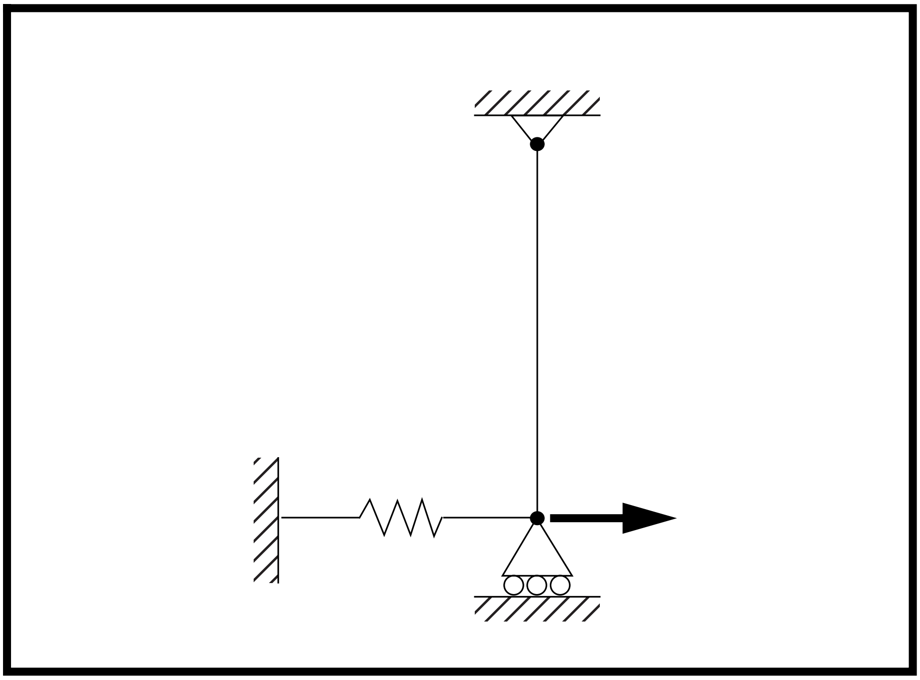
WORKSHOP PROBLEM 1a

Spring Element with Nonlinear Analysis Parameters (large displacements off)

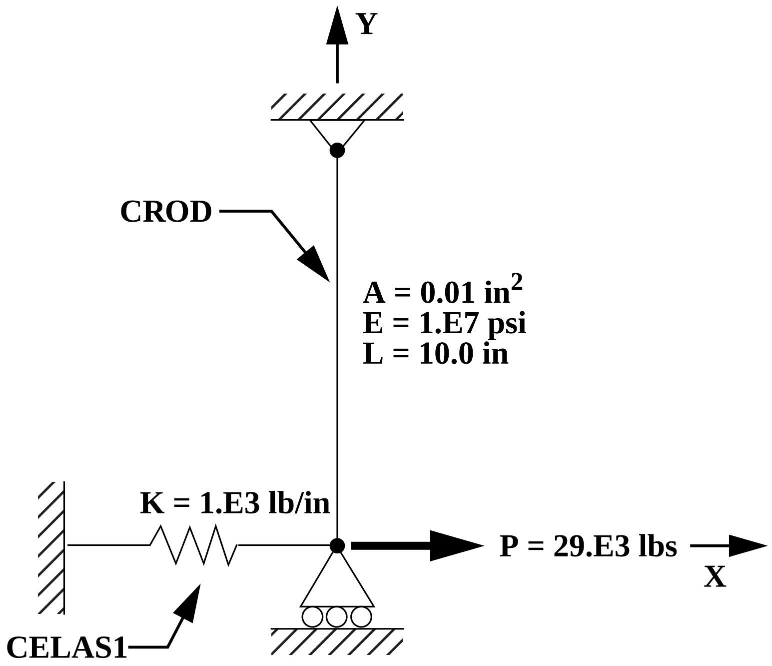


Objectives:

- Demonstrate how to run a simple linear analysis in SOL 106 (nonlinear statics).
- Demonstrate how to interpret the results.
- Understand the difference between linear and nonlinear behavior.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

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

2. Apply a $29x10^3$ lbs load in a single subcase with four incremented steps. Use the 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.
- Select Element Forces as part of the output (FORCE=ALL).
- For Case Control, insert the static load set selection (LOAD) and the nonlinear static analysis parameter selection (NLPARM).
- For Bulk Data, insert the relevant nonlinear static analysis parameter (NLPARM).
- Prepare the model for a geometric linear static analysis (turn off large displacements).
    - PARAM, LGDISP, -1
- Generate an input file and submit it to the MSC/NASTRAN solver for a nonlinear static analysis.
- Review the results.

