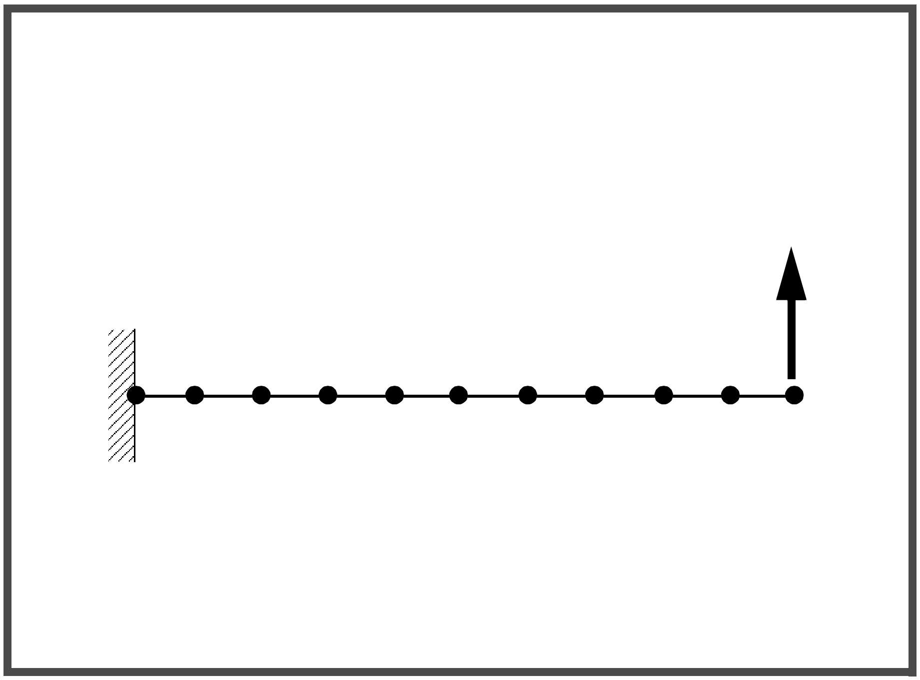
WORKSHOP PROBLEM 2a

Geometric Linear Analysis of Cantilever Beam

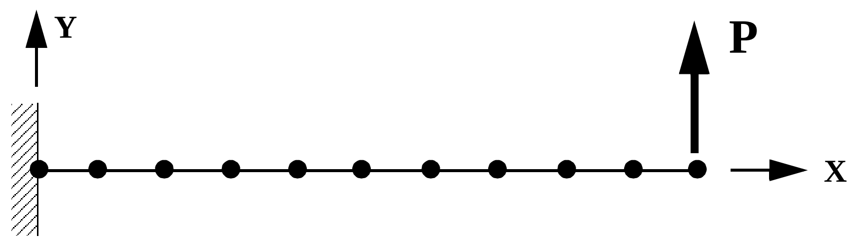


Objectives:

- Demonstrate the use of geometric linear analysis.
- Observe the behavior of the cantilever beam under four increasing load magnitudes.
- Create an accurate deformation plot of the model.
- Create a plot of the load factor vs. displacement.

Model Description:

For the structure below:



L = 10.0
A = 1.0
I = $1x10^{-2}$
E = 10^6
ν = 0.3

Add Case Control commands and Bulk Data Entries to:

1. Perform a geometric linear analysis.

2. Determine the behavior of the cantilever beam for the following four load cases:

    1) P = 2000
    2) P = 4000
    3) P = 6000
    4) P = 8000

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate loading conditions and linear static analysis control parameters.
- For Case Control, insert static load set selection (LOAD) in each subcase.
- Prepare the model for a linear static analysis (SOL 101).
- Generate an input file and submit it to the MSC/NASTRAN solver for a linear static analysis.
- Review the results.

Input File for Modification:

