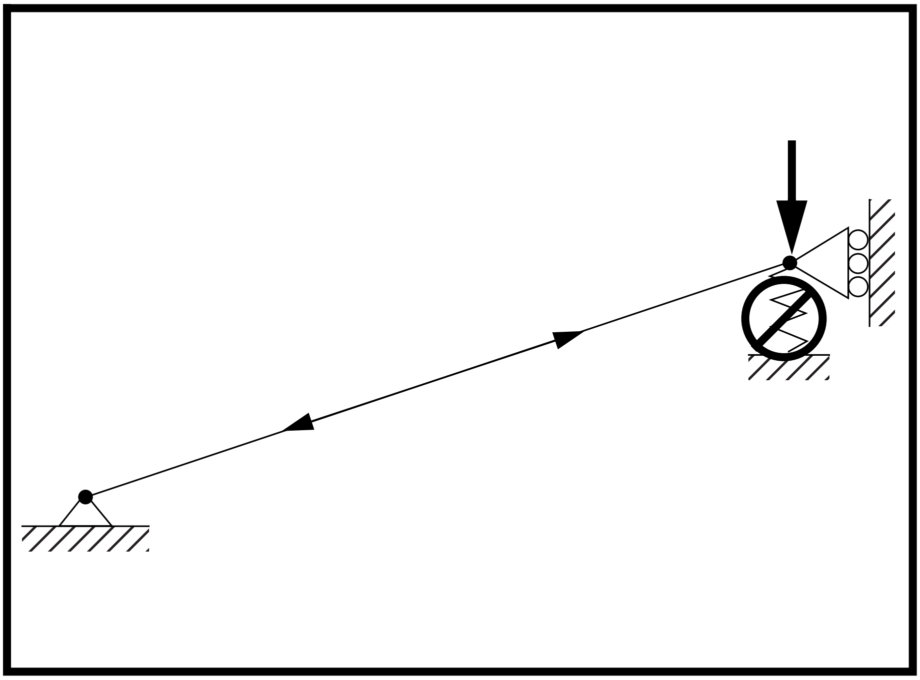
WORKSHOP PROBLEM 4a

Linear Buckling Load Analysis (without spring)

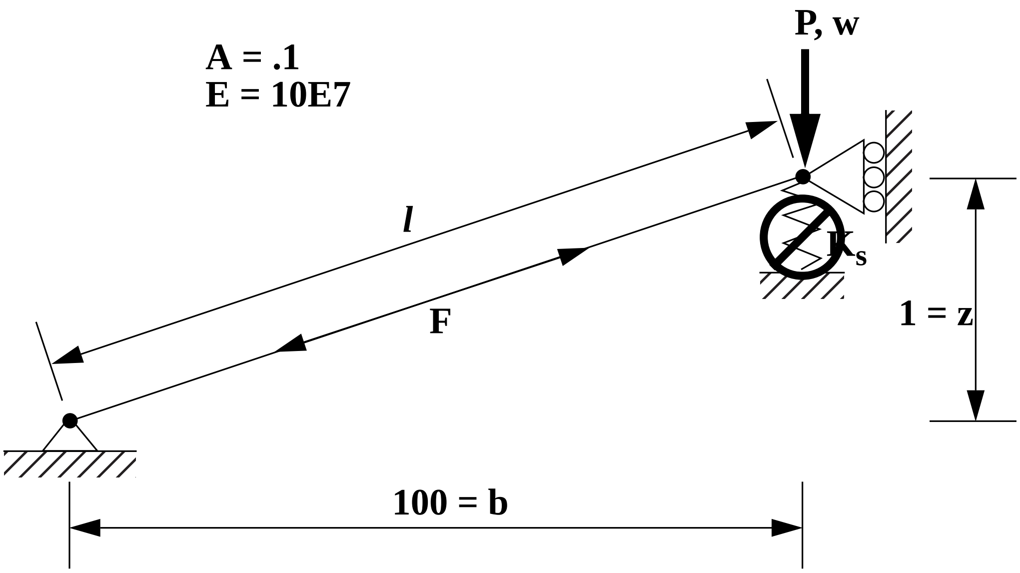


Objectives:

- Demonstrate the use of linear buckling analysis.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Calculate the buckling load without a spring.

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate loading conditions and buckling analysis control parameters.
- Prepare the model for a linear buckling analysis (SOL105).
- For Case Control, insert the static load set selection (LOAD) and buckling set (EIGB) in subcases.
- For Bulk Data, comment out all references to the spring element (CELAS), (PELAS), and insert all the appropriate buckling analysis parameters (EIGB).
- Generate an input file and submit it to the MSC/NASTRAN solver for a linear buckling analysis.
- Review the results.

