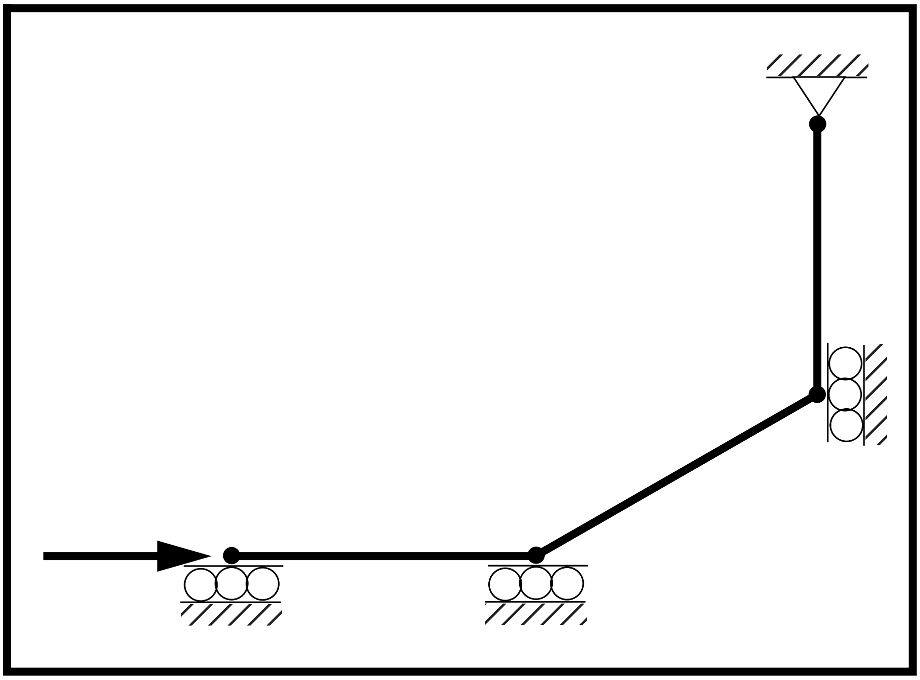
WORKSHOP PROBLEM 3

Load Deflection of a 3-Rod Structure

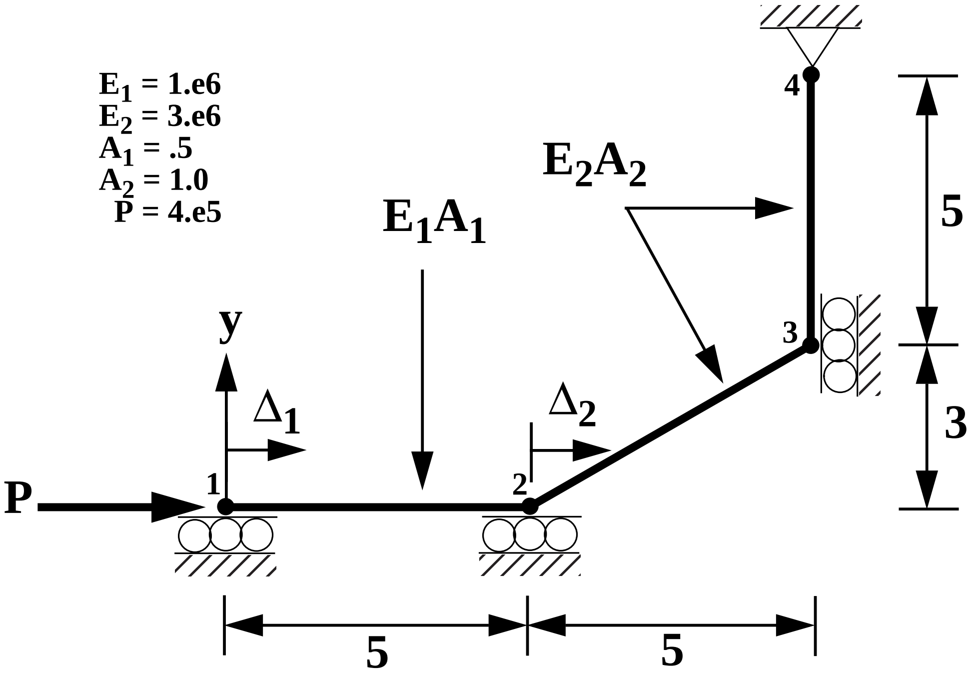


Objectives:

- Demonstrate the use of geometric nonlinear analysis.
- Demonstrate the use of arc length increments.
- Create an animation of the deformation of the structure.
- Create an XY Plot of Load Factor vs. Displacement.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Perform a geometric nonlinear analysis.
2. Compute the load-deflection behavior of the three-rod structure shown above.