Input File for Modification:

prob1a.dat

```
ID NAS103, WORKSHOP 1A
TIME 10
SOL  106 $ NONLIN
CEND
TITLE=SIMPLE ROD SPRING - COLD ANALYSIS AND RESTART WORKSHOP
SUBTITLE=GEOMETRIC NONLINEAR
ECHO=BOTH
DISP=ALL
OLOAD=ALL
$
$ APPLY X LOAD
$
SUBCASE 10 $ LOAD=29.E03
 LABEL=APPLY LOAD P IN X DIRECTION = 29E+03
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
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
ENDDATA
```

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 12, otherwise, proceed to step 2.

2. Create a new database called **prob1a.db**.

File/New...

New Database Name	**prob1a**
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, **prob1.ses**. If you choose to build the model yourself, proceed to step 4.

File/Session/Play...

Session File List	**prob1.ses**
Apply	

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

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

4. Create a 10 unit long beam.

◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**
Vector Coordinate List	**<0, 10, 0>**
Apply	

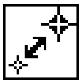
5. Mesh the curve with one BAR2 element.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length	**10**
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 =	**1.E7**

Apply

Cancel

7. Create the property for the beam.

◆ **Properties**

Action:	**Create**

Dimension:	**1D**
Type:	**Rod**
Property Set Name	**prop_1**
Input Properties...	
Material Name	**m:mat_1**
Area	**0.01**
OK	
Select Members	**Curve 1** *(Select the curve.)*
Add	
Apply	

8. Create a grounded spring at the bottom of the beam.

First, create a 0-D element to be used for spring constant assignment at the bottom of the beam.

◆ **Finite Elements**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Point**
Topology:	**Point**
Node 1 =	**Node 1** *(Select the bottom node.)*
Apply	

Next, create the grounded spring property for the newly created element.

◆ **Properties**

Action:	**Create**
Object:	**0D**
Method:	**Grounded Spring**

Property Set Name	**prop_2**
Input Properties...	
Spring Constant	**1.E3**
Dof at Node 1	**UX**
OK	

You can either type in the point element into the databox or click on the point itself. This is done by first, clicking the Select Members databox and then clicking on the following icon.

Point Element

Select Members:	**Elm 2** *(Select the point element previously created.)*
Add	
Apply	

9. Create the Loads/BCs for the model.

First, fix the top 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: **Point 2**

(Select point at top of beam.)

Add

OK

Apply

Next, create the guided support Load/BC at the base (free in x-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 **Point 1**

(Select point at bottom of beam.)

Add

OK

Apply

10. Create the loading for the model.

For this analysis model, the load will be incremented up to the final load in a single subcase.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name **load_3**

Input Data...

Force <F1 F2 F3> **<29.E3, 0, 0>**

OK

Select Application Region...

Select Geometry Entities **Point 1**

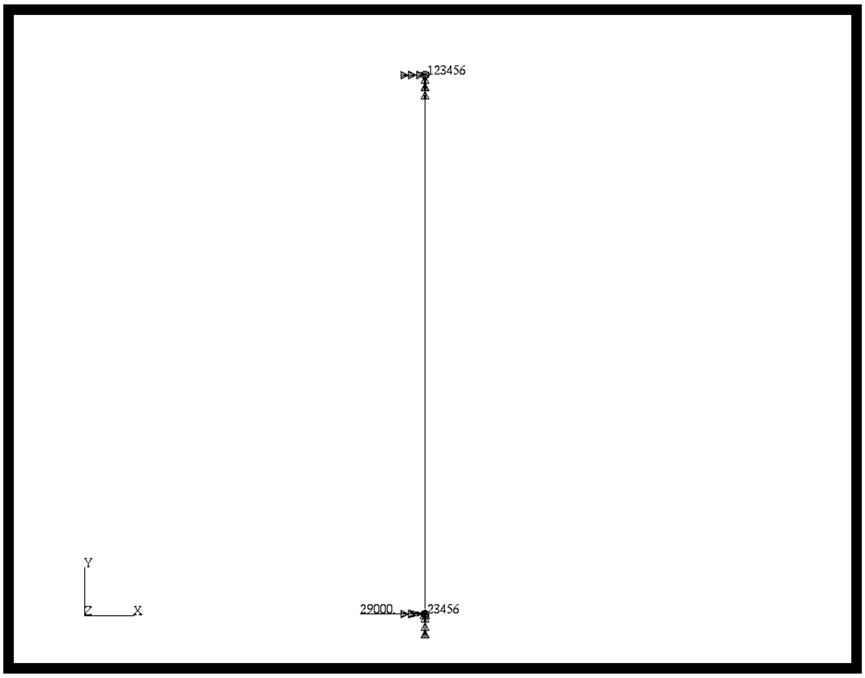
(Select point at bottom of beam.)

