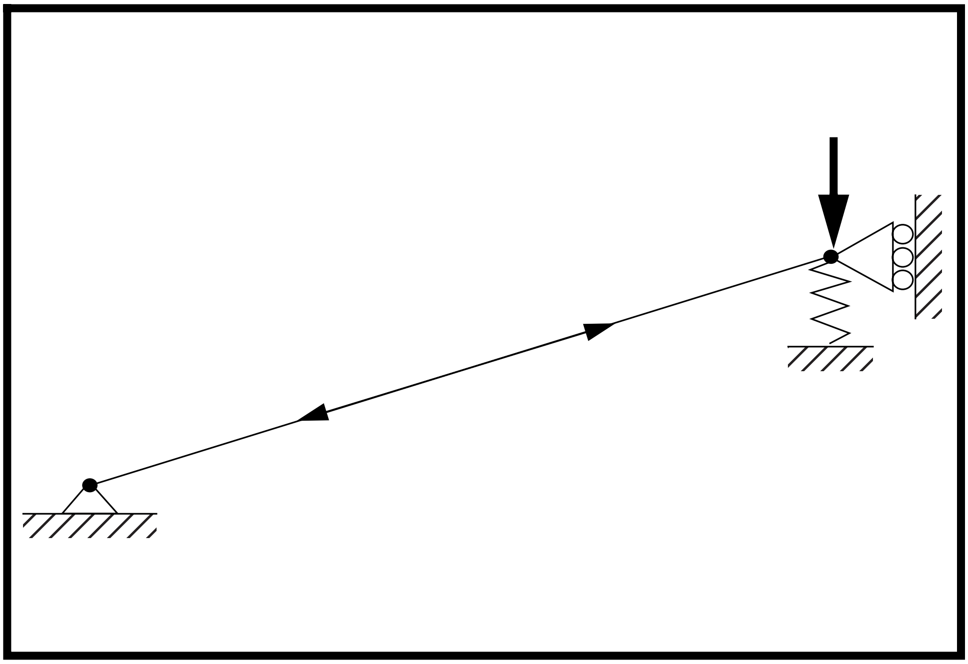
WORKSHOP PROBLEM 4c

Nonlinear Snap-Through Load Analysis (different spring constants)



Objectives:

- Create and prepare the appropriate model for the analysis.
- Demonstrate the use of a nonlinear static analysis for a snap-through load.
- Observe the effect of different spring constants on the load-deflection curve.

Model Description:

Below is Figure 4c.1 a finite element representation of a structure composed of a cantilever beam and a spring. A load will be applied at the junction of the beam and the spring. In this exercise, multiple nonlinear analyses will be performed on the model with different spring constants.