Input File for Modification:

prob4a.dat

```
ID NAS103, WORKSHOP 4A
TIME 10
CEND
TITLE=SIMPLE ONE DOF GEOMETRIC NONLINEAR PROBLEM
LABEL=REF: STRICKLIN AND HAISLER; COMP. AND STRUCT.; 7:125-136 (1977)
ECHO=UNSORT
  DISP(SORT2)=ALL
BEGIN BULK
PARAM,POST,0
$
$ GEOMETRY
$
GRID, 1,  , 0., 0., 0.,  , 123456
GRID, 2,  , 100., 1., 0.,  , 13456
$
$ CONNECTIVITY
$
CROD, 10, 10, 1, 2
CELAS1, 20, 20, 2, 2, 0, 0
$
$ PROPERTIES
$
PROD, 10, 1, .1
PELAS, 20, 3.
MAT1, 1, 10.E7
$
$ LOADS
$
FORCE, 6, 2,  , -6., 0., 1., 0.
$
$ SOLUTION STRATEGY
$
ENDDATA
```

Exercise Procedure:

1. **Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 11, otherwise, proceed to step 3.**

2. Create a new database called **prob4a.db**.

File/New...

New Database Name:	**prob4a**
OK	

In the **New Model Preference** form set the following:

Tolerance:	● **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**
OK	

3. Those who do not wish to set up the model themselves may want to play the session file, **prob4a.ses**. If you choose to build the model yourself, proceed to the step 4.

File/Session/Play...

Session File List:	**prob4.ses**
Apply	

The model has now been created. Skip to **Step 10**.

Whenever possible click ❑ **Auto Execute** (turn off).

4. Create the long beam.

◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinate List:	**<100, 1, 0>**
Apply	

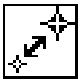
5. Mesh the curve with one BAR2 element

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**100**
Element Topology:	**Bar2**
Curve List:	**Curve 1** *(Select the curve.)*

Apply

For clarity, increase the node size using the following toolbar icon.

 **Node Size**

6. Create the material property for the beam.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**mat_1**

Input Properties...

Elastic Modulus =	**10.E7**

Apply

Cancel

7. Create the property for the beam.

◆ **Properties**

Action:	**Create**

Dimension:	**1D**
Type:	**Rod**
Property Set Name:	**beam**
Input Properties...	
Material Name	**m:mat_1**
Area	**0.1**
OK	
Select Members	**Curve 1** *(Select the curve.)*
Add	
Apply	

8. Create the LBCs for the model.

First, fix the left end of the beam.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name	**constraint_1**
Input Data...	
Translation < T1 T2 T3 >	**< 0, 0, 0 >**
Rotation < R1 R2 R3 >	**< 0, 0, 0 >**
OK	
Select Application Region...	
Select Geometry Entities	*(Select point at left of beam.)*
Add	
OK	
Apply	

Next, create the guided support LBC at the right end (free in y-direction, and fixed in all other DOFs).

New Set Name: **constraint_2**

Input Data...

Translation < T1 T2 T3 > **< 0, , 0 >**

Rotation < R1 R2 R3 > **< 0, 0, 0 >**

OK

Select Application Region...

Select Geometry Entities *(Select point at right of beam.)*

Add

OK

Apply

9. Create the loading for the model.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name **load_1**

Input Data...

Force <F1 F2 F3> **< 0, -6, 0>**

OK

Select Application Region...

Select Geometry Entities *(Select point at right of beam.)*

Add

OK

Apply

10. Now you are ready to generate an input file for analysis.

Click on the **Analysis** radio button on the Top Menu Bar and set up the subcases as follows:

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name	**prob4a**
Solution Type...	
Solution Type:	● **BUCKLING**
Solution Parameters...	
EigenValue Extraction ...	
Extraction Method:	**Inverse Power**
Lower =	**0.0**
Upper =	**3.0**
Estimated Number of Roots:	**20**
Number of Desired Positive Roots:	**2**
Number of Desired Negative Roots:	**2**
OK	
OK	
OK	
Subcase Create...	
Available Subcases:	**Default**
Output Requests...	
Form Type:	**Advanced**
Output Requests:	**SPCFORCES (SORT 1...**
Delete	
Output Requests:	**DISPLACEMENT(SORT1...**

Sorting: **By Freq/Time**

Modify

OK

Apply

Cancel

Apply

An input file called **prob4a.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN users should now proceed to **Step 12**.

Generating an input file for MSC/NASTRAN Users:

11. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob4a.dat**):

```
ASSIGN OUTPUT2 = ‘prob4a.op2’ , UNIT=12
ID NAS103, WORKSHOP 4A SOLUTION
TIME 10
SOL 105
CEND
TITLE=SIMPLE ONE DOF GEOMETRIC NONLINEAR PROBLEM
LABEL=REF: STRICKLIN AND HAISLER; COMP. AND STRUCT.; 7:125-136 (1977)
ECHO=UNSORT
 DISP(SORT2)=ALL
SUBCASE 10
  LOAD=6
SUBCASE 20
  METHOD=30
BEGIN BULK
PARAM,POST,0
$
$ GEOMETRY
$
GRID, 1,  , 0., 0., 0.,  , 123456
GRID, 2,  , 100., 1., 0.,  , 13456
$
$ CONNECTIVITY
CROD, 10, 10, 1, 2
$CELAS1, 20, 20, 2, 2, 0, 0
$
$ PROPERTIES
$
PROD, 10, 1, .1
$PELAS, 20, 3.
MAT1, 1, 10.E7
$
$ LOADS
$
FORCE, 6, 2,  , -6., 0., 1., 0.
$
$ SOLUTION STRATEGY
$
EIGB, 30, INV, 0.0, 3.0, 20, 2, 2,  , +EIGB
+EIGB, MAX
ENDDATA
```

