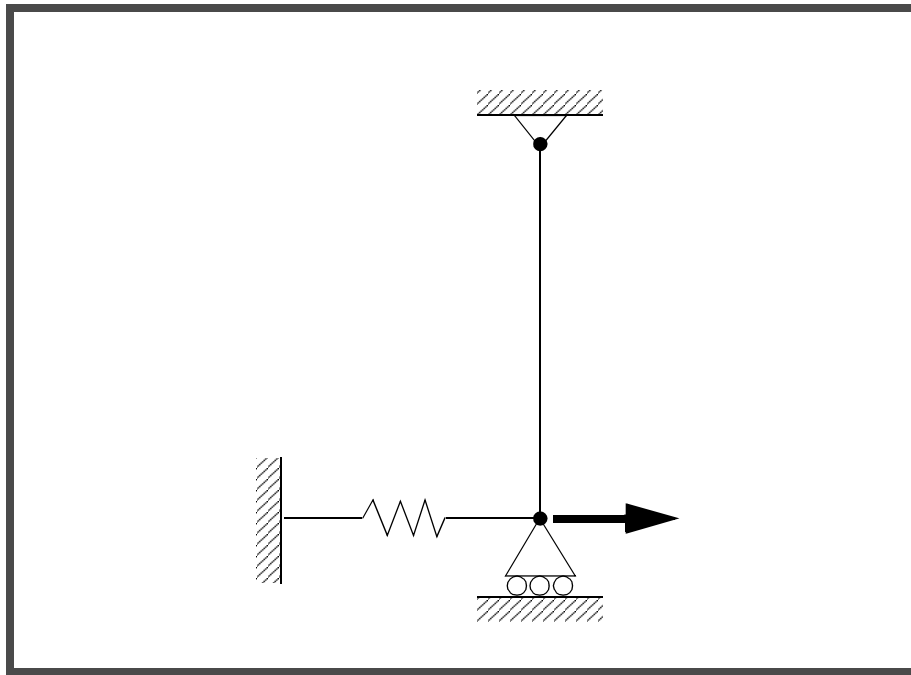

WORKSHOP PROBLEM 1a

*Spring Element with Nonlinear
Analysis Parameters
(large displacements off)*

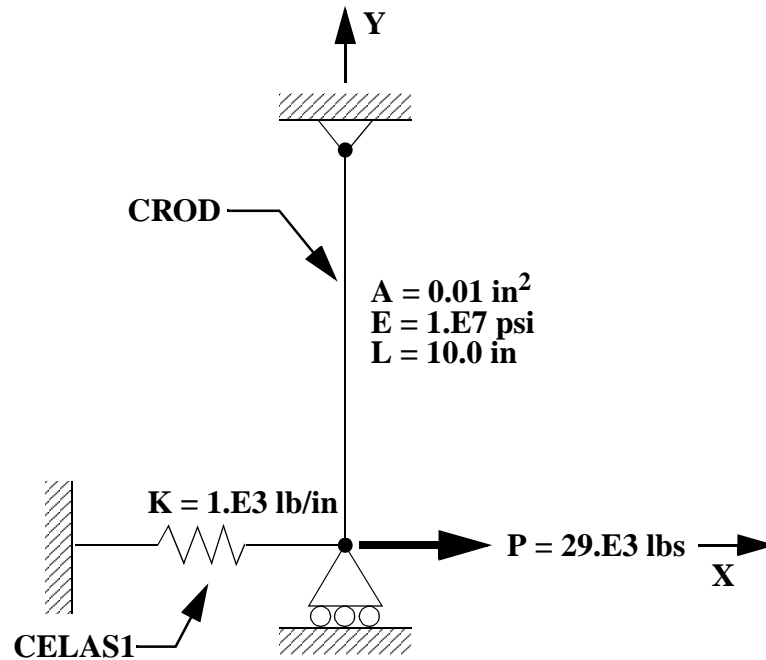


Objectives:

- 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 29×10^3 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 (NLPARAM).
- For Bulk Data, insert the relevant nonlinear static analysis parameter (NLPARAM).
- 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 utilizing 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**

<i>Action:</i>	Create
<i>Object:</i>	Mesh
<i>Type:</i>	Curve
<i>Global Edge Length</i>	10
<i>Element Topology:</i>	Bar2
<i>Curve List</i>	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**

<i>Action:</i>	Create
<i>Object:</i>	Isotropic
<i>Method:</i>	Manual Input
<i>Material Name</i>	mat_1

Input Properties...

<i>Elastic Modulus =</i>	1.E7
--------------------------	-------------

Apply

Cancel

7. Create the property for the beam.

◆ **Properties**

<i>Action:</i>	Create
----------------	---------------

<i>Dimension:</i>	<input type="text" value="1D"/>
<i>Type:</i>	<input type="text" value="Rod"/>
<i>Property Set Name</i>	<input type="text" value="prop_1"/>
<input type="button" value="Input Properties..."/>	
<i>Material Name</i>	<input type="text" value="m:mat_1"/>
<i>Area</i>	<input type="text" value="0.01"/>
<input type="button" value="OK"/>	
<i>Select Members</i>	<input type="text" value="Curve 1"/> <i>(Select the curve.)</i>
<input type="button" value="Add"/>	
<input type="button" value="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**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Element"/>
<i>Method:</i>	<input type="text" value="Edit"/>
<i>Shape:</i>	<input type="text" value="Point"/>
<i>Topology:</i>	<input type="text" value="Point"/>
<i>Node 1 =</i>	<input type="text" value="Node 1"/> <i>(Select the bottom node.)</i>
<input type="button" value="Apply"/>	

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

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="0D"/>
<i>Method:</i>	<input type="text" value="Grounded Spring"/>

<i>Property Set Name</i>	<input type="text" value="prop_2"/>
Input Properties...	
<i>Spring Constant</i>	<input type="text" value="1.E3"/>
<i>Dof at Node 1</i>	<input type="text" value="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.