prob2a.dat

```
ID NAS103, WORKSHOP 2A
SOL 106
TIME 10
CEND
TITLE=TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM
SUBTITLE=REF.: BISSHOPP AND DRUCKER; QAM 3(1):272-275; 1945
SPC=1
DISP=ALL
OLOAD=ALL
$
SUBCASE 10
LOAD=200
$
SUBCASE 20
LOAD=400
$
SUBCASE 30
LOAD=600
$
SUBCASE 40
LOAD=800
$
BEGIN BULK
$ GEOMETRY
GRID,1,,0.,0.,0.,,345
=,*(1),=,*(1.),==$
=(9)
GRID,100,,0.,0.,1.,,123456
$ CONNECTIVITY
CBEAM,101,1,1,2,100
=,*(1),=,*(1),*(1),==$
=(8)
$ PROPERTIES
PBEAM,1,1,1.,1.-2,1.-2
MAT1,1,10.+6,,.0
$ CONSTRAINTS
SPC,1,1,123456
$ LOADING
FORCE,11,11,,1.+4,0.,1.,0.
```

```
LOAD,200,.2,1.,11
LOAD,400,.4,1.,11
LOAD,600,.6,1.,11
LOAD,800,.8,1.,11
$ 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 13, otherwise, proceed to step 2.

2. Create a new database called **prob2a.db**.

File/New...

New Database Name: **prob2a**

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, **prob2.ses**. If you choose to build the model yourself, proceed to the step 4.

File/Session/Play...

Session File List: **prob2.ses**

Apply

The model has now been created. Skip to **Step 12**.

4. Create a 10 unit long beam.

◆ **Geometry**

Action: **Create**

Object: **Curve**

Method: **XYZ**

Vector Coordinate List **<10, 0, 0>**

Apply

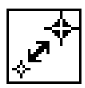
5. Mesh the curve with ten BAR2 elements.

◆ **Finite Elements**

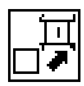
Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**1**
Element Topology:	**Bar2**
Curve List:	**Curve 1** *(Select the curve.)*

Apply

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



Node Size



Show Labels

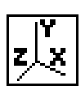
6. Create the reference grid for the bar orientation.

◆ **Finite Elements**

Action:	**Create**
Object:	**Node**
Method:	**Edit**
Node ID List	**100**
	❒ **Associate with Geometry**
Node Location List	**[0, 0, 1]**

Apply

For clarity, change the view using the following toolbar icon:



Iso 1 View

7. Create the material property for the beam.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**
Input Properties...	
Elastic Modulus =	**10.E6**
Poisson's Ratio =	**.3**
Apply	
Cancel	

8. Create the property for the beam.

◆ **Properties**

Action:	**Create**	
Object:	**1D**	
Method:	**Beam**	
Property Set Name:	**beam**	
Option(s):	**Tapered Section**	
Input Properties...		
Material Name:	**m:mat_1**	
Bar Orientation:	**Node 100**	**Node ID**
Cross Sect. Areas:	**1.0**	
[Inertias 1,1]:	**1e-2**	
[Inertias 2,2]:	**1e-2**	
OK		
Select Members:	**Curve 1** *(Select the curve.)*	

Add

Apply

9. Create the Loads/BCs for the model.

First, fix the left end of the beam and the orientation nodes.

◆ **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...

Geometry Filter	● **FEM**
Select Nodes	**Node 1, 100**

Add

OK

Apply

Next, constrain all the nodes to planar translation and rotation.

New Set Name	**constraint_2**

Input Data...

Translation < T1 T2 T3 >	**< , , 0 >**
Rotation < R1 R2 R3 >	**< 0, 0, >**

OK

Select Application Region...

Geometry Filter	● **FEM**

Select Nodes **Node 1:11**

Add

OK

Apply

10. Create the loading for the model.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name **force_1**

Input Data...

Force <F1 F2 F3> **<0, 2000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**
(Right end of beam.)

Add

OK

Apply

Create three more forces.

New Set Name **force_2**

Input Data...

Force <F1 F2 F3> **<0, 4000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

New Set Name **force_3**

Input Data...

Force <F1 F2 F3> **<0, 6000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

Finally, the last force.

New Set Name **force_4**

Input Data...

Force <F1 F2 F3> **<0, 8000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

11. Create the four load cases for the four loading conditions.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**subcase_1**

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet	**Displ_constraint_1** **Displ_constraint_2** **Force_force_1**

OK

Apply

Be sure that the Loads/BC Scale Factor is 1 for each of the Loads/BCs Types appearing in the spreadsheet.

Load Case Name:	**subcase_2**

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet	**Displ_constraint_1** **Displ_constraint_2** **Force_force_2**

OK

Apply

Load Case Name:	**subcase_3**

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet	**Displ_constraint_1** **Displ_constraint_2** **Force_force_3**

OK

Apply

Load Case Name	**subcase_4**
Assign/Prioritize Loads/BCs	
Remove All Rows	
Select Loads/BCs to Add to Spreadsheet	**Displ_constraint_1** **Displ_constraint_2** **Force_force_4**
OK	
Apply	

12. 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 analysis as follows:

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name	**prob2a**
Solution Type...	
Solution Type:	● **LINEAR STATIC**
OK	
Subcase Create...	
Available Subcases:	**subcase_1**
Output Requests...	
Form Type:	**Advanced**
Output Requests:	**STRESS(SORT...**
Delete	
Output Requests:	**SPCFORCE(SORT...**
Delete	
Select Result Type:	**Applied Loads**

Create

OK

Apply

Repeat the above procedure to create the second, third, and fourth subcases.

Available Subcases	**subcase_2**
Output Requests...	
Form Type:	**Advanced**
Select Result Type	**Applied Loads**
Output Requests	*(Deselect all but **DISPL(...**)*
Delete	
Create	
OK	
Apply	

Now create the third subcase.

Available Subcases	**subcase_3**
Output Requests...	
Form Type:	**Advanced**
Output Requests	*(Deselect all but **DISPL(...**)*
Delete	
Select Result Type	**Applied Loads**
Create	
OK	
Apply	

Finally, create the fourth subcase.

Available Subcases **subcase_4**

Output Requests...

Form Type: **Advanced**

Output Requests *(Select all but **DISPL(...**)*

Delete

Select Result Type **Applied Loads**

Create

OK

Apply

Cancel

Lastly, select all the subcases.

Subcase Select...

Subcases for Solution Sequence

subcase_1
subcase_2
subcase_3
subcase_4

(Select one after the other)

Subcases Selected: **Default**

OK

Apply

An input file called **prob2a.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 14**.

Generating an input file for MSC/NASTRAN Users:

13. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob2a.dat**):

