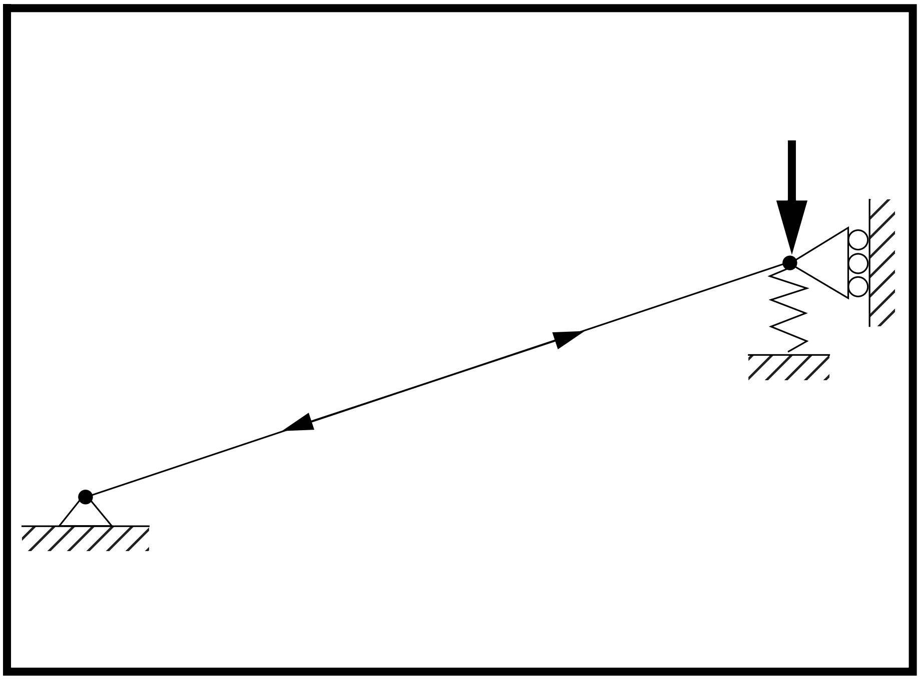
WORKSHOP PROBLEM 4c

Nonlinear Snap-Through Load Analysis (different spring constants)

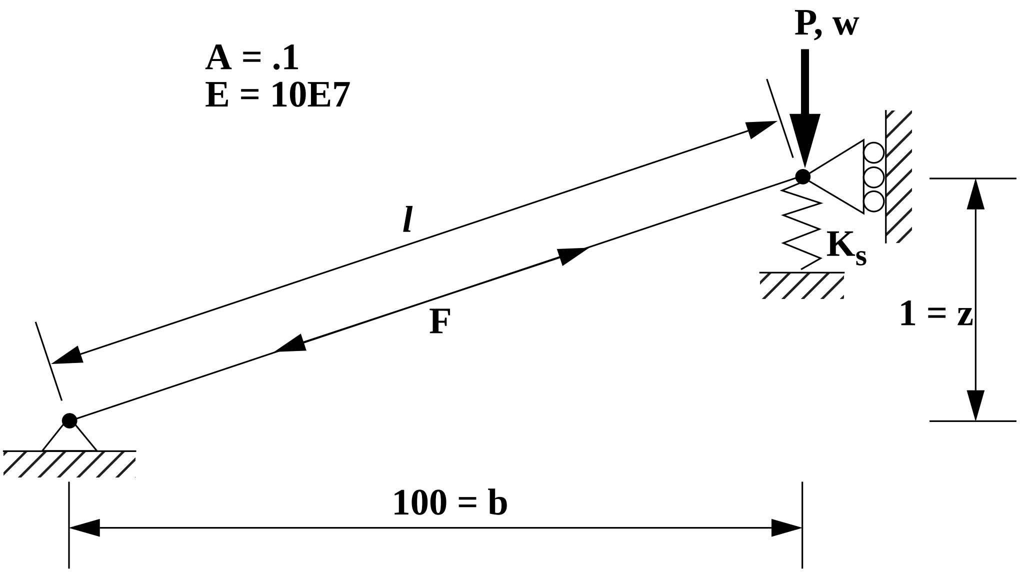


Objectives:

- Demonstrate the use of a nonlinear static analysis for a snap-through load.
- Demonstrate the effect of different spring constants on the load-deflection curve.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Perform a nonlinear static analysis on the model for the cases when K_s=0, K_s=3, and K_s=6.

