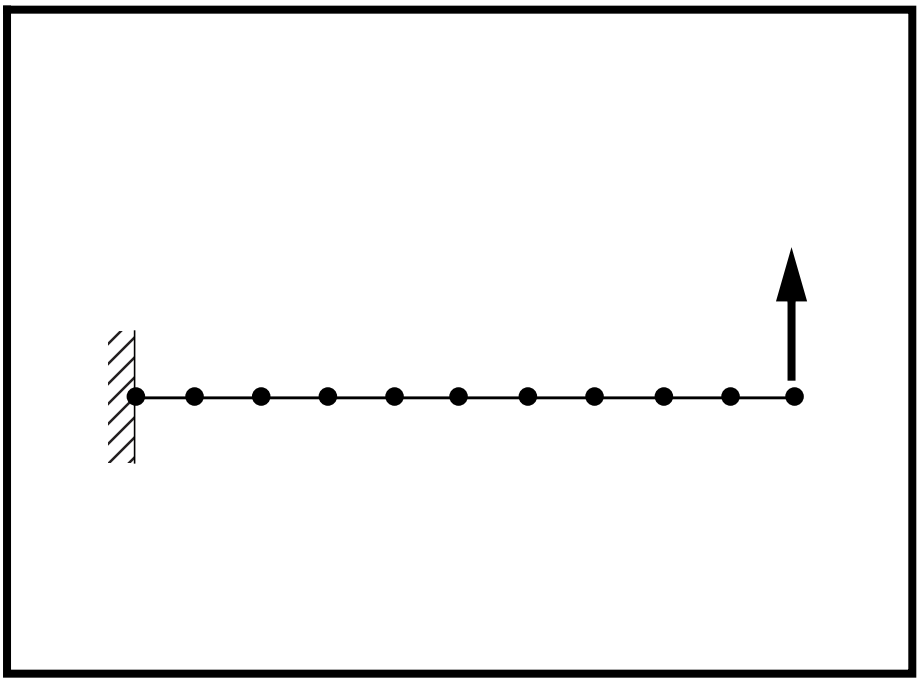
WORKSHOP PROBLEM 2b

Geometric Nonlinear Analysis of Cantilever Beam

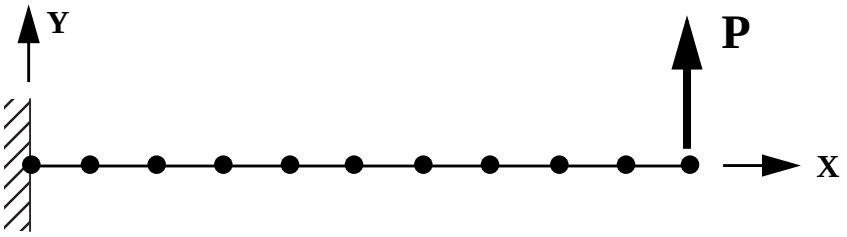


Objectives:

- Demonstrate the use of geometric nonlinear analysis
- Determine the behavior of the cantilever beam under four increasing load magnitudes.
- Create an accurate deformation plot of the model.
- Create a plot of displacement vs. distance for all the subcases.
- 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 nonlinear 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 nonlinear static analysis control parameters.
- Prepare the model for a nonlinear static analysis (SOL 106).
  - PARAM, LGDISP, 1
- Insert all relevant nonlinear static analysis parameters for both case control and subcases (NLPARM).
- Generate an input file and submit it to the MSC/NASTRAN solver for nonlinear static analysis.
- Review the results.

Input File from Workshop 2a for Modification:

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
```

Exercise Procedure:

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

2. Open the existing database called **prob2a.db**.

File/Open...

Database List **prob2a**

OK

3. Change the analysis from linear static to nonlinear static.

First, you will need to define the appropriate nonlinear analysis parameters:

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 **prob2b**

Solution Type...

Solution Type ● **NONLINEAR 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**

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

Delete

Select Result Type **Applied Loads**

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, the fourth subcase.

Available Subcases **subcase_4**

Output Requests...

Form Type: **Advanced**

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

Delete

Select Result Type **Applied Loads**

Create

OK

Apply

Cancel

Finally, 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: *(Deselect **Default**)*

OK

Apply

An input file called **prob2b.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN users should now proceed to **Step 5**.

Generating an input file for MSC/NASTRAN Users:

4. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob2b.dat**):