```
ASSIGN OUTPUT2 = ‘prob2a.op2’, UNIT = 12
ID NAS103, WORKSHOP 2A SOLUTION
SOL 101
TIME 10
CEND
TITLE = TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM
SUBTITLE=REF.: BISSHOPP AND DRUCKER; QAM 3(1):272-275; 1945
SPC=1
DISP=ALL
OLOAD=ALL
$
SUBCASE 10
  LOAD = 200
$
SUBCASE 20
  LOAD = 400
$
SUBCASE 30
  LOAD = 600
$
SUBCASE 40
  LOAD = 800
$
BEGIN BULK
$ GEOMETRY
GRID,1,,0.,0.,0.,,345
=,*(1),=,*(1.),==$
=(9)
GRID,100,,0.,0.,1.,,123456
$ CONNECTIVITY
CBEAM,101,1,1,2,100
=,*(1),=,*(1),*(1),==$
=(8)
$ PROPERTIES
PBEAM,1,1,1.,1.-2,1.-2
MAT1,1,10.+6,,.3
```

```
$ CONSTRAINTS
SPC,1,1,123456
$ LOADING
FORCE,11,11,,1.+4,0.,1.,0.
LOAD,200,.2,1.,11
LOAD,400,.4,1.,11
LOAD,600,.6,1.,11
LOAD,800,.8,1.,11
$ PARAMETERS
PARAM,POST,-1
ENDDATA
```

Submit the input file for analysis:

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

14a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob2a.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

14b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob2a.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

15. When the analysis is completed, edit the **prob2a.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.

15a. While still editing **prob2a.f06**, search for the word:

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

What is the y-displacement of **Node 11** for the first subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the second subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the third subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the fourth subcase?

T2 = ____________________

Comparison of Results:

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

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
11	G	0.0	6.671867E+00	0.0	0.0	0.0	1.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	1.334373E+01	0.0	0.0	0.0	2.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	2.001560E+01	0.0	0.0	0.0	3.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	2.668747E+01	0.0	0.0	0.0	4.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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

18. Proceed with the Reverse Translation process, that is, importing the **prob2a.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	**prob2a.op2**
OK	
Apply	

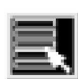
When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

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)	**subcase_4, Static Subcase**
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

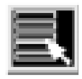
The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action: **Create**

Object: **Deformation**

