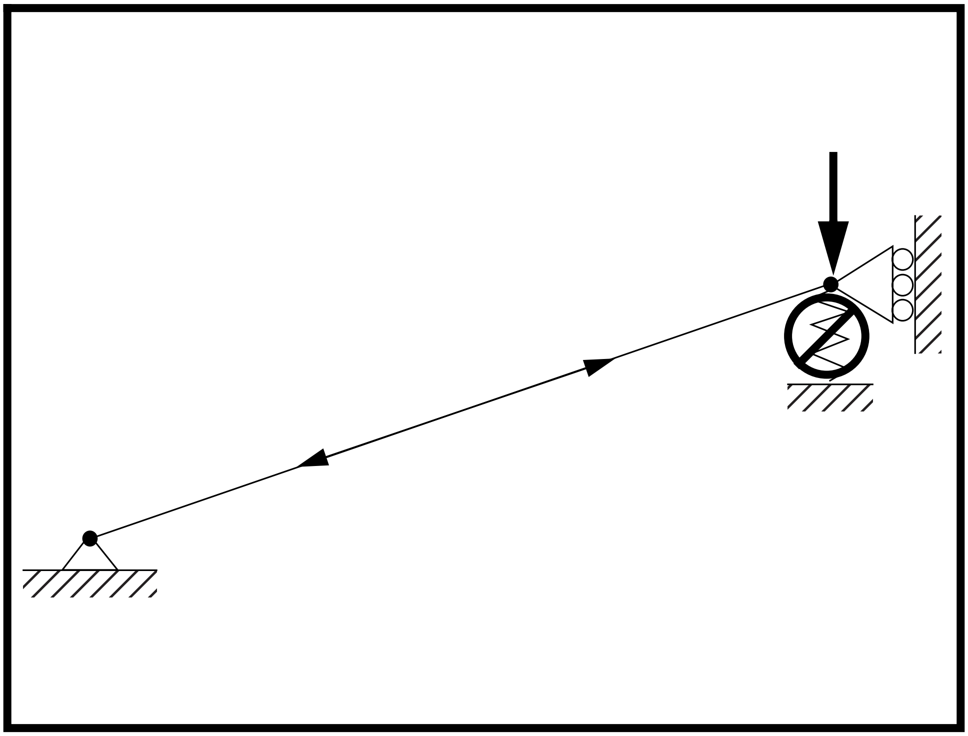
WORKSHOP PROBLEM 4a

Linear Buckling Load Analysis (without spring)



Objectives:

- Create and prepare the appropriate model for the analysis.
- Demonstrate the use of linear buckling analysis.

Model Description:

Below in Figure 4a.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 linear buckling analysis will be performed on the model without the spring element.

Figure 4a.1

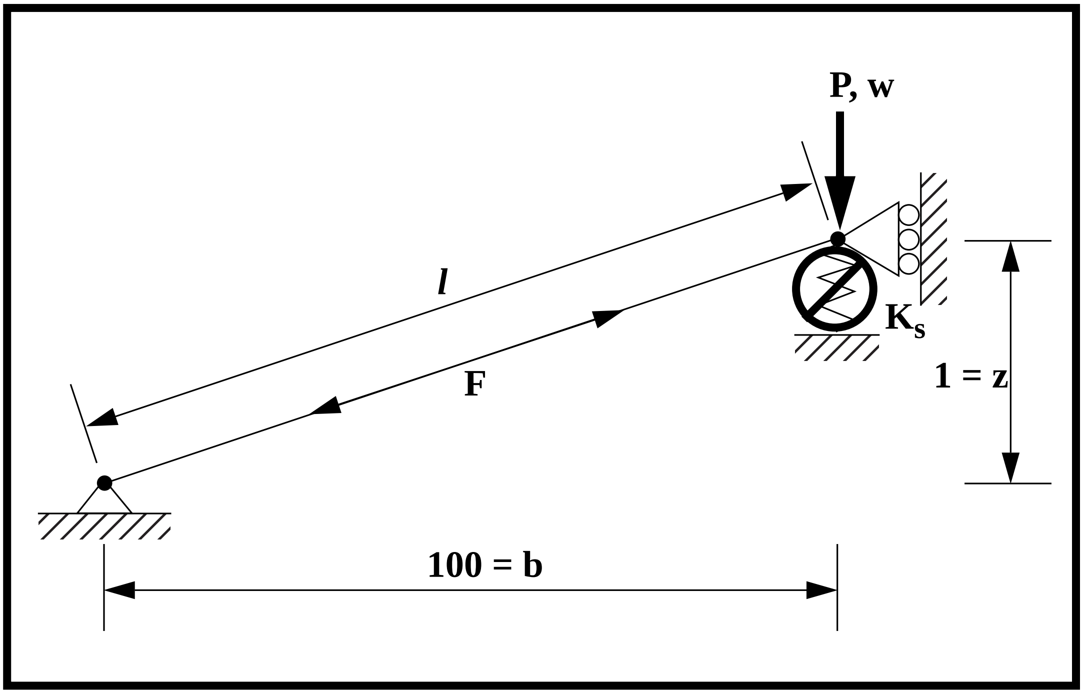


Table 4a.1 - Properties

Elastic Modulus:	**10.E7 psi**
Bar Cross Sectional Area:	**0.1 in^2**
Load, P:	**6 lbs.**

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**

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.

