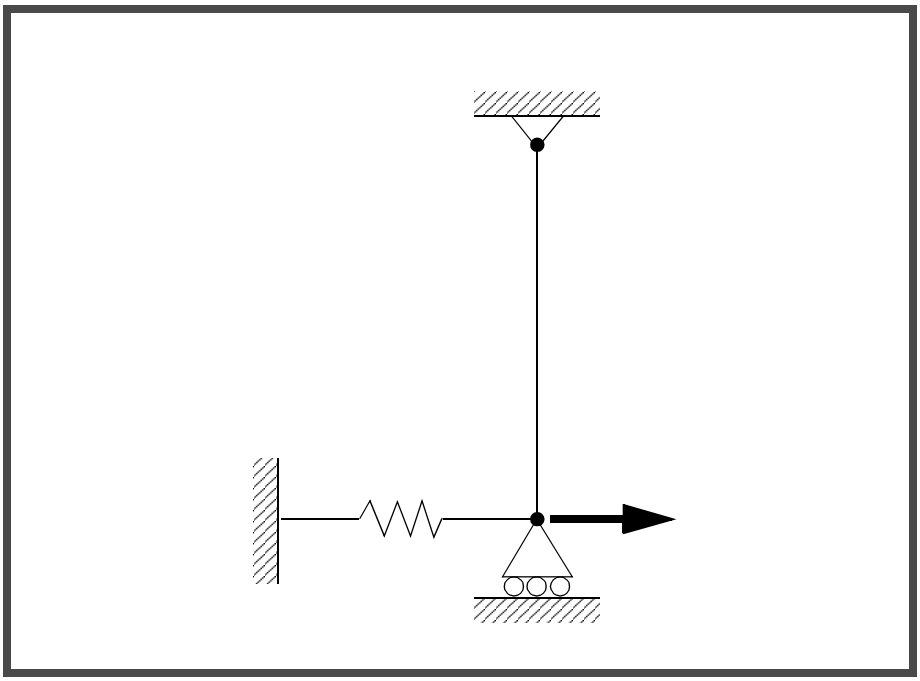
WORKSHOP PROBLEM 1b

Spring Element with Nonlinear Analysis Parameters (large displacements on)

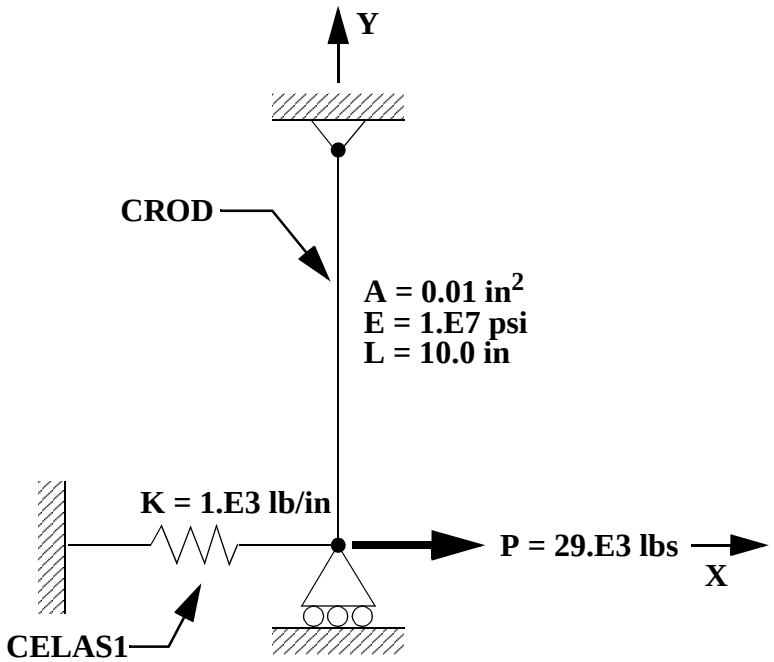


Objectives:

- Demonstrate the effects of geometric nonlinear analysis in SOL 106 (nonlinear statics).
- Demonstrate how to interpret the results.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Perform a geometric nonlinear analysis in SOL 106 (nonlinear statics) with the large displacements option turned on.

2. Apply a $29x10^3$ lbs load in a single subcase with 4 incremented steps. Use default values for all other subcase parameters. Request the output of all grid displacements and all element forces.

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 relevant nonlinear static analysis parameters for each subcase (NLPARM).
- Prepare the model for a nonlinear static analysis (turn on large displacements).
  - PARAM, LGDISP, 1
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modes analysis.
- Review the results.