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

 **Select Results**

Select Result Case(s) **subcase_4, Static Subcase**

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.

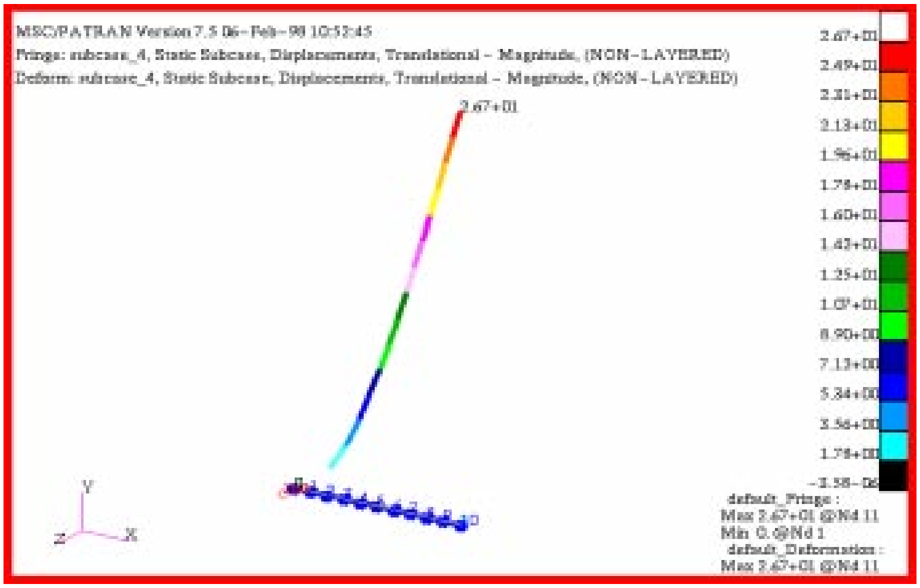
 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

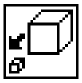
Apply

Your resulting plot should look similar to the following.



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

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

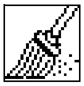


Zoom Out

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

Notice the extremely large deflection of the beam is. This suggests that a nonlinear geometric analysis is necessary in order to obtain a reasonable solution.

Click the Reset Graphics icon to clear the post-processing results and obtain the original model in the viewport.



Reset Graphics

19. 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: **Y Component**

X: **Result**

Select X Result...

Select X Result **Displacements, Translational**

Quantity: **Magnitude**

OK

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

Target Entities

Target Entity: **Nodes**

Select Nodes **Node 11**

(Select node at end of beam.)

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 Loads**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

Apply

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**

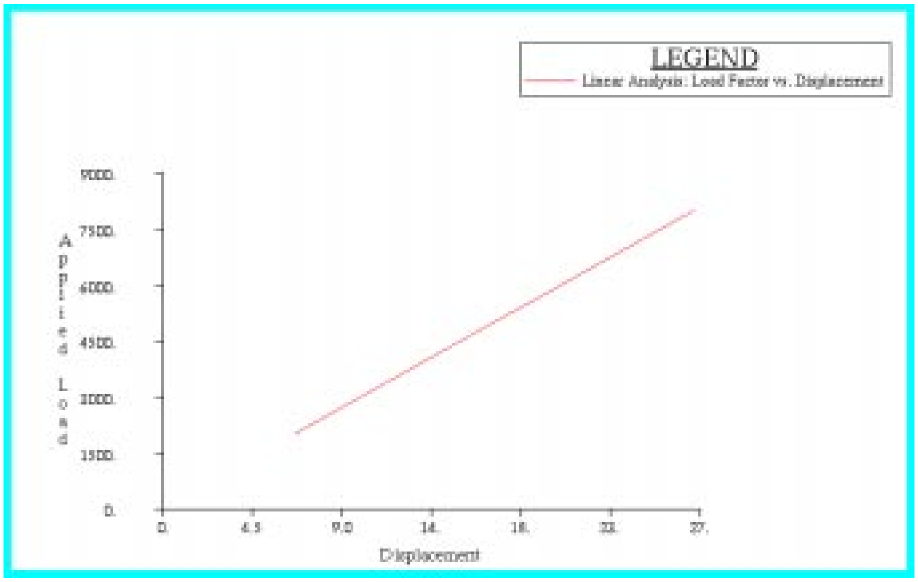
Title...

Curve Title Text **Linear Analysis: Load Factor vs. Displacement**

Apply

Cancel

Your XY Plot should appear as follows:



Notice the linear relationship between the displacement and the load factor.

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

◆ **XY Plot**

Action:	**Post**
Object:	**XYWindow**
Post/Unpost XYWindows	*(<ctrl>, click on **Results Graph** to de-select it.)*

Apply

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob2a.bdf

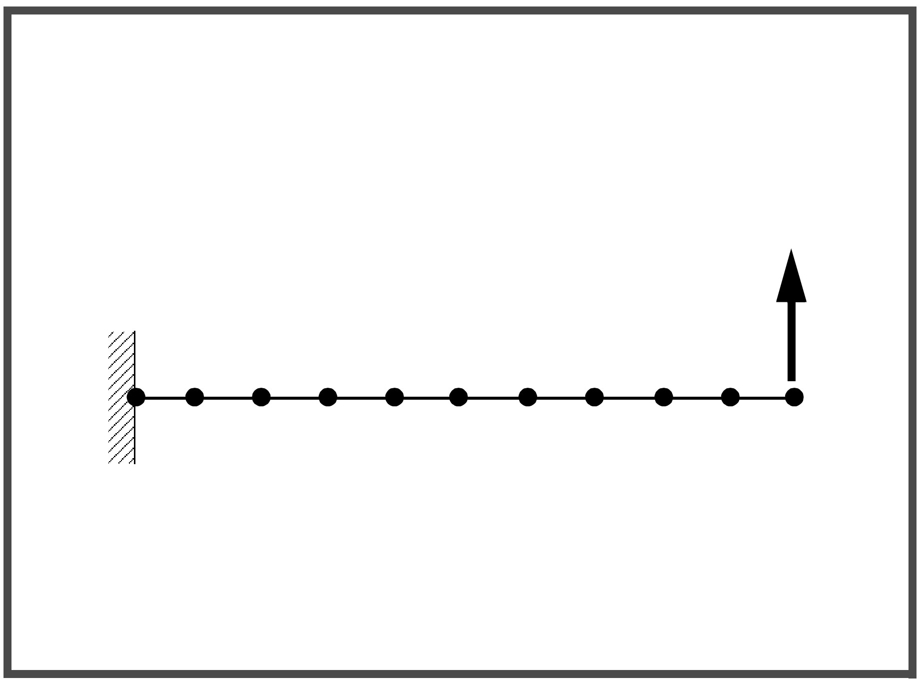
```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 13:10:51.
ASSIGN OUTPUT2 = ‘prob2a.op2’, UNIT = 12
$ Direct Text Input for File Management Section
$ Linear Static Analysis, Database
SOL 101
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 15-Jan-98 at 13:07:43
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : subcase_1
  SUBTITLE=subcase_1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : subcase_2
  SUBTITLE=subcase_2
  SPC = 2
  LOAD = 4
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 3
$ Subcase name : subcase_3
  SUBTITLE=subcase_3
  SPC = 2
  LOAD = 6
  DISPLACEMENT(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 4
$ Subcase name : subcase_4
```