Submit the input file for analysis:

12. Submit the input file to MSC/NASTRAN for analysis.

    12a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob4a.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

    12b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob4a.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

13. When the analysis is completed, edit the **prob4a.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

13a. While still editing **prob4a.f06**, search for the word:

R E A L E I G E N V A L U E S (spaces are necessary).

What is the first eigenvalue obtained from the analysis?

EIG = ____________________

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

P_{cr} = ____________________

While still editing **prob4a.f06**, search for the word:

D I S P L A C E M E N T S (spaces are necessary).

What is the y-displacement of Node 2?

T2 = ____________________

Comparison of Results:

14. Compare the results obtained in the **.f06** file with the results on the following page:

POINT-ID = 2

D I S P L A C E M E N T   V E C T O R

SUBCASE	TYPE	T1	T2	T3	R1	R2	R3
10	G	0.0	-6.000900E-01	0.0	0.0	0.0	0.0

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	1.666583E+00	1.290962E+00	2.054630E-01	5.999400E+00	9.998501E+00

15. This ends the exercise for MSC/NASTRAN users. MSC/PATRAN users should proceed to the next step.

16. Proceed with the Reverse Translation process, that is, importing the **prob4a.op2** results file into MSC/PATRAN. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File	**prob4a.op2**
OK	
Apply	

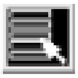
17. When the translation is complete bring up the **Results** form.

Now we will generate the fringe plot of the model.

◆ **Results**

Action:	**Create**
Object:	**Fringe**

Now click on the **Select Results** icon.

 **Select Results**

Select Result Case(s)	**Default, Static Subcase**
Select Fringe Result	**Displacements, Translational**
Quantity:	**Magnitude**

Next click on the **Target Entities** icon.

 **Target Entities**

Target Entity: **Current Viewport**

Note: This feature allows you to view fringe plots of specific elements of your choice.

Click on the **Display Attributes** icon.

 **Display Attributes**

Style: **Discrete/Smooth**

Display: **Free Edges**

For better visual quality of the fringe plot, change the width of the line.

Width: *(Select the third line from top.)*

Note: The **Display Attributes** form allows you the ability to change the displayed graphics of fringe plots.

Now click on the **Plot Options** icon.

 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

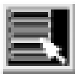
The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action: **Create**

Object: **Deformation**

Now click on the **Select Results** icon.

 **Select Results**

Select Result Case(s) **Default, Static Subcase**

Select Fringe Result **Displacements, Translational**

Show As: **Resultant**

Click on the **Display Attributes** icon.

 **Display Attributes**

Line Width: *(Select the third line from top.)*

In order to see the deformation results accurately, set the Scale Interpretation to True Scale with a Scale Factor of 1.

Scale Interpretation ● *True Scale*

Scale Factor **1.0**

■ *Show Undeformed*

Line Width: *(Select the third line from top.)*

Now click on the **Plot Options** icon .

 **Plot Options**

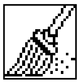
Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

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

To clear the post-processing results and obtain the original model in the viewport, select the **Reset Graphics** icon.



Reset Graphics

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob4a.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 14:12:46.
ASSIGN OUTPUT2 = ‘prob4.op2’, UNIT = 12
$ Direct Text Input for File Management Section
$ Buckling Analysis, Database
SOL 105
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 15-Jan-98 at 14:11:15
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT2,REAL)=ALL
SUBCASE 2
$ Subcase name : Default
  SUBTITLE=Default
  SPC = 2
  METHOD = 1
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC YES
PARAM   COUPMASS -1
PARAM    K6ROT   0.
PARAM    WTMASS  1.
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
EIGB     1       INV     0.      3.      20      2       2               +       A
+       A MAX
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : beam
PROD     1       1       .1
CROD     1       1       1       2
$ Referenced Material Records
```

```
$ Material Record : mat_1
$ Description of Material : Date: 11-Jun-97          Time: 11:15:21
MAT1    1       1.+8
$ Nodes of the Entire Model
GRID    1               0.      0.      0.
GRID    2               100.    1.      0.
$ Loads for Load Case : Default
SPCADD  2       1       3
LOAD    2       1.      1.      1
$ Displacement Constraints of Load Set : constraint_1
SPC1    1       123456  1
$ Displacement Constraints of Load Set : constraint_2
SPC1    3       13456   2
$ Nodal Forces of Load Set : load_1
FORCE   1       2       0       6.      0.      -1.     0.
$ Referenced Coordinate Frames
ENDDATA 87e89bd5
```