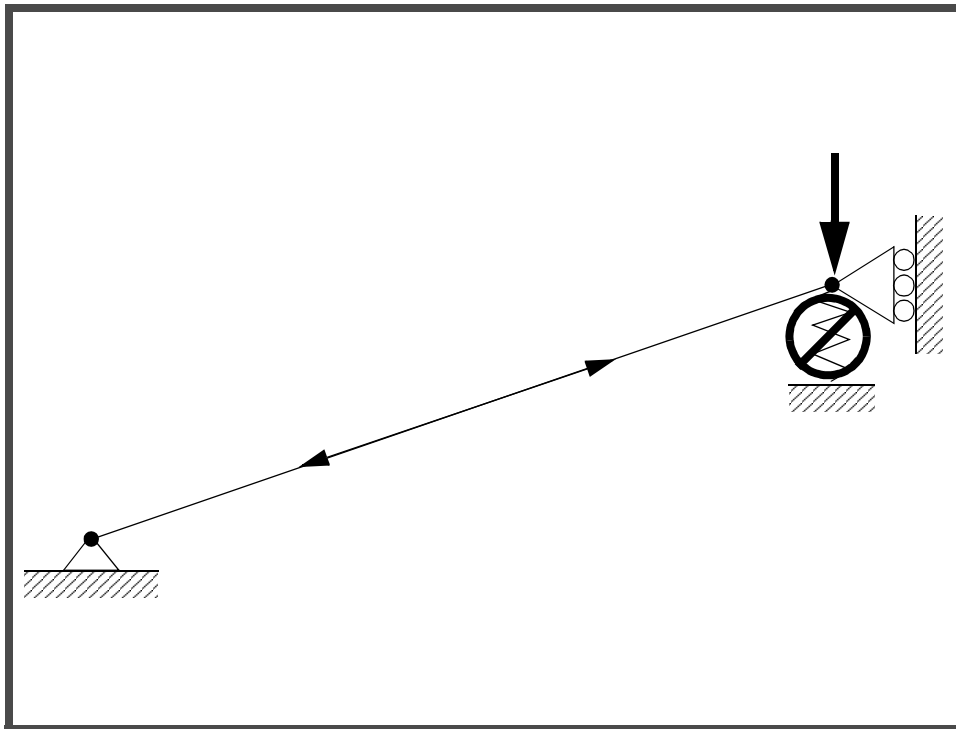


WORKSHOP 4a

Linear Buckling Load Analysis (without spring)