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate loading conditions and nonlinear static analysis control parameters.
- For Case Control, insert the static load set selection (LOAD) and the nonlinear static analysis parameter selection (NLPARM) in each subcase.
- For Bulk Data, insert all the relevant nonlinear static analysis parameters for each subcase (NLPARM).
- Prepare the model for a nonlinear static analysis.
  - PARAM, LGDISP, 1
- Insert the parameters for arc-length methods (NLPCI).
- Generate an input file and submit it to the MSC/NASTRAN solver for a nonlinear static analysis.
- Review the results.
- Modify the existing model, adjusting the spring constant.
- Generate another input file and submit it to the MSC/NASTRAN solver for a normal modes analysis.
- Review the results.
- Modify the existing model, adjusting the spring constant.
- Generate the final input file and submit it to the MSC/NASTRAN solver for a normal modes analysis.
- Review the results.

Input File Workshop 4a for Modification:

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
```

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 7, otherwise, proceed to step 2.

2. Create a new database called **prob4c.db**.

File/New...

New Database Name:	**prob4c**

OK

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

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

OK

3. Import the model data from the database **prob4a.db**.

File/Import...

Object:	**Model**
Source:	**MSC/PATRAN DB**
PATRAN Databases:	**prob4a.db**

Apply

When the summary form appears, clear it by clicking on **OK**.

OK

4. Modify the model loading to 15 lbs.

◆ **Loads/BCs**

Action:	**Modify**
Object:	**Force**
Type:	**Nodal**

Select Set to Modify:	**load_1**
Modify Data...	
Force <F1 F2 F3 >	**< 0, -15, 0>**
OK	
Apply	

5. Make sure the spring constant is set to k=0.

◆ **Properties**

Action:	**Modify**
Dimension:	**0D**
Type:	**Grounded Spring**
Select Prop. Set to Modify	**spring**
Modify Properties...	
Spring Constant:	**0**
OK	
Apply	

6. Now you are ready to generate an input file for the 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	**prob4c_1**
Solution Type...	
Solution Type:	● **NONLINEAR STATIC**
OK	

Subcase Create...	
Subcase Name:	**nonlinear**
Output Requests...	
Form Type:	**Advanced**
Output Requests	*(Deselect all except* ***DISPL...****)*
Delete	
Select Result Type	**Applied Loads**
Sorting:	**By Freq/Time**
Create	
Output Requests	*(Select* ***DISPL...****)*
Sorting:	**By Freq/Time**
Modify	
Intermediate Output Option:	**Yes**
OK	
Apply	
Cancel	
Subcase Select...	
Subcases for Solution Sequence:	**nonlinear**
Subcases Selected:	*(Deselect* ***Default)***
OK	
Direct Text Input...	
	● **Bulk Data Section**
	NLPCI,1,CRIS,1.,1., , , ,25 *(Type in blank text box.)*
OK	
Apply	

An input file called **prob4c_1.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 8**.

Note: You must edit the input file before submitting it for an analysis.

7. Edit the input file.

Enter a text editor and make the following changes to the input file:

In the Bulk Data section, look for the CELAS entry and change the last parameter to 2.

CELAS 2 2 2 **2**

Generating an input file for MSC/NASTRAN Users:

8. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob4c_1.dat**):

```
ASSIGN OUTPUT2 = 'prob4c_1.op2' , UNIT=12
ID NAS103, WORKSHOP 4A SOLUTION
TIME 10
SOL 106
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
  OLOAD(SORT2)=ALL
SUBCASE 10
   LOAD=15
   NLPARM=30
BEGIN BULK
PARAM,POST,0
PARAM,LGDISP,1
$
$ 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, 0.
MAT1, 1, 10.E7
$
$ LOADS
$
FORCE, 15, 2,  , -15., 0., 1., 0.
$
$ SOLUTION STRATEGY
$
NLPARM, 30, 10, , AUTO, 5, 25, PW, YES
```