<i>Select Members:</i>	<input type="text" value="Elm 2"/>
	<i>(Select the point element previously created.)</i>
Add	
Apply	

9. Create the Loads/BCs for the model.

First, fix the top end of the beam.

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Displacement"/>
<i>Method:</i>	<input type="text" value="Nodal"/>
<i>New Set Name</i>	<input type="text" value="constraint_1"/>
Input Data...	
<i>Translation < T1 T2 T3 ></i>	<input type="text" value="< 0, 0, 0 >"/>
<i>Rotation < R1 R2 R3 ></i>	<input type="text" value="< 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

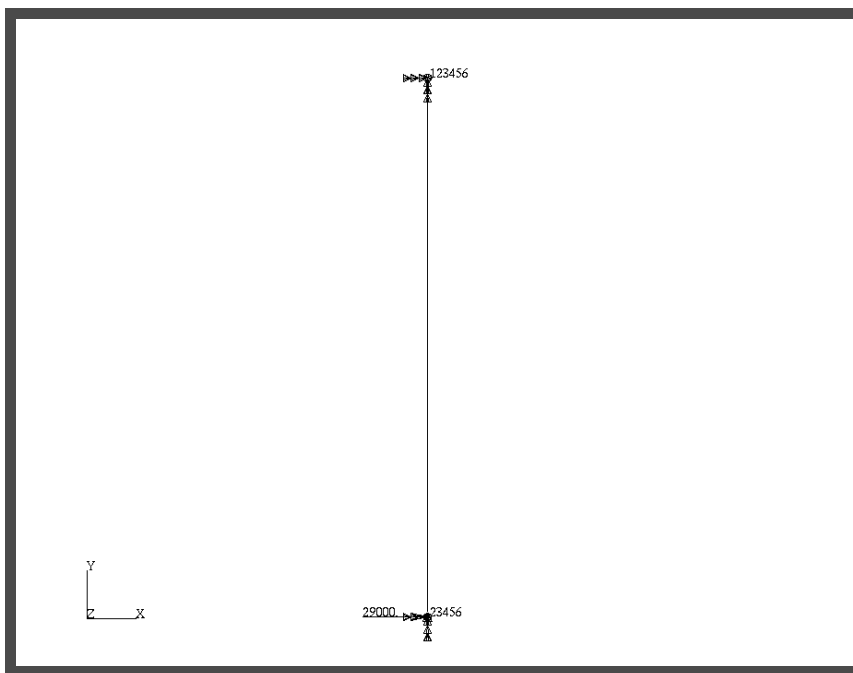
(Select point at bottom of beam.)

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

probl1a

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 probla.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 probla.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.
14. When the analysis is completed, edit the **probla.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 **probla.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

POINT ID.	TYPE	D I S P L A C E M E N T V E C T O R					
		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

E L E M E N T		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)	
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.

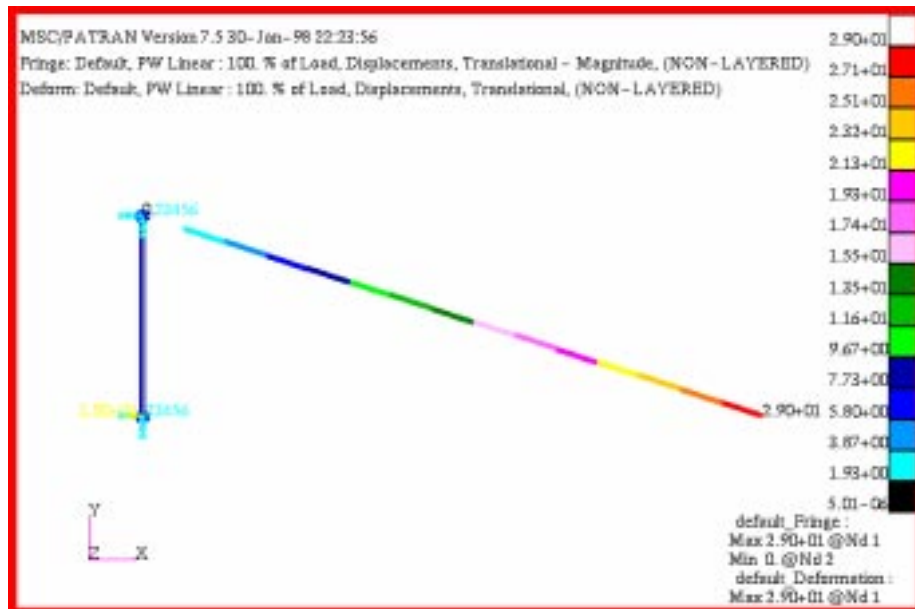
To better fit the results on the screen, zoom out a couple times using the following toolbar icon:



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
```