```
  SUBTITLE=subcase_4
  SPC = 2
  LOAD = 8
  DISPLACEMENT(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    INREL   0
PARAM    ALTRED  NO
PARAM   COUPMASS -1
PARAM    K6ROT   0.
PARAM    WTMASS  1.
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : beam
PBEAM    1       1       1.      .01     .01                             +      A
+      A                                                         +      B
+      B YES     1.      1.      .01     .01                             +      C
+      C
CBEAM    1       1       1       2       100
CBEAM    2       1       2       3       100
CBEAM    3       1       3       4       100
CBEAM    4       1       4       5       100
CBEAM    5       1       5       6       100
CBEAM    6       1       6       7       100
CBEAM    7       1       7       8       100
CBEAM    8       1       8       9       100
CBEAM    9       1       9       10      100
CBEAM    10      1       10      11      100
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 28-May-97           Time: 11:45:28
MAT1     1       1.+7            .3
$ Nodes of the Entire Model
GRID     1               0.      0.      0.
GRID     2               1.      0.      0.
GRID     3               2.      0.      0.
GRID     4               3.      0.      0.
GRID     5               4.      0.      0.
GRID     6               5.      0.      0.
GRID     7               6.      0.      0.
```

```
GRID     8               7.      0.      0.
GRID     9               8.00000 0.      0.
GRID     10              9.00000 0.      0.
GRID     11              10.     0.      0.
GRID     100             0.      0.      1.
$ Loads for Load Case : subcase_1
SPCADD   2       10      12
LOAD     2       1.      1.      1
$ Loads for Load Case : subcase_2
LOAD     4       1.      1.      3
$ Loads for Load Case : subcase_3
LOAD     6       1.      1.      5
$ Loads for Load Case : subcase_4
LOAD     8       1.      1.      7
$ Displacement Constraints of Load Set : constraint_1
SPC1     10      123456  1       100
$ Displacement Constraints of Load Set : constraint_2
SPC1     12      345     1       THRU    11
$ Nodal Forces of Load Set : force_1
FORCE    1       11      0       2000.   0.      1.      0.
$ Nodal Forces of Load Set : force_2
FORCE    3       11      0       4000.   0.      1.      0.
$ Nodal Forces of Load Set : force_3
FORCE    5       11      0       6000.   0.      1.      0.
$ Nodal Forces of Load Set : force_4
FORCE    7       11      0       8000.   0.      1.      0.
$ Referenced Coordinate Frames
ENDDATA a7d89e05
```

WORKSHOP PROBLEM 2a

Geometric Linear Analysis of Cantilever Beam

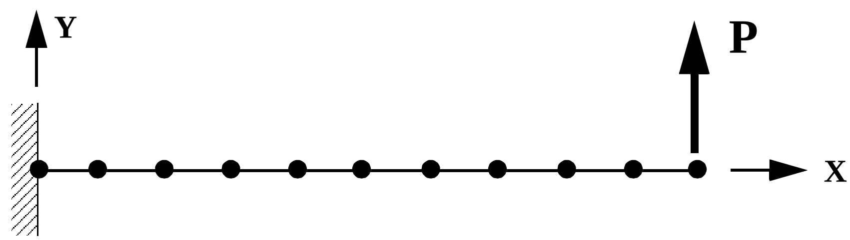


Objectives:

- Demonstrate the use of geometric linear analysis.
- Observe the behavior of the cantilever beam under four increasing load magnitudes.
- Create an accurate deformation plot of the model.
- Create a plot of the load factor vs. displacement.

Model Description:

For the structure below:



L = 10.0
A = 1.0
I = $1x10^{-2}$
E = 10^{6}
ν = 0.3

Add Case Control commands and Bulk Data Entries to:

1. Perform a geometric linear analysis.

2. Determine the behavior of the cantilever beam for the following four load cases:

   1) P = 2000
   2) P = 4000
   3) P = 6000
   4) P = 8000

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate loading conditions and linear static analysis control parameters.
- For Case Control, insert static load set selection (LOAD) in each subcase.
- Prepare the model for a linear static analysis (SOL 101).
- Generate an input file and submit it to the MSC/NASTRAN solver for a linear static analysis.
- Review the results.

Input File for Modification:

prob2a.dat

```
ID NAS103, WORKSHOP 2A
SOL 106
TIME 10
CEND
TITLE=TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM
SUBTITLE=REF.: BISSHOPP AND DRUCKER; QAM 3(1):272-275; 1945
SPC=1
DISP=ALL
OLOAD=ALL
$
SUBCASE 10
LOAD=200
$
SUBCASE 20
LOAD=400
$
SUBCASE 30
LOAD=600
$
SUBCASE 40
LOAD=800
$
BEGIN BULK
$ GEOMETRY
GRID,1,,0.,0.,0.,,345
=,*(1),=,*(1.),==$
=(9)
GRID,100,,0.,0.,1.,,123456
$ CONNECTIVITY
CBEAM,101,1,1,2,100
=,*(1),=,*(1),*(1),==$
=(8)
$ PROPERTIES
PBEAM,1,1,1.,1.-2,1.-2
MAT1,1,10.+6,,.0
$ CONSTRAINTS
SPC,1,1,123456
$ LOADING
FORCE,11,11,,1.+4,0.,1.,0.
```

```
LOAD,200,.2,1.,11
LOAD,400,.4,1.,11
LOAD,600,.6,1.,11
LOAD,800,.8,1.,11
$ 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 13, otherwise, proceed to step 2.

2. Create a new database called **prob2a.db**.

File/New...

New Database Name:	**prob2a**

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, **prob2.ses**. If you choose to build the model yourself, proceed to the step 4.

File/Session/Play...

Session File List:	**prob2.ses**

Apply

The model has now been created. Skip to **Step 12**.