Input File from Workshop 1a for Modification:

prob1a.dat

```
ASSIGN OUTPUT2 = 'prob1a.op2' , UNIT=12
ID NAS103, WORKSHOP 1A SOLUTION
TIME 10
SOL  106 $ NONLIN
CEND
TITLE=SIMPLE ROD SPRING - COLD ANALYSIS AND RESTART WORKSHOP
SUBTITLE=GEOMETRIC NONLINEAR
ECHO=BOTH
DISP=ALL
OLOAD=ALL
FORCE=ALL$
$ APPLY X LOAD
$
SUBCASE 10 $ LOAD=29.E03
 LABEL=APPLY LOAD P IN X DIRECTION = 29E+03
 LOAD=3
 NLPARM=10
OUTPUT(PLOT)
  SET 1 ALL
  MAXI DEFO 5.
  AXES Z, X, Y
  VIEW  0., 0., 0.
  FIND SCALE ORIGIN 1 SET 1
  PLOT STATIC 0 MAXIMUM DEFORMATION 5. SET 1
BEGIN BULK
PARAM, POST, -1
PARAM, PATVER, 3.0
GRID, 1, 0, 0.0, 0.0, 0.0,  , 23456
GRID, 3, 0, 0.0, 10.0, 0.0,  , 123456
CROD, 3, 3, 3, 1
CELAS1, 2, 2, 1, 1, 0
PROD, 3, 3, .01
PELAS, 2, 1.0E3
```

```
MAT1, 3, 1.0E7
FORCE, 1, 1, 0, 1.6E4, 1.0
FORCE, 2, 1, 0, 2.4E4, 1.0
FORCE, 3, 1, 0, 2.9E4, 1.0
PARAM, LGDISP,-1
NLPARM, 10, 4
ENDDATA
```

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 4, otherwise, proceed to step 2.

2. Open the database created from the last exercise called **prob1a.db**.

File/Open...

Database List: **prob1a**

OK

3. 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 **prob1b**

Solution Type...

Solution Type: ● **NONLINEAR STATIC**

Solution Parameters...

(Make sure large displacements are on.) ■ **Large Displacements**

OK

OK

Apply

An input file called **prob1b.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 5**.

Generating an input file for MSC/NASTRAN Users:

4. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob1b.dat**):

```
ASSIGN OUTPUT2 = 'prob1b.op2' , UNIT=12
ID NAS103, WORKSHOP 1B SOLUTION
TIME 10
SOL  106 $ NONLIN
CEND
TITLE=SIMPLE ROD SPRING - COLD ANALYSIS AND RESTART WORKSHOP
SUBTITLE=GEOMETRIC NONLINEAR
ECHO=BOTH
DISP=ALL
OLOAD=ALL
FORCE=ALL$
$ APPLY X LOAD
$
SUBCASE 10 $ LOAD=29.E03
 LABEL=APPLY LOAD P IN X DIRECTION = 29E+03
 LOAD=3
 NLPARM=10
OUTPUT(PLOT)
  SET 1 ALL
  MAXI DEFO 5.
  AXES Z, X, Y
  VIEW  0., 0., 0.
  FIND SCALE ORIGIN 1 SET 1
  PLOT STATIC 0 MAXIMUM DEFORMATION 5. SET 1
BEGIN BULK
PARAM, POST, -1
PARAM, PATVER, 3.0
GRID, 1, 0, 0.0, 0.0, 0.0,  , 23456
GRID, 3, 0, 0.0, 10.0, 0.0,  , 123456
CROD, 3, 3, 3, 1
CELAS1, 2, 2, 1, 1, 0
PROD, 3, 3, .01
PELAS, 2, 1.0E3
```

```
MAT1, 3, 1.0E7
FORCE, 1, 1, 0, 1.6E4, 1.0
FORCE, 2, 1, 0, 2.4E4, 1.0
FORCE, 3, 1, 0, 2.9E4, 1.0
PARAM, LGDISP,1
NLPARM, 10, 4
ENDDATA
```

Submit the input file for analysis:

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

   5a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob1b.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

   5b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob1b.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

6. When the analysis is completed, edit the **prob1b.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.

6a. While still editing **prob1b.f06**, search for the word:

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

What is the x-displacement of the guided end at the end of the analysis?

T1 = _____

What is the force in the spring element at the end of the analysis?

FORCE = _____

What is the force in the rod element at the end of the analysis?