```
NLPCI, 30, CRIS, 1., 1.,  ,  ,  , 25
ENDDATA
```

Submit the input file for analysis:

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

   9a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob4c_1.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

   9b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob4c_1.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

10. When the analysis completed, edit the **prob4c_1.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether the existing WARNING messages indicate any modeling errors.

10a. While still editing **prob4c_1.f06**, search for the word:

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

What is the y-displacement of Node 2 at the end of the last step?

T2 = ____________________

Comparison of Results:

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

0 SUBCASE 1

POINT-ID = 2

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

STEP	TYPE	T1	T2	T3	R1	R2	R3
7.862365E-02	G	0.0	-1.500225E-01	0.0	0.0	0.0	0.0
1.189938E-01	G	0.0	-3.000450E-01	0.0	0.0	0.0	0.0
.							
.							
.							
2.347448E-01	G	0.0	-2.250337E+00	0.0	0.0	0.0	0.0
4.484861E-01	G	0.0	-2.400360E+00	0.0	0.0	0.0	0.0
7.252314E-01	G	0.0	-2.550382E+00	0.0	0.0	0.0	0.0
1.000000E+00	G	0.0	-2.671816E+00	0.0	0.0	0.0	0.0

L O A D   V E C T O R

STEP	TYPE	T1	T2	T3	R1	R2	R3
7.862365E-02	G	0.0	-1.179354E+00	0.0	0.0	0.0	0.0
1.189938E-01	G	0.0	-1.784906E+00	0.0	0.0	0.0	0.0
.							
.							
.							
2.347448E-01	G	0.0	-3.521173E+00	0.0	0.0	0.0	0.0
4.484861E-01	G	0.0	-6.727292E+00	0.0	0.0	0.0	0.0
7.252314E-01	G	0.0	-1.087847E+01	0.0	0.0	0.0	0.0
1.000000E+00	G	0.0	-1.500000E+01	0.0	0.0	0.0	0.0

12. MSC/NASTRAN users have finished the first part of this exercise. MSC/PATRAN users should proceed to the next step.

MSC/NASTRAN users should proceed to Step 15.

13. Proceed with the Reverse Translation process, that is, importing the **prob4c_1.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:	**prob4c_1.op2**
OK	
Apply	

14. Create an XY plot of Element Force vs Displacement.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**
Select Result Case(s)	*(Select all cases.)*
Y:	**Result**
Select Y Result	**Applied Loads, Translational**
Quantity:	**Y Component**

X: **Result**

Select X Result...

Select X Result **Displacements, Translational**

Quantity: **Y Component**

OK

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

 **Target Entities**

Target Entity: **Nodes**

Select Nodes **Node 2**

(Select node on the right.)

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Show X Axis Label**

X Axis Label: **Displacements**

X Axis Scale ● **Linear**

X Axis Format...

Label Format: **Fixed**

OK

■ **Show Y Axis Label**

Y Axis Label: **Applied Load**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

Apply

To change the title, do the following:

◆ **XY Plot**

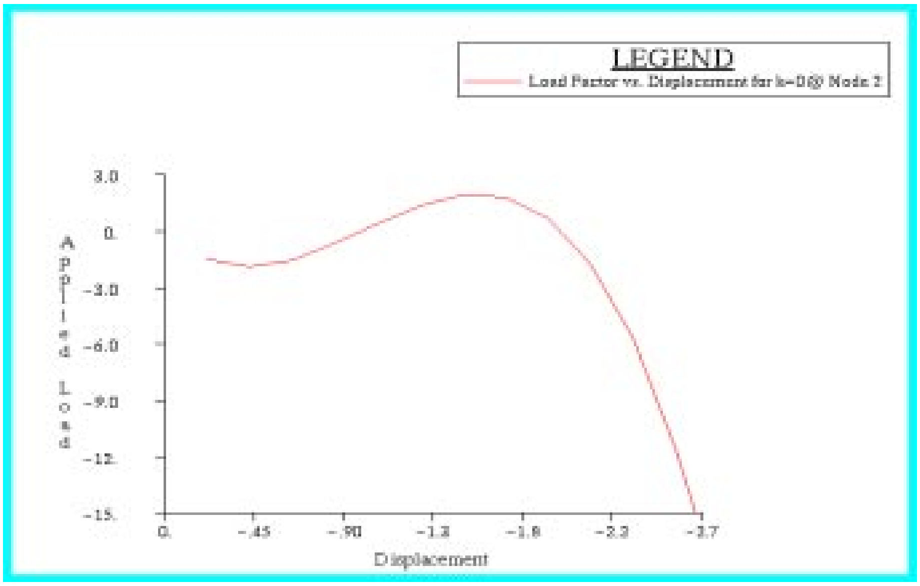
Action:	**Modify**
Object:	**Curve**
Curve List	**default_GraphResults Graph 0**
Title...	
Curve Title Text	**Load Factor vs. Displacement for k=0 @ Node 2**
Apply	
Cancel	

15. Modify the X-axis of the XY plot in order to better view the results.

◆ **XY Plot**

Action:	**Modify**
Object:	**Axis**
Active Axis	● **X**
Scale...	
Assignment Method	● **Range**
Enter Lower and Upper Values	**0 , -2.7**
Number of Primary Tick Marks	**7**
Apply	
Cancel	

The following XY plot should appear on the screen.



Notice the drastic displacement change for a small load increase above 1.5 lbs. This represents the snap-through aspect of the problem. Next, lets run the analysis for the cases when k=3 and k=6, and plot those curves as well.