4. Create a 10 unit long beam.

◆ **Geometry**

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

Apply

5. Mesh the curve with ten BAR2 elements.

◆ **Finite Elements**

Action: **Create**

Object: **Mesh**

Type: **Curve**

Global Edge Length: **1**

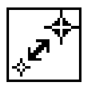
Element Topology: **Bar2**

Curve List: **Curve 1**

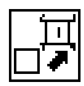
(Select the curve.)

Apply

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



Node Size



Show Labels

6. Create the reference grid for the bar orientation.

◆ **Finite Elements**

Action: **Create**

Object: **Node**

Method: **Edit**

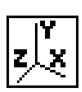
Node ID List **100**

❒ **Associate with Geometry**

Node Location List **[0, 0, 1]**

Apply

For clarity, change the view using the following toolbar icon:



Iso 1 View

7. Create the material property for the beam.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**
Input Properties...	
Elastic Modulus =	**10.E6**
Poisson's Ratio =	**.3**
Apply	
Cancel	

8. Create the property for the beam.

◆ **Properties**

Action:	**Create**	
Object:	**1D**	
Method:	**Beam**	
Property Set Name:	**beam**	
Option(s):	**Tapered Section**	
Input Properties...		
Material Name:	**m:mat_1**	
Bar Orientation:	**Node 100**	**Node ID**
Cross Sect. Areas:	**1.0**	
[Inertias 1,1]:	**1e-2**	
[Inertias 2,2]:	**1e-2**	
OK		
Select Members:	**Curve 1** *(Select the curve.)*	

Add

Apply

9. Create the Loads/BCs for the model.

First, fix the left end of the beam and the orientation nodes.

◆ **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...

Geometry Filter	● **FEM**
Select Nodes	**Node 1, 100**

Add

OK

Apply

Next, constrain all the nodes to planar translation and rotation.

New Set Name	**constraint_2**

Input Data...

Translation < T1 T2 T3 >	**< , , 0 >**
Rotation < R1 R2 R3 >	**< 0, 0, >**

OK

Select Application Region...

Geometry Filter	● **FEM**

Select Nodes	**Node 1:11**
Add	
OK	
Apply	

10. Create the loading for the model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Method:	**Nodal**
New Set Name	**force_1**
Input Data...	
Force <F1 F2 F3>	**<0, 2000, 0>**
OK	
Select Application Region...	
Geometry Filter	● **FEM**
Select Nodes	**Node 11** *(Right end of beam.)*
Add	
OK	
Apply	

Create three more forces.

New Set Name	**force_2**
Input Data...	
Force <F1 F2 F3>	**<0, 4000, 0>**
OK	
Select Application Region...	

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

New Set Name **force_3**

Input Data...

Force <F1 F2 F3> **<0, 6000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

Finally, the last force.

New Set Name **force_4**

Input Data...

Force <F1 F2 F3> **<0, 8000, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes **Node 11**

(Right end of beam.)

Add

OK

Apply

11. Create the four load cases for the four loading conditions.

◆ **Load Cases**

Action: **Create**

Load Case Name: **subcase_1**

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet **Displ_constraint_1**
Displ_constraint_2
Force_force_1

OK

Apply

Be sure that the Loads/BC Scale Factor is 1 for each of the Loads/BCs Types appearing in the spreadsheet.

Load Case Name: **subcase_2**

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet **Displ_constraint_1**
Displ_constraint_2
Force_force_2

OK

Apply

Load Case Name: **subcase_3**

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet **Displ_constraint_1**
Displ_constraint_2
Force_force_3

OK

Apply

Load Case Name **subcase_4**

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet **Displ_constraint_1**
Displ_constraint_2
Force_force_4

OK

Apply

12. 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 analysis as follows:

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name **prob2a**

Solution Type...

Solution Type: ● **LINEAR STATIC**

OK

Subcase Create...

Available Subcases: **subcase_1**

Output Requests...

Form Type: **Advanced**

Output Requests: **STRESS(SORT...**

Delete

Output Requests: **SPCFORCE(SORT...**

Delete

Select Result Type: **Applied Loads**

Create

OK

Apply

Repeat the above procedure to create the second, third, and fourth subcases.

Available Subcases	**subcase_2**
Output Requests...	
Form Type:	**Advanced**
Select Result Type	**Applied Loads**
Output Requests	*(Deselect all but **DISPL(...**)*
Delete	
Create	
OK	
Apply	

Now create the third subcase.

Available Subcases	**subcase_3**
Output Requests...	
Form Type:	**Advanced**
Output Requests	*(Deselect all but **DISPL(...**)*
Delete	
Select Result Type	**Applied Loads**
Create	
OK	
Apply	

Finally, create the fourth subcase.

Available Subcases	**subcase_4**
Output Requests...	
Form Type:	**Advanced**
Output Requests	*(Select all but **DISPL(...**)*
Delete	
Select Result Type	**Applied Loads**
Create	
OK	
Apply	
Cancel	

Lastly, select all the subcases.

Subcase Select...	
Subcases for Solution Sequence	**subcase_1** **subcase_2** **subcase_3** **subcase_4** *(Select one after the other)*
Subcases Selected:	**Default**
OK	
Apply	

An input file called **prob2a.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 14**.