FORCE = _____

Comparison of Results:

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

```
LOAD STEP =  1.00000E+00
                                      D I S P L A C E M E N T   V E C T O R

POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
       1      G      8.540191E+00   0.0            0.0            0.0            0.0            0.0
       2      G      0.0            0.0            0.0            0.0            0.0            0.0
0                                                                                                          SUBCASE 1


LOAD STEP =  1.00000E+00
                         F O R C E S   I N   S C A L A R   S P R I N G S       ( C E L A S 1 )
ELEMENT         FORCE            ELEMENT         FORCE            ELEMENT         FORCE            ELEMENT         FORCE
   ID.                              ID.                              ID.                              ID.
      2      8.540190E+03
0                                                                                             NONLINEAR    SUBCASE 1


LOAD STEP =  1.00000E+00
                                    F O R C E S   I N   R O D   E L E M E N T S     ( C R O D )
ELEMENT           AXIAL                                        ELEMENT           AXIAL
   ID.              FORCE          TORQUE                         ID.              FORCE          TORQUE
       1        3.150470E+04   0.0
```

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

9. Proceed with the Reverse Translation process, that is, importing the **prob1b.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 **prob1b.op2**

OK

Apply

10. 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, PW Linear: 100.% of Load_2**

*(Select the second **Default** case)*

Select Fringe Result **Displacements, Translational**

Quantity: **Magnitude**

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

Now create the deformation plot.

◆ **Results**

Action:	**Create**
Object:	**Deformation**

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

 **Select Results**

Select Result Case(s)	**Default, PW Linear: 100.% of Load_2**

*(Select second **Default** case)*

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

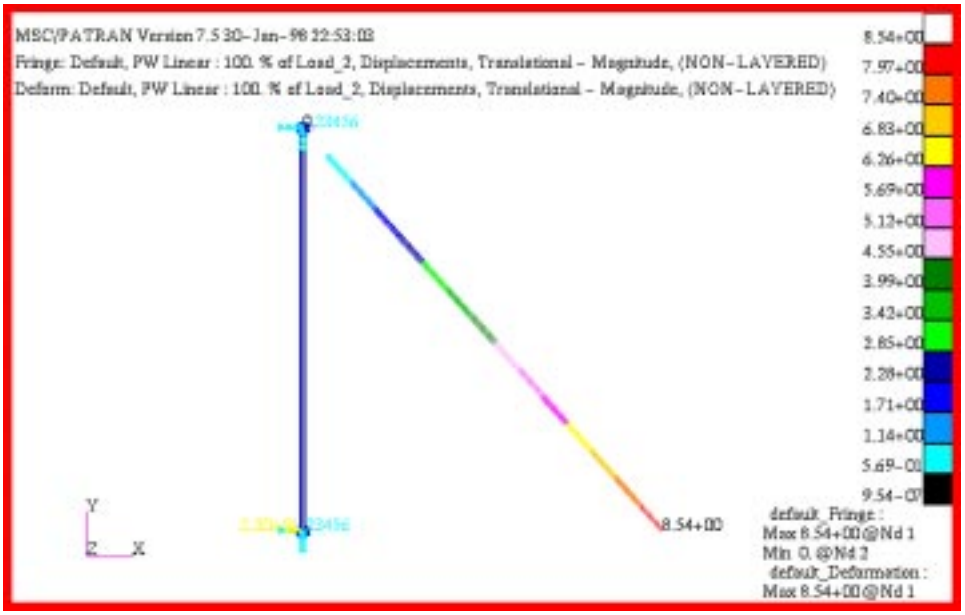
The resulting fringe plot should display the displacement spectrum in addition to the physical deformation of the model.

To better fit the results on the screen, zoom out a couple times using the following toolbar icon:

Zoom Out

Alternatively, use any number of the toolbar icons to better view the resulting fringe plot.

Your viewport should now contain the following image:



Notice that the deflection is now a much more reasonable approximation. The geometric nonlinearity of the problem has been better accounted for by using the Large Displacements option.

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: prob1b.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   16, 1998 at
$ 09:01:56.
ASSIGN OUTPUT2 = 'prob1b.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 16-Jan-98 at 08:25:44
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  FORCE(SORT1,REAL,BILIN)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC NO
PARAM   COUPMASS -1
PARAM    K6ROT   100.
PARAM    WTMASS  1.
PARAM    LGDISP  1
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
NLPARM   1       4               AUTO    5       25      PW      NO      +       A
+       A         .001    1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : prop_1
PROD     1       1       .01
CROD     1       1       1       2
$ Elements and Element Properties for region : prop_2
PELAS    2       1000.
CELAS1   2       2       1       1
```

```
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 19-Jun-97          Time: 15:12:40
MAT1     1       1.+7
$ Nodes of the Entire Model
GRID     1              0.      0.      0.
GRID     2              0.      10.     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  2
$ Displacement Constraints of Load Set : constraint_2
SPC1     3       23456   1
$ Nodal Forces of Load Set : load_3
FORCE    1       1       0       29000.  1.      0.      0.
$ Referenced Coordinate Frames
ENDDATA f7b9b4a4
```