Figure 4c.1

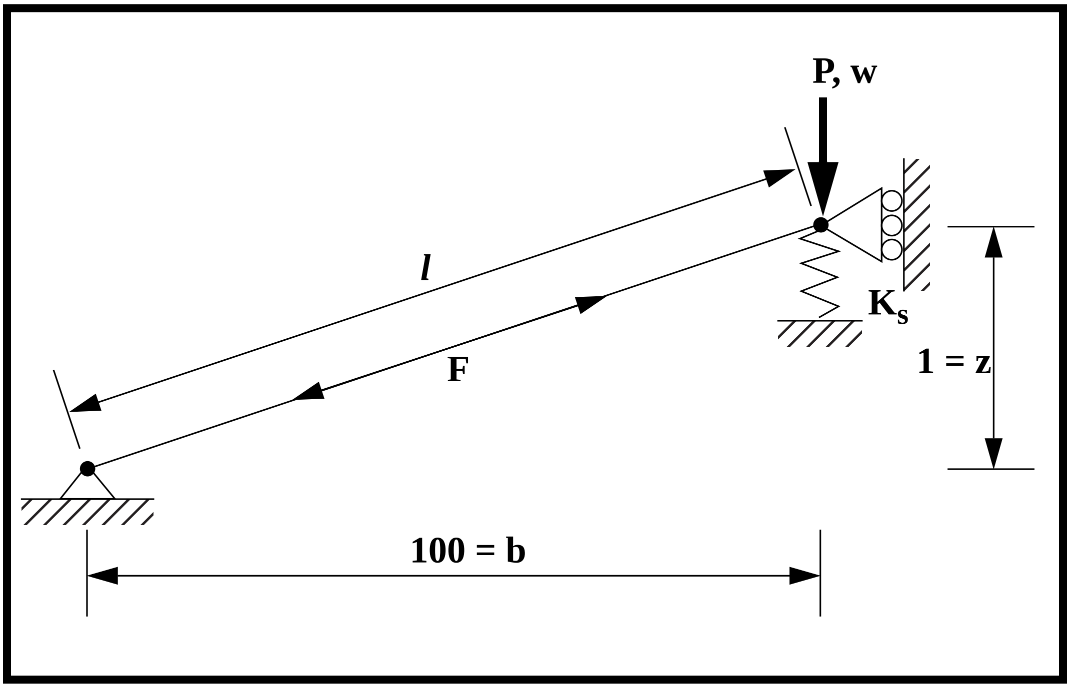


Table 4c.1 - Properties

Elastic Modulus:	**10.E7 psi**
Bar Cross Sectional Area:	**0.1 in^2**
Load, P:	**15 lbs.**
Spring Constant, K_s:	**0, 3, 6 lbs./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: **10.E7**

OK

Cancel

3. Create the property that will define the beam element and the spring element.

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: **0.1**

OK

Create the grounded spring property.

Elem/Property Type...

Line Elements: ● **DOF Spring**

OK

Title: **prop_2**

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

End A: ● **TY**

End B: ● **TY**

Stiffness: **0**

OK

Cancel

4. Create the NASTRAN finite element model.

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:	**100**	**1**	**0**

OK

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

View/Autoscale

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

Model/Node...

	X:	Y:	Z:
Coordinates:	**100**	**1**	**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: **2** **3**

OK

Cancel

5. 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**.

OK

Fixed

OK

Next, select **Node 2.**

OK

☒ **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:	**10**
Stiffness Updates/Method:	**1..AUTO**
Output Control/Intermediate:	**1..YES**

Next, define the parameters for the arc-length methods (NPLCI) .

Advanced...

Under *Arc-Length Solution Strategy*, enter the following:

Constraint Type:	**1..Crisfield**
Min ArcLen Adjust Ratio:	**1.0**
Max ArcLen Adjust Ratio:	**1.0**
Max Increments:	**25**

OK

OK

Now create the load.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force.**

FY ☒ **-15**

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: **prob4c_1**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear Snap-Through Analysis w/ Spring**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Applied Load**

OK

OK

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

Yes

File name: **prob4c**

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\prob4c_1.xdb," respond **Yes**.

Yes

8. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**10..Case 10 Step 1.000000**
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**2**

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 T2 displacement **Node 2**?

T2 displacement @ Node 2 = ______________

9. Modify the spring constant and resubmit the analysis. A sample of the algorithm for the next two analyses are as follows:

Change the spring constant to k=#.

Modify/Edit/Property...

ID: **2**

OK

Stiffness: # (3 or 6)

OK

Resubmit the job for analysis.

File/Export/Analysis Model...

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

OK

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

File name: **prob4c_#** (2 or 3)

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear Snap-Through Analysis w/ Spring**

OK

Under *Output Requests*, change the output to:

1..PostProcess Only

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Applied Load**

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\prob4c_#.xdb," respond **Yes** (where # is 2 or 3).

Yes

To list the results, select the following:

List/Output/Query...

Output Set:	**#0..Case 10 Step 1.000000** (# = 2 or 3)
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**2**

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 T2 displacement **Node 2**?

For k=3, T2 displacement @ Node 2 = ____________

For k=6, T2 displacement @ Node 2 = ____________

10. Create an XY plot of Displacement versus Load Step Value.

First, plot the k=0 case.

View/Select...

XY Style: ● **XY vs Set Value**

XY Data...

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

Output Set: **10..Case 10 Step 1.000000**

Output Vectors: **3..T2 Translation**

Output Location/Node: **2**

Show Output Sets:

From: **1**

To: **10**

OK

OK

Now, plot the k=3 case.

View/Select...

XY Style: ● **XY vs Set Value**

XY Data...

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

Output Set: **20..Case 10 Step 1.000000**

Output Vectors: **3..T2 Translation**

Output Location/Node: **2**

Show Output Sets:

From: **11**

To: **20**

OK

OK

Finally, plot the k=6 case.

View/Select...

XY Style: ● **XY vs Set Value**

XY Data...

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

Output Set: **30..Case 10 Step 1.000000**

Output Vectors: **3..T2 Translation**

Output Location/Node: **2**

Show Output Sets:

From: **21**

To: **30**

OK

OK

Notice that as k increases, the load required to produce “snap-through” also increases. “Snap-through” occurs when the displacement is greater that the maximum allowable value (y = -1.0).

This concludes the exercise.

Disp Y @ Node 2, k = 0:	**-2.67182**
Disp Y @ Node 2, k = 3:	**-2.43834**
Disp Y @ Node 2, k = 6:	**-2.16169**