Generating an input file for MSC/NASTRAN Users:

13. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob2a.dat**):

```
ASSIGN OUTPUT2 = ‘prob2a.op2’, UNIT = 12
ID NAS103, WORKSHOP 2A SOLUTION
SOL 101
TIME 10
CEND
TITLE = TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM
SUBTITLE=REF.: BISSHOPP AND DRUCKER; QAM 3(1):272-275; 1945
SPC=1
DISP=ALL
OLOAD=ALL
$
SUBCASE 10
  LOAD = 200
$
SUBCASE 20
  LOAD = 400
$
SUBCASE 30
  LOAD = 600
$
SUBCASE 40
  LOAD = 800
$
BEGIN BULK
$ GEOMETRY
GRID,1,,0.,0.,0.,,345
=,*(1),=,*(1.),==$
=(9)
GRID,100,,0.,0.,1.,,123456
$ CONNECTIVITY
CBEAM,101,1,1,2,100
=,*(1),=,*(1),*(1),==$
=(8)
$ PROPERTIES
PBEAM,1,1,1.,1.-2,1.-2
MAT1,1,10.+6,,.3
```

```
$ CONSTRAINTS
SPC,1,1,123456
$ LOADING
FORCE,11,11,,1.+4,0.,1.,0.
LOAD,200,.2,1.,11
LOAD,400,.4,1.,11
LOAD,600,.6,1.,11
LOAD,800,.8,1.,11
$ PARAMETERS
PARAM,POST,-1
ENDDATA
```

Submit the input file for analysis:

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

 14a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob2a.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

 14b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob2a.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

15. When the analysis is completed, edit the **prob2a.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.

15a. While still editing **prob2a.f06**, search for the word:

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

What is the y-displacement of **Node 11** for the first subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the second subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the third subcase?

T2 = ____________________

What is the y-displacement of **Node 11** for the fourth subcase?

T2 = ____________________

Comparison of Results:

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

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
11	G	0.0	6.671867E+00	0.0	0.0	0.0	1.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	1.334373E+01	0.0	0.0	0.0	2.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	2.001560E+01	0.0	0.0	0.0	3.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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
11	G	0.0	2.668747E+01	0.0	0.0	0.0	4.000000E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

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

18. Proceed with the Reverse Translation process, that is, importing the **prob2a.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	**prob2a.op2**
OK	
Apply	

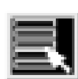
When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

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)	**subcase_4, Static Subcase**
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

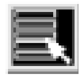
The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action: **Create**

Object: **Deformation**

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

 **Select Results**

Select Result Case(s) **subcase_4, Static Subcase**

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.

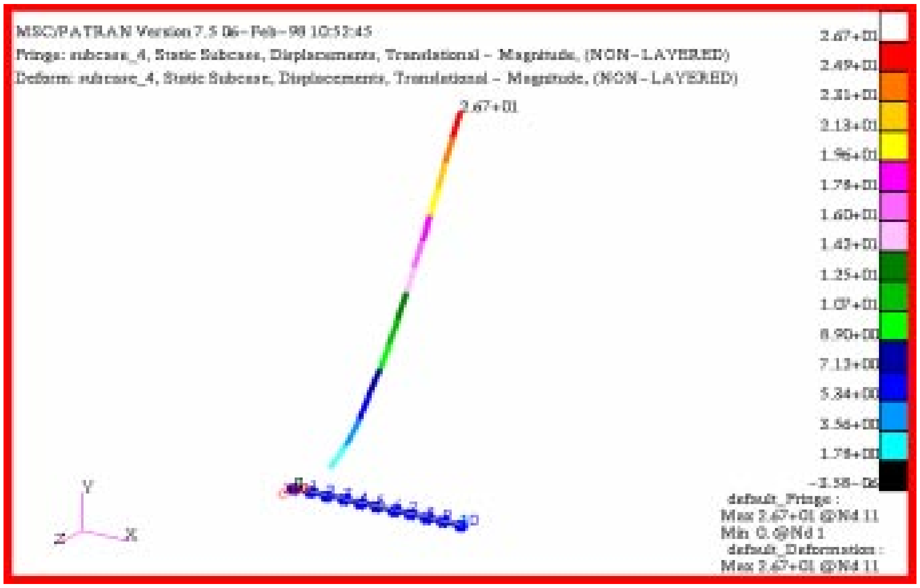
 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

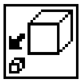
Apply

Your resulting plot should look similar to the following.



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

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

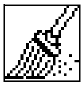


Zoom Out

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

Notice the extremely large deflection of the beam is. This suggests that a nonlinear geometric analysis is necessary in order to obtain a reasonable solution.

Click the Reset Graphics icon to clear the post-processing results and obtain the original model in the viewport.



Reset Graphics

19. 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: **Y Component**

X: **Result**

Select X Result...

Select X Result **Displacements, Translational**

Quantity: **Magnitude**

OK

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

 **Target Entities**

Target Entity: **Nodes**

Select Nodes **Node 11**

(Select node at end of beam.)