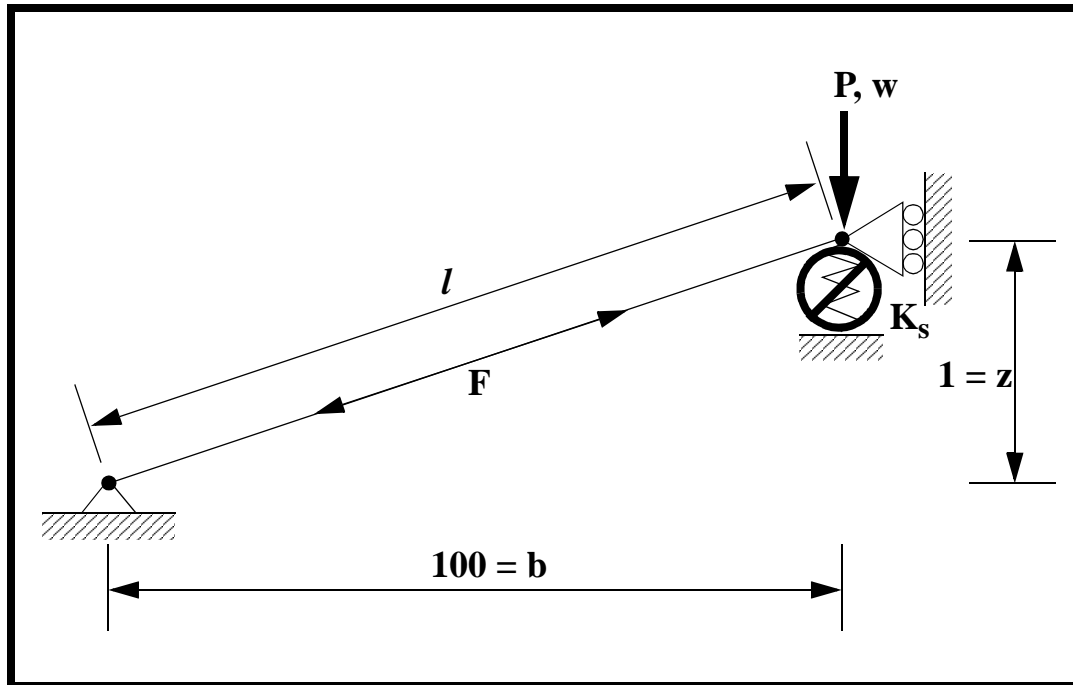


Objectives:

- 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 in²
Load, P:	6 lbs.

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, 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:

Mesh Size/#Nodes/Dir1:

X: Y: Z:

Corner 1: **0** **0** **0**

X: Y: Z:

Corner 2: **100** **1** **0**

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:

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select **Node 1**.

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...

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

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?

$$EIG = \underline{\hspace{2cm}}$$

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

$$P_{cr} = \underline{\hspace{2cm}}$$

Click **Cancel** when you are done.

Cancel

9. Plot the deformation of the beam.

View/Options...

Quick Options...
Labels Off

Deselect the following:

Draw: **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:

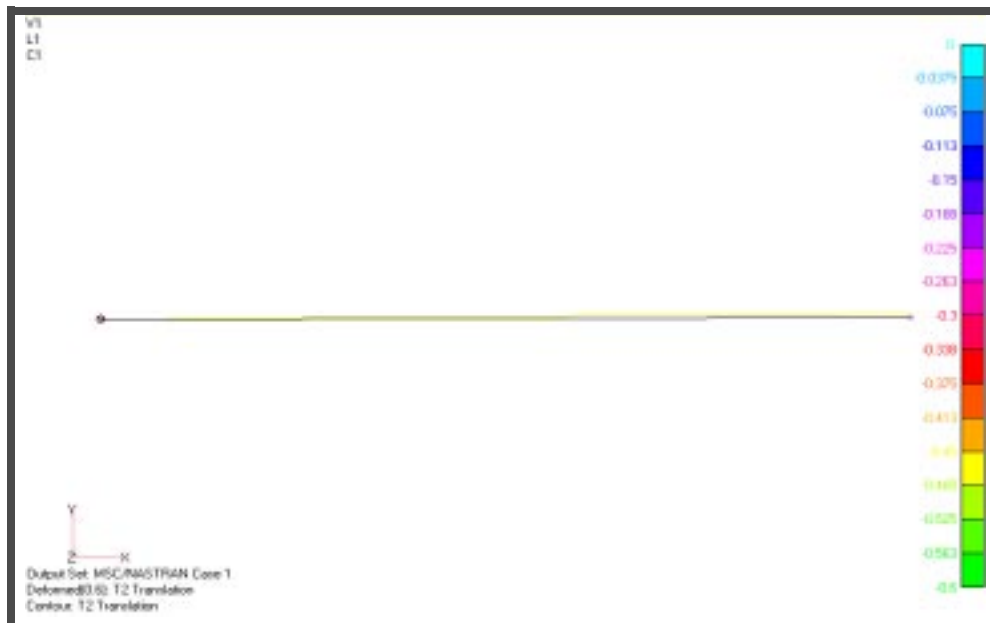
Deformed Style

% 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.

<i>Disp Y @ Node 2:</i>	-0.60009
<i>Eigenvalue:</i>	1.66658
<i>Critical Load:</i>	9.99948

