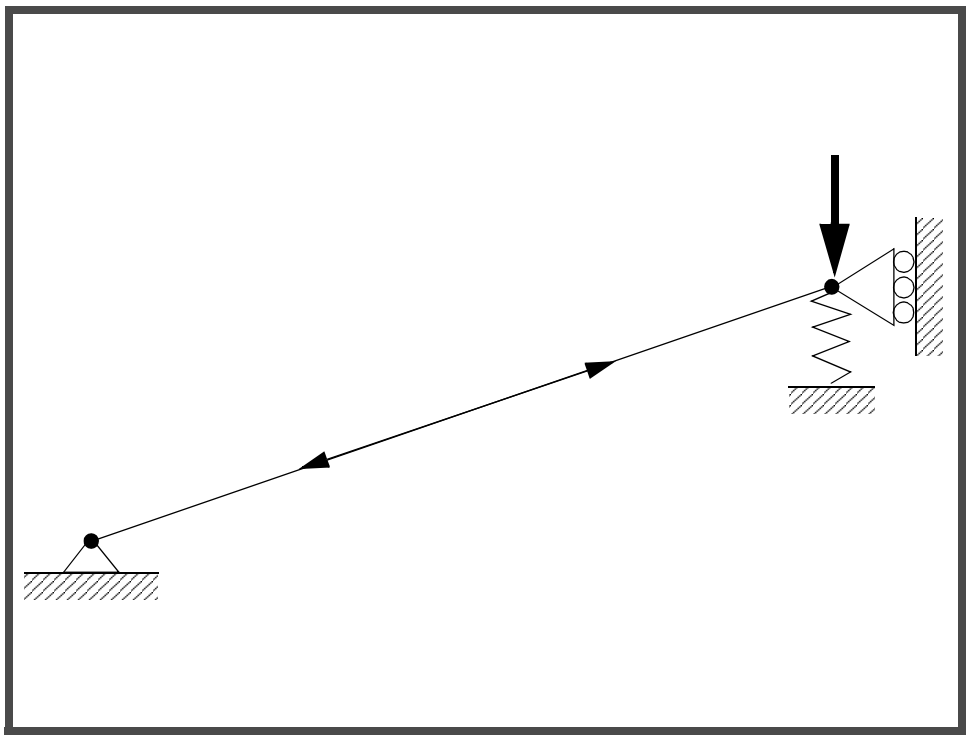
WORKSHOP 4b

Nonlinear Buckling Load Analysis (with spring)



Objectives:

- Create and prepare the appropriate model for the analysis.
- Demonstrate the use of a nonlinear static analysis with buckling parameters.

Model Description:

Below in Figure 4b.1 is 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, a nonlinear buckling analysis will be performed on the model. As an option, the analysis can be performed with different spring constants to see the effect.