```
ASSIGN OUTPUT2 = 'prob2b.op2', UNIT = 12
ID NAS103, WORKSHOP 2B SOLUTION
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
NLPARM=10
$
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
PARAM,LGDISP,1
NLPARM,10,10
ENDDATA
```

Submit the input file for analysis:

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

   5a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob2b.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

   5b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob2b.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

6. When the analysis is completed, edit the **prob2b.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

6a. While still editing **prob2b.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:

7. 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	-1.605319E+00	4.941084E+00	0.0	0.0	0.0	7.820106E-01
100	G	0.0	0.0	0.0	0.0	0.0	0.0

1 TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM MAY 28, 1997 MSC/NASTRAN 5

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	-3.288413E+00	6.712009E+00	0.0	0.0	0.0	1.121878E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

1 TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM MAY 28, 1997 MSC/NASTRAN

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	-4.345254E+00	7.462511E+00	0.0	0.0	0.0	1.284553E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

1 TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM MAY 28, 1997 MSC/NASTRAN 7

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	-5.047949E+00	7.870426E+00	0.0	0.0	0.0	1.375393E+00
100	G	0.0	0.0	0.0	0.0	0.0	0.0

1 TRACE LARGE DEFLECTION OF A CANTILEVERED BEAM MAY 28, 1997 MSC/NASTRAN

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

9. Open a new database to import the results into.

First, close the present database.

File/Close

Next, open a new database titled **prob2b.db**

File/New...

New Database Name: **prob2b**

OK

In the **New Model Preference** form set the following:

Tolerance: ● **Default**

Analysis Code: **MSC/NASTRAN**

Analysis Type: **Structural**

OK

10. Proceed with the Reverse Translation process, that is, importing the **prob2b.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action: **Read Output2**

Object: **Both**

Method: **Translate**

Select Results File...

Selected Results File: **prob2b.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, PW Linear...**
Select Fringe Result	**Displacements, Translational**
Quantity:	**Magnitude**

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

 **Target Entities**

Target Entity:	**Current Viewport**

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.)*

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, PW Linear...**

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 .

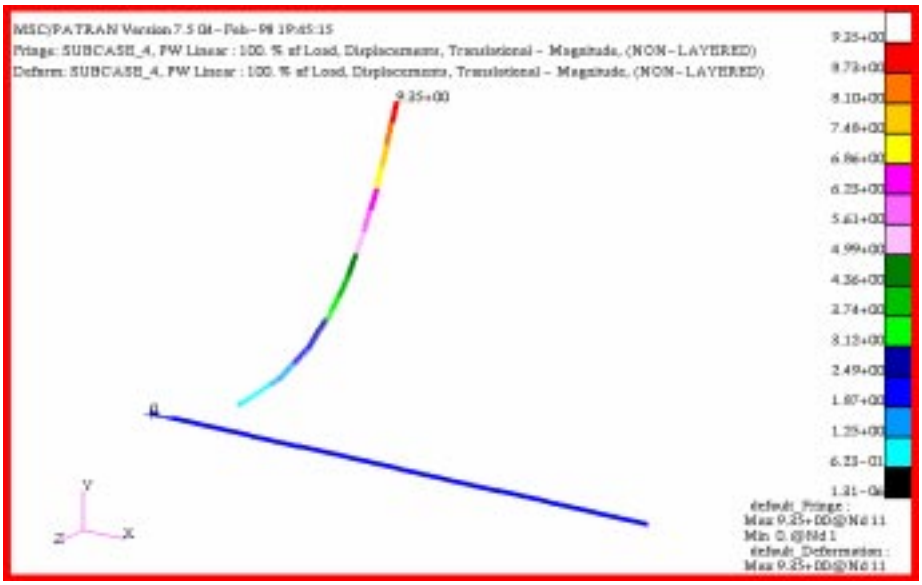
Plot Options

Coordinate Transformation: | **None**

Scale Factor | **1.0**

Apply

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 fringe plot.

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 that the deformation of the beam is much more reasonable. This is a result of using a nonlinear static analysis, which acounts for large displacements.

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

 **Reset Graphics**

11. Create an XY plot of Displacement vs. Distance for all four subcases.

First create a XY plot for the first subcase.

◆ **Results**

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

Select all the Result Cases by highlighting them.

Select Result Case(s)	**subcase_1, PW Linear: 100.% of Load**
Y:	**Result**
Select Y Result	**Displacements, Translational**
Quantity:	**Y Component**
X:	**Coordinate**
Select Coordinate Axis	**Coord 0.1**

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

 **Target Entities**

Target Entity:	**Nodes**
Select Nodes	**Node 1:11** *(Select nodes along length of beam.)*

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Show X Axis Label**

X Axis Label: **Distance**

X Axis Scale ● **Linear**

X Axis Format...

Label Format: **Fixed**

OK

■ **Show Y Axis Label**

Y Axis Label: **Displacements**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

■ **Append Curves in XY Window**

Now click on the **Plot Options** icon

 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

Now change the title in the Legend. Later, change the title of the other subcases by selecting the related graph under Curve List.

◆ **XY Plot**

Action: **Modify**

Object: **Curve**

Curve List **default_GraphResults Graph 0**

Title...

Curve Title Text **Nonlinear Displacement - Subcase 1**

