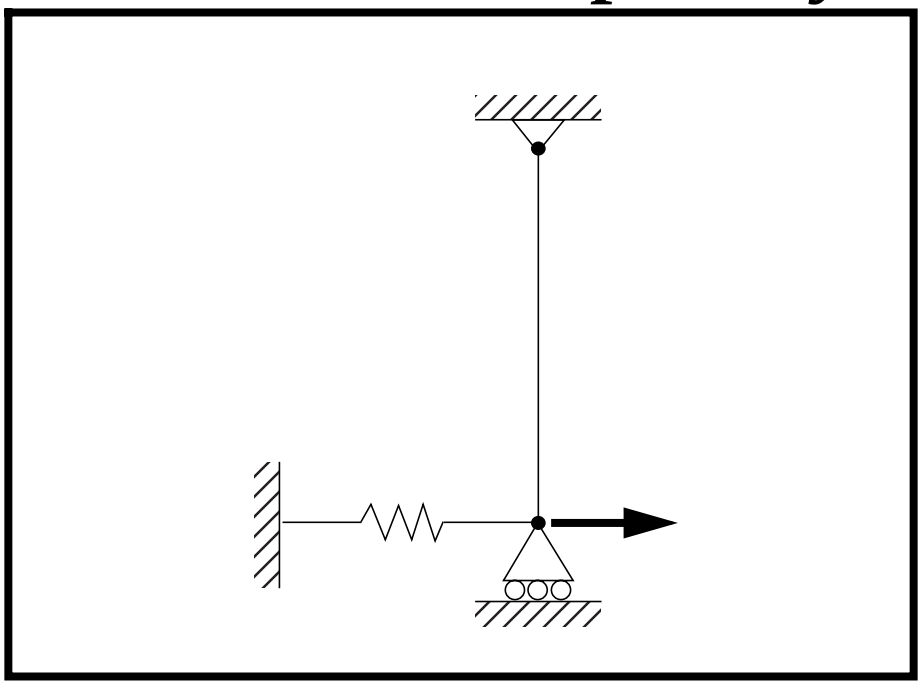
WORKSHOP PROBLEM 1d

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



Objectives:

- Demonstrate the use of the restart feature by introducing an intermediate load case and using the data obtained from the first part of the previous analysis.

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. 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 1d.1

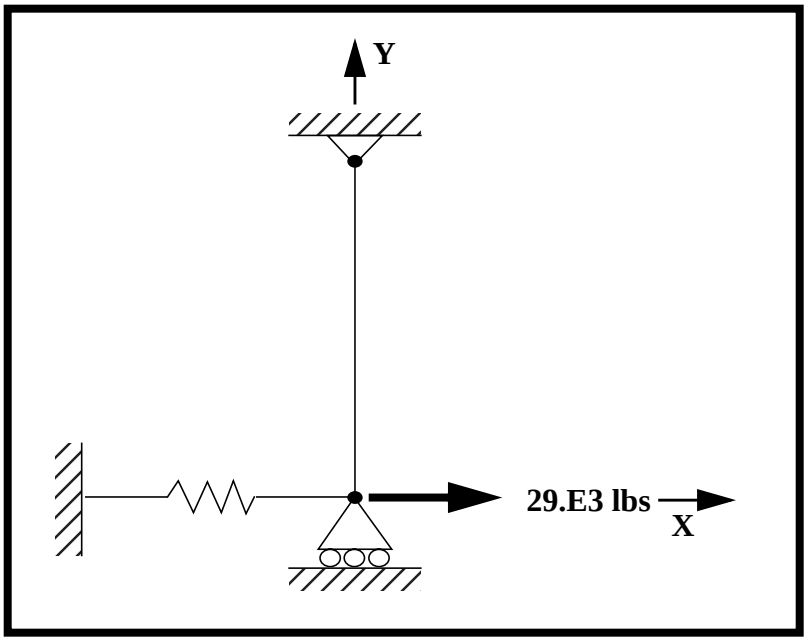


Table 1d.1 - Properties

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 1d.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 work criteria for convergence and print output at every load step.
3	24.E3 lbs	8	Request output at every load step.
4	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, change the directory to **C:\temp**.

Open Model File: **prob1c**

(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 the new load set.

Model/Load/Combine...

From Set: **2..load_2**

Last One

3. Modify the nonlinear analysis parameters for the load set.

Model/Load/Nonlinear Analysis...

Convergence Tolerances: ☒ **Load**

OK

4. Restart the analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob1d**

Write

☒ **Run Analysis**

Restarts...

Restart Control: ● **Restart Previous Analysis**

OK

On the *Restart From Database* form, change the directory to **C:\temp**.

File name: **prob1c.MASTER**

Open

Advanced...

Problem ID: **Spring Element Problem, Restart Multi Load Cases**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Element Force**

Type Input...

Current Line: **PARAM, LOOPID, 8**

More

Current Line: **PARAM, SUBID, 3**

OK

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 =** **4..Combined Set**

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:	**4**
☒ **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

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\prob1d.xdb," respond **Yes**.

Yes

5. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**17..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 1.5 Max T1 @ Node 1 = _____
Step 3 Max T1 @ Node 1 = _____
Step 4 Max T1 @ Node 1 = _____

6. Display the deformed plot and the fringe plot on the screen.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

Data Selection/Category: **0..Any Output**

Output Set: *(Sequentially select the result cases.)*

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

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

OK

OK

As you look at each result case, you will notice that the change in deflection lessens as more of the loading force is axially distributed. This is the benefit of running a nonlinear geometric analysis, which accounts for large displacements that change the distribution of the force along the beam.

This concludes the exercise.

Step 1 Disp X:	**6.30076**
Step 1.5 Disp X:	**7.06266**
Step 3 Disp X:	**7.75118**
Step 4 Disp X:	**8.54017**