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 Loads**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

Apply

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**

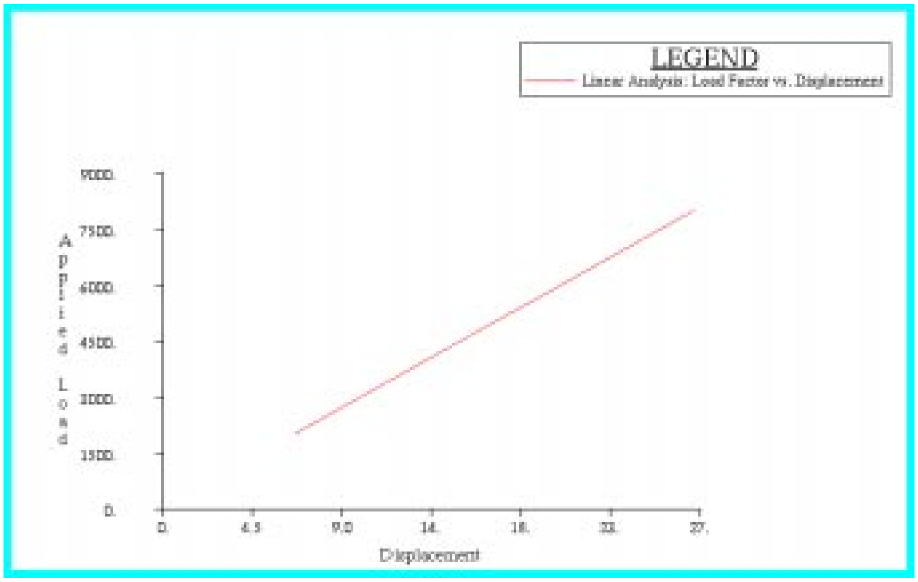
Title...

Curve Title Text **Linear Analysis: Load Factor vs. Displacement**

Apply

Cancel

Your XY Plot should appear as follows:



Notice the linear relationship between the displacement and the load factor.

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

◆ **XY Plot**

Action:	**Post**
Object:	**XYWindow**
Post/Unpost XYWindows	*(<ctrl>, click on **Results Graph** to de-select it.)*

Apply

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob2a.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 13:10:51.
ASSIGN OUTPUT2 = ‘prob2a.op2’, UNIT = 12
$ Direct Text Input for File Management Section
$ Linear Static Analysis, Database
SOL 101
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 15-Jan-98 at 13:07:43
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : subcase_1
  SUBTITLE=subcase_1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : subcase_2
  SUBTITLE=subcase_2
  SPC = 2
  LOAD = 4
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 3
$ Subcase name : subcase_3
  SUBTITLE=subcase_3
  SPC = 2
  LOAD = 6
  DISPLACEMENT(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 4
$ Subcase name : subcase_4
```

```
  SUBTITLE=subcase_4
  SPC = 2
  LOAD = 8
  DISPLACEMENT(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    INREL   0
PARAM    ALTRED  NO
PARAM   COUPMASS -1
PARAM    K6ROT   0.
PARAM    WTMASS  1.
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : beam
PBEAM    1       1       1.      .01     .01                             +      A
+      A                                                         +      B
+      B YES     1.      1.      .01     .01                             +      C
+      C
CBEAM    1       1       1       2       100
CBEAM    2       1       2       3       100
CBEAM    3       1       3       4       100
CBEAM    4       1       4       5       100
CBEAM    5       1       5       6       100
CBEAM    6       1       6       7       100
CBEAM    7       1       7       8       100
CBEAM    8       1       8       9       100
CBEAM    9       1       9       10      100
CBEAM    10      1       10      11      100
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 28-May-97           Time: 11:45:28
MAT1     1       1.+7            .3
$ Nodes of the Entire Model
GRID     1               0.      0.      0.
GRID     2               1.      0.      0.
GRID     3               2.      0.      0.
GRID     4               3.      0.      0.
GRID     5               4.      0.      0.
GRID     6               5.      0.      0.
GRID     7               6.      0.      0.
```

```
GRID     8              7.      0.      0.
GRID     9              8.00000 0.      0.
GRID     10             9.00000 0.      0.
GRID     11             10.     0.      0.
GRID     100            0.      0.      1.
$ Loads for Load Case : subcase_1
SPCADD   2       10      12
LOAD     2       1.      1.      1
$ Loads for Load Case : subcase_2
LOAD     4       1.      1.      3
$ Loads for Load Case : subcase_3
LOAD     6       1.      1.      5
$ Loads for Load Case : subcase_4
LOAD     8       1.      1.      7
$ Displacement Constraints of Load Set : constraint_1
SPC1     10      123456  1       100
$ Displacement Constraints of Load Set : constraint_2
SPC1     12      345     1       THRU    11
$ Nodal Forces of Load Set : force_1
FORCE    1       11      0       2000.   0.      1.      0.
$ Nodal Forces of Load Set : force_2
FORCE    3       11      0       4000.   0.      1.      0.
$ Nodal Forces of Load Set : force_3
FORCE    5       11      0       6000.   0.      1.      0.
$ Nodal Forces of Load Set : force_4
FORCE    7       11      0       8000.   0.      1.      0.
$ Referenced Coordinate Frames
ENDDATA a7d89e05
```