16. MSC/NASTRAN users may modify the PELAS entry in the input file to account for the different spring constants, as well as change the input and output file names.

After this is done, MSC/NASTRAN users have completed the exercise.

17. MSC/PATRAN users will modify the spring constant, resubmit the analysis, and import the results. A sample algorithm for the next two analyses is as follows:

Set the spring constant to k=#.

◆ **Properties**

Action:	**Modify**
Dimension:	**0D**
Type:	**Grounded Spring**
Select Prop. Set to Modify	**spring**

Input Properties...

Spring Constant: | **#** *(# is 3 or 6)*

OK

Apply

Set up the analysis.

◆ **Analysis**

Action: | **Analyze**

Object: | **Entire Model**

Method: | **Analysis Deck**

Job Name | **prob4c_#** *(# is 2 or 3)*

Direct Text Input...

● **Bulk Data Section**

(Verify that text still says.) | **NLPCI,1.,CRIS,1.,1., , , ,25**

OK

Subcase Select...

Subcases for Solution Sequence: | **nonlinear**

Subcases Selected: | *(Deselect* ***Default.****)*

OK

Apply

Note: Be sure to edit the CELAS entry as show in Step 7.

Run the **.bdf** file through NASTRAN.

nastran prob4c_# scr=yes (where # is 2 or 3)

Check the **.f06** file for errors, and look at the displacements.

Read in the **.op2** file into PATRAN.

◆ **Analysis**

Action: **Read Output2**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Selected Results File: **prob4c_#.op2** *(# is 2 or 3)*

OK

Apply

Now add the curve to your XY plot.

◆ **Results**

Action: **Create**

Object: **Graph**

Method: **Y vs X**

Select the set of result cases that are to be added by highlighting them.

Select Result Case(s) *(Select the second or third set of cases.)*

Repeat the previous procedure for the new subcases with only one difference. Under the **Display Attributes** window in the **Results** form, click *on* the **Append Curves in XY Window** as shown below.

■ **Append Curves in XY Window**

Then click Apply.

Apply

To change the title, do the following:

◆ **XY Plot**

Action: **Modify**

Object: **Curve**

Curve List **default_GraphResults Graph ...**

(Select the corresponding graph.)

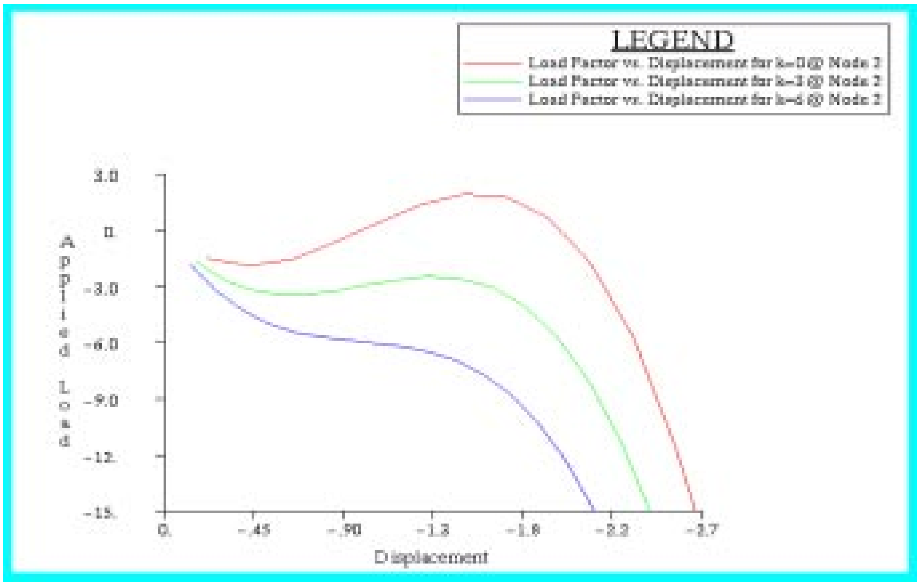
Title...

Curve Title Text | **Load Factor vs. Displacement for k=# @ Node 2**
(# is 3 or 6)

Apply

Cancel

After repeating the procedure for the remaining two cases, your plot should appear as follows:



Notice that as k is increased, the required load to produce a "snap-through" also increased.

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob4c_1.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 22:07:48.
ASSIGN OUTPUT2 = ‘prob4c_1.op2’, UNIT = 12
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 106
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 15-Jan-98 at 22:05:39
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : nonlinear
   SUBTITLE=Default
   NLPARM = 1
   SPC = 2
   LOAD = 2
   DISPLACEMENT(SORT2,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC YES
PARAM   COUPMASS -1
PARAM    K6ROT   100.
PARAM    WTMASS  1.
PARAM    LGDISP  1
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
NLPARM   1       70              ITER    1       25      PW      YES     +       A
+       A        .001    1.-7
$ Direct Text Input for Bulk Data
NLPCI,1,CRIS,1.,1.,,,,25
$ Elements and Element Properties for region : beam
PROD     1       1       .1
CROD     1       1       1       2
$ Elements and Element Properties for region : spring
PELAS    2       0.
CELAS1   2       2       2       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       15.     0.      -1.     0.
$ Referenced Coordinate Frames
ENDDATA 29a86b98
```