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

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

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 1**.

OK

Fixed

OK

Next, select **Node 2.**

OK

DOF: ☒ **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

Next, create the load.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force.**

FY ☒ **-6**

OK

Cancel

7. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **7..Buckling**

OK

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

File name: **prob4a**

Write

☒ **Run Analysis**

Advanced...

Define the buckling analysis parameters.

Method ID: **1**

Model Solution Method: ● **Inverse Power**

Under *Range of Interest*, enter the following:

From: **0**

To: **3**

Now enter the number of eigenvalues and eigenvectors to be estimated under *Eigenvalues and Eigenvectors*.

Number Estimated: **20**

Number Desired: **2**

OK

Problem ID: **Linear Buckling Load Analysis w/o 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

File name: **prob4a**

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\prob4a.xdb," respond **Yes**.

Yes

8. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set: **1..MSC/NASTRAN Case 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 displacement **Node 2**?

T2 displacement @ Node 2 = ____________

You can use the query function to find the answer to the remaining questions.

List/Output/Query...

Output Set: **2..Eigenvalue**

What is the first eigenvalue obtained from the analysis?

EIG = ________________

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

P_{cr} = ________________

Click **Cancel** when you are done.

Cancel

9. Plot the deformation of the beam.

View/Options...

Quick Options...

Labels Off

Deselect the following:

☐ **Load - Force**

☐ **Constraint**

Done

OK

Create the plot.

View/Select...

Deformed Style: ● **Deform**

Contour Style: ● **Contour**

Deformed and Contour Data...

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

Output Set: **1..MSC/NASTRAN Case 1**

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

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

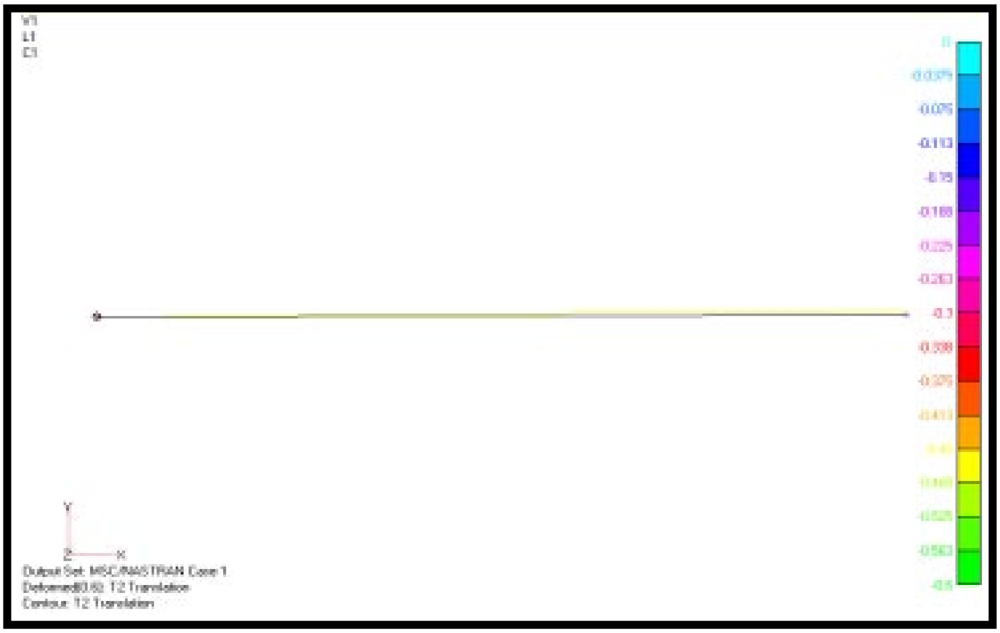
Options: **Deformed Style**

% of Model (Actual) ☐

OK

The XY View should appear as follows:

Figure 4a.2



As seen from the fringe values, the beam has a maximum downward deflection of 0.600. Since the load is less than the calculated critical load, the beam does not “snap-through” the maximum compression (deflection = 1).

This concludes the exercise.

Disp Y @ Node 2:	**-0.60009**
Eigenvalue:	**1.66658**
Critical Load:	**9.99948**