Figure 4b.1

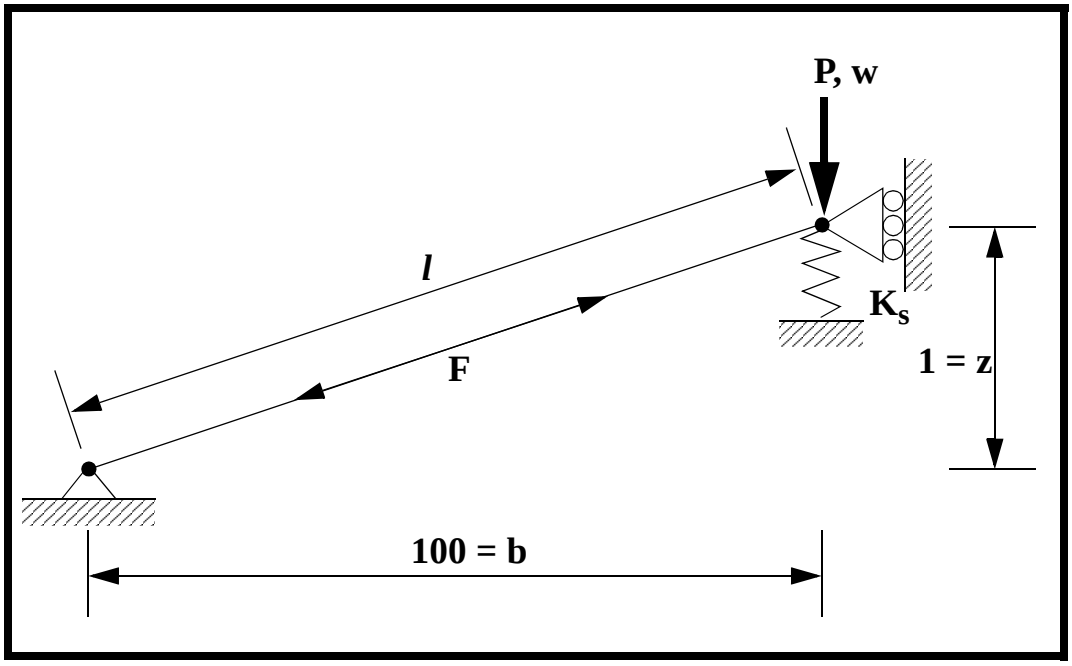


Table 4b.1 - Properties

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

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

On the *Open Model File* form, change the directory to **C:\temp**.

Open Model File: **prob4a**

2. Create the grounded spring property.

Model/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

3. Create the NASTRAN finite element model of the grounded spring.

First, 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

4. Define the nonlinear parameter for the model loading.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Number of Increments: **10**

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

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

OK

5. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob4b_1**

Write

☒ **Run Analysis**

Restarts...

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

OK

Advanced...

Problem ID: **Nonlinear Buckling Load Analysis w/ Spring**

OK

Under *Output Requests*, change the output to:

2..Print and PostProcess

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

6. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**10..Case 10 Time 1.**
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 displacements of **Node 2** at the end of the analysis?

T2 displacement @ Node 2 = _____

7. Plot the deformation of the beam.

View/Select...

Deformed Style:	● **Deform**
Contour Style:	● **Contour**

Deformed and Contour Data...

Data Selection/Category:	**1..Displacement**
Output Set:	**10..Case 10 Time 1.**
Output Vectors/Deformation:	**3..T2 Translation**
Output Vectors/Contour:	**3..T2 Translation**

OK

OK

8. Create a second load set to be used for the buckling analysis.

Model/Load/Set...

ID: **2**

Title: **load_2**

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: **70**

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

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

OK

Next, create the load.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force**.

FY ☒ **-6**

OK

Cancel

9. Resubmit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

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

File name: **prob4b_2**

Write

☒ **Run Analysis**

Restarts...

Restart Control: ● **Restart Previous Analysis**

OK

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

File name: **prob4b_1.MASTER**

Open

Advanced...

Problem ID: **Nonlinear Buckling Load Analysis w/ Spring 2**

OK

Under *Output Requests*, change the output to:

0..Print Only

Also deselect all the boxes except the following:

☒ **Displacement**

☒ **Applied Load**

Now manually enter in the parameter required for the buckling analysis.

Type Input...

Current Line: **PARAM, LOOPID, 3**

More

Current Line: **PARAM, SUBID, 2**

More

Current Line: **METHOD = 30**

More

Current Line: **PARAM, BUCKLE, 1**

OK

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**

OK

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

OK

Type Input...

Now enter the parameter for eigenvalues extraction.

Current Line: **EIGRL, 30, 0.0, 3.0, 20**

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

10. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**11..MSC/NASTRAN Case 2**
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 = _____

11. In the **prob4b_2.f06** files, search for the following key word for the results:

E I G E N V A L U E S (Spaces are necessary) :

What is the eigenvalue obtained from the analysis?

EIG = _____

What is the critical buckling load (Eigenvalue * applied load)?

Pcr = _____

12. Plot the deformation of the beam.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

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

Output Set: **24..MSC/NASTRAN Case 2**

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

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

OK

OK

If you wish, you may adjust the spring constant and repeat the previous exercise to see the effect.

This concludes the exercise.

Disp Y @ Node 2:	**-2.36976**
Disp Y @ Node 2 (run 2):	**-0.42118**
Eigenvalue:	**0.33394**
Critical Load:	**2.00364**