Add

OK

Apply

Your viewport should now appear as follow:



11. 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	**prob1a**

Solution Type...	
Solution Type:	● **NONLINEAR STATIC**
Solution Parameters...	
(Turn off large displacements.)	❐ **Large Displacements**
OK	
OK	
Subcase Create...	
Available Subcases:	**Default**
Subcase Parameters...	
Number of Load Increments:	**4**
OK	
Output Requests...	
Output Requests:	**STRESS(SORT1...**
Delete	
Output Requests:	**SPCFORCES(SORT1...**
Delete	
Select Result Type:	**Element Forces**
OK	
Apply	
Cancel	
Apply	

An input file called **prob1a.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 13**.

Generating an input file for MSC/NASTRAN Users:

12. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**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
```

Submit the input file for analysis:

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

    13a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob1a.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

    13b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob1a.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

14. When the analysis is completed, edit the **prob1a.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.

14a. While still editing **prob1a.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 = ____________________

Comparison of Results:

15. 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	2.900000E+01	0.0	0.0	0.0	0.0	0.0
2	G	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 ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE	ELEMENT ID.	FORCE
2	2.900000E+040						

SUBCASE 1

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

17. Proceed with the Reverse Translation process, that is, importing the **prob1a.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 **prob1a.op2**

OK

Apply

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

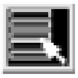
Now we will generate the fringe plot of the model.

◆ **Results**

Action: **Create**

Object: **Fringe**

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



Select Results

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

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

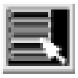
The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action: **Create**

Object: **Deformation**

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

 **Select Results**

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

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**

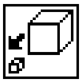
Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

The resulting fringe plot should display the displacement spectrum in addition to the physical deformation of the model. The following step will combine the two previous plots into one. MSC/PATRAN automatically combines both the fringe and deformation plots into one.

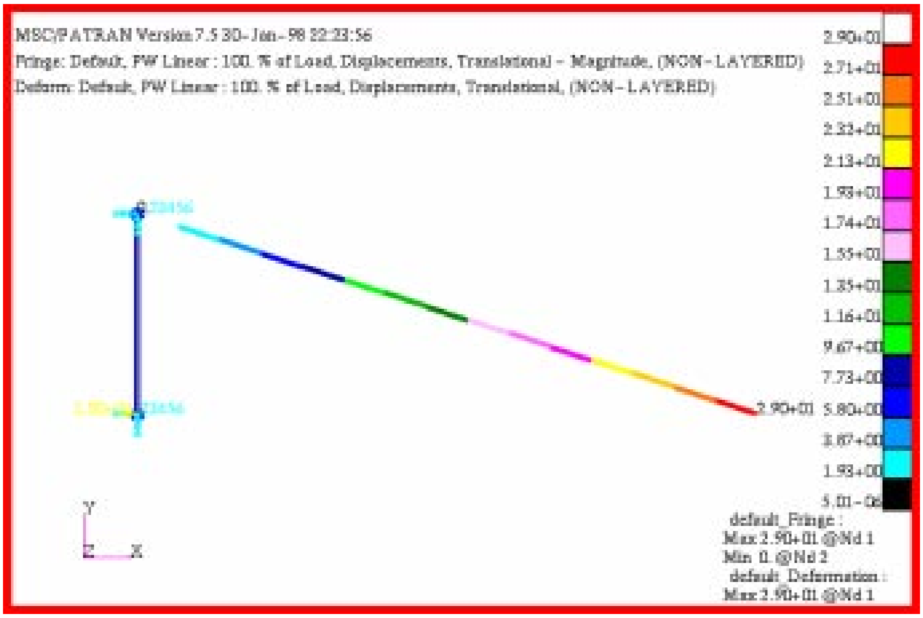
To better fit the results on the screen, zoom out a couple times using the following toolbar icon:



Zoom Out

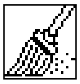
Alternatively, use any number of the toolbar icons to better view the resulting fringe plot.

Your viewport should now contain the following image:



Notice that the deflection is almost 3 times the length of the beam! This suggests that a nonlinear analysis which accounts for large displacements is necessary to obtain a more accurate answer.

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: prob1a.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   16, 1998 at
$ 08:27:27.
ASSIGN OUTPUT2 = 'prob1a.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 c9d4ca67
```