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 nonlinear static analysis parameter selection (NLPARM).
- For Bulk Data, insert all relevant nonlinear static analysis parameters (NLPARM).
- Prepare the model for a nonlinear static analysis.
    - PARAM, LGDISP, 1
- Insert 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.

Input File for Modification:

prob3.dat

```
ID NAS103, WORKSHOP 3
TIME 10
SOL 106
CEND
TITLE=GEOMETRIC NONLINEAR PROBLEM
SUBTITLE=REF: POWELL AND SIMONS, IJNME, 17:1455-1467, 1981
DISP=ALL
SPCF=ALL
OLOAD=ALL
SPC=12
LOAD=10
BEGIN BULK
$ GEOMETRY
GRID,1,,0.0,0.
GRID,2,,5.,0.
GRID,3,,10.,3.
GRID,4,,10.,8.
GRDSET,,,,,,,3456
$ CONNECTIVITY
CONROD,1,1,2,10,.5
CONROD,2,2,3,11,1.
CONROD,3,3,4,11,1.
$ PROPERTIES
MAT1,10,1.+6
MAT1,11,3.+6
$ CONSTRAINTS
SPC1,12,1,3
SPC1,12,2,1,2
SPC1,12,12,4
$ LOADING
FORCE,10,1,,4.+5,1.,0.,0.
$ PARAMETERS
PARAM,POST,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 2.**

2. Create a new database called **prob3.db**.

File/New...

New Database Name	**prob3**

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, **prob3.ses**. If you choose to build the model yourself, proceed to step 4.

File/Session/Play...

Session File List	**prob3.ses**

Apply

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

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

4. Create the baseline NASTRAN finite element model.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
	❐ **Associate with Geometry**
Node ID List	**1**

Node Location List	**[0, 0, 0]**

Apply

Repeat the above procedure to create the other 3 nodes.

Node ID List	**2**
Node Location List	**[5, 0, 0]**

Apply

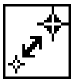
Node ID List	**3**
Node Location List	**[10, 3, 0]**

Apply

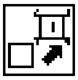
Node ID List	**4**
Node Location List	**[10, 8, 0]**

Apply

For clarity, increase the node size and turn on the entity labels by using the following toolbar icons:



Node Size



Show Labels

5. Create the rod elements for the model.

◆ **Finite Elements**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Bar**
Node 1 =	**Node 1**
Node 2 =	**Node 2**

Apply

Repeat the above proccedure to create the other two elements.

Node 1 =	**Node 2**
Node 2 =	**Node 3**

Apply

Node 1 =	**Node 3**
Node 2 =	**Node 4**

Apply

6. Create the material properties for the beams.

◆ **Materials**

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

Input Properties...

Elastic Modulus =	**1.E6**

Apply

Cancel

Material Name:	**mat_2**

Input Properties...

Elastic Modulus =	**3.E6**

Apply

Cancel

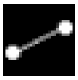
7. Define the rod properties.

◆ **Properties**

Action:	**Create**

Object:	**1D**
Method:	**Rod**
Property Set Name	**rod_1**
Input Properties...	
Material Name	**m:mat_1**
Area:	**0.5**
OK	

In order to pick the beam elements, click on the following entity select icon from the small menu window:

 **Beam Element**

Select Members	**Elm 1**
Add	
Apply	

Create the second property set.

Property Set Name	**rod_2**
Input Properties...	
Material Name	**mat_2**
Area:	**1.0**
OK	
Select Members	**Elm 2, 3**
Add	
Apply	

8. Create the load for the model.

◆ **Loads/BCs**

Action:	**Create**

Object:	**Force**
Method:	**Nodal**
New Set Name	**load_1**
Input Data...	
Force <F1 F2 F3>	**<4.E5, 0, 0>**
OK	
Select Application Region...	
Geometry Filter	● **FEM**
Select Nodes	**Node 1**
Add	
OK	
Apply	

9. Create the three boundary constraints for the model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name	**constraint_1**
Input Data...	
Translation < T1 T2 T3 >:	**< 0, , >**
OK	
Select Application Region...	
Select Nodes:	**Node 3**
Add	
OK	
Apply	
New Set Name	**constraint_2**

Input Data...

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

OK

Select Application Region...

Select Nodes: **Node 1, 2**

Add

OK

Apply

New Set Name: **constraint_3**

Input Data...

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

OK

Select Application Region...

Select Nodes: **Node 4**

Add

OK

Apply

10. Now generate the analysis input file.

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

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name **prob3**

Solution Type...

Solution Type:	● **NONLINEAR STATIC**
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Load Increments=	**4**
OK	
Output Requests...	
Form Type:	**Advanced**
Output Requests	*(Select **STRESS(SORT1,...)***
Delete	
Select Result Type:	*(Select **Applied Loads.**)*
Create	
Intermediate Output Option:	**Yes**
OK	
Apply	
Cancel	
Direct Text Input...	
(Make sure this is selected!)	● **Bulk Data Section**
Bulk Data Section	**NLPCI,1** *(Type in blank box.)*
OK	
Apply	

An input file called **prob3.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 (**prob3.dat**):

```
ASSIGN OUTPUT2 = ‘prob3.op2’, UNIT = 12
ID NAS103, WORKSHOP 3 SOLUTION
TIME 10
SOL 106
CEND
TITLE=GEOMETRIC NONLINEAR PROBLEM
SUBTITLE=REF: POWELL AND SIMONS, IJNME, 17:1455-1467, 1981
DISP=ALL
SPCF=ALL
OLOAD=ALL
SPC=12
LOAD=10
SUBCASE 10
NLPARM=10
BEGIN BULK
$ GEOMETRY
GRID,1,,0.0,0.
GRID,2,,5.,0.
GRID,3,,10.,3.
GRID,4,,10.,8.
GRDSET,,,,,,,3456
$ CONNECTIVITY
CONROD,1,1,2,10,.5
CONROD,2,2,3,11,1.
CONROD,3,3,4,11,1.
$ PROPERTIES
MAT1,10,1.+6
MAT1,11,3.+6
$ CONSTRAINTS
SPC1,12,1,3
SPC1,12,2,1,2
SPC1,12,12,4
```

```
$ LOADING
FORCE,10,1,,4.+5,1.,0.,0.
$ PARAMETERS
PARAM,POST,0
$ SOLUTION STRATEGY
NLPARM,10,40,,,,,,YES
NLPCI,10
PARAM,LGDISP,1
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 prob3.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 prob3.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

13. When the analysis is completed, edit the **prob3.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 **prob3.f06**, search for the word:

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

What is the x-displacement of **Node 1** at load step = 1.0?

T1 = ____________________

What is the x-displacement of **Node 2** at load step = 1.0?

T1 = ____________________

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

0 SUBCASE 1

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	1.510921E+01	0.0	0.0	0.0	0.0	0.0
2	G	1.110920E+01	0.0	0.0	0.0	0.0	0.0
3	G	0.0	-2.951657E-01	0.0	0.0	0.0	0.0
4	G	0.0	0.0	0.0	0.0	0.0	0.0

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 **prob3.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:	**prob3.op2**
OK	
Apply	

When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

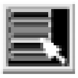
17. Create an animation that demonstrates the snap-through deformation.

Now we will generate the fringe plot of the model.

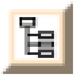
◆ **Results**

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

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

 **Select Results**

Make sure that the **View Subcases** icon is *off*. It should appear like the following.



View Subcases

Select Result Case(s)	*(Select the last case.)*
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

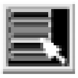
The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action: **Create**

Object: **Deformation**

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

 **Select Results**

Select Result Case(s) *(Select the last case.)*

Select Fringe Result **Displacements, Translational**

Show As: **Resultant**

Click on the **Display Attributes** icon.

 **Display Attributes**

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

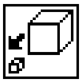
Now click on the **Animation Options** icon.

 **Animation Options**

Animation Method:	**Ramp**
Animation Graphics	● **2D**
Number of Frames:	**9**
Interpolation	**Linear**

Apply

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.

You can see the physical deformation of the model as well as the amount of deformation from the fringe.

Notice the loading on the middle section shifts from compression to tension as the lower section “snaps-through” the initial setup. This in an instance when nonlinear analysis is definitely required to obtain an accurate solution.

When you wish to stop the animation, click on the following button:

Stop Animation

When asked if you wish to clear the current animation, select **Yes**.

Yes

18. Create an XY plot of X-Displacement vs. Percent of Load.

Create an XY plot of Load Factor vs. Displacement.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

Select all the Result Cases by highlighting them.

Select Result Case(s)	*(Select all cases.)*
Y:	**Result**
Select Y Result	**Applied Loads, Translational**
Quantity:	**X Component**
X:	**Result**

Select X Result...

Select X Result	**Displacements, Translational**
Quantity:	**X Component**

OK

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

 **Target Entities**

Target Entity:	**Nodes**
Select Nodes	**Node 1** *(Select node at end of beam.)*

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Show X Axis Label**

X Axis Label:	**Displacement**
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	

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

 **Plot Options**

Coordinate Transformation:	**None**
Scale Factor	**1.0**
Apply	

To change the title, do the following:

◆ **XY Plot**

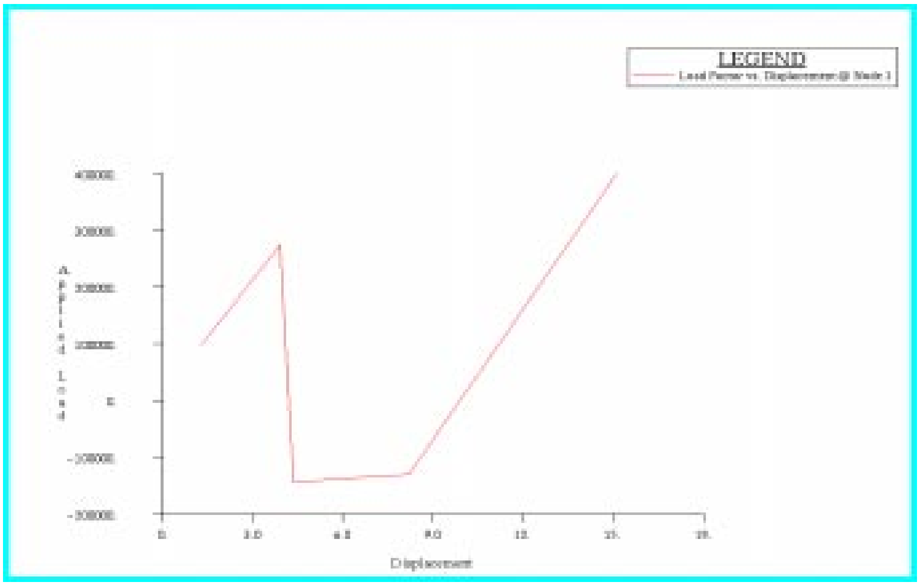
Action:	**Modify**
Object:	**Curve**
Curve List	**default_GraphResults Graph 0**
Title...	

Curve Title Text	**Load Factor vs. Displacement @ Node 1**

Apply

Cancel

The following XY plot should appear on your screen:



Notice the load factor makes a dramatic shift to negative values when the deformation is between 5 to 6 inches. This is caused by the snap-through behavior of the model. Eventually, the load does reach its full (positive) value as the snap-through effects die out.

Again, behavior such as this can only be determined by a geometric nonlinear analysis. A linear static analysis would not reveal this behavior.

When done viewing, delete the XY plot by doing the following:

◆ **XY Plot**

Action:	**Post**
Object:	**XYWindow**

Post/Unpost XY Windows: *(hold* ***<ctrl>*** *and click on* **Results Graph** *to deselect it.)*

Apply

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob3.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 14:08:17.
ASSIGN OUTPUT2 = 'prob3.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 14:06:49
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
   SPCFORCES(SORT1,REAL)=ALL
   OLOAD(SORT1,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       40              AUTO    5       25      PW      YES     +       A
+       A        .001    1.-7
$ Direct Text Input for Bulk Data
NLPCI, 1
$ Elements and Element Properties for region : rod_1
```

```
PROD     1       1       .5
CROD     1       1       1       2
$ Elements and Element Properties for region : rod_2
PROD     2       2       1.
CROD     2       2       2       3
CROD     3       2       3       4
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 30-May-97          Time: 17:00:10
MAT1     1       1.+6
$ Material Record : mat_2
$ Description of Material : Date: 30-May-97          Time: 17:00:10
MAT1     2       3.+6
$ Nodes of the Entire Model
GRID     1               0.      0.      0.
GRID     2               5.      0.      0.
GRID     3               10.     3.      0.
GRID     4               10.     8.      0.
$ Loads for Load Case : Default
SPCADD   2       1       3       4
LOAD     2       1.      1.      1
$ Displacement Constraints of Load Set : constraint_1
SPC1     1       1       3
$ Displacement Constraints of Load Set : constraint_2
SPC1     3       2       1       2
$ Displacement Constraints of Load Set : constraint_3
SPC1     4       12      4
$ Nodal Forces of Load Set : load_1
FORCE    1       1       0       400000. 1.      0.      0.
$ Referenced Coordinate Frames
ENDDATA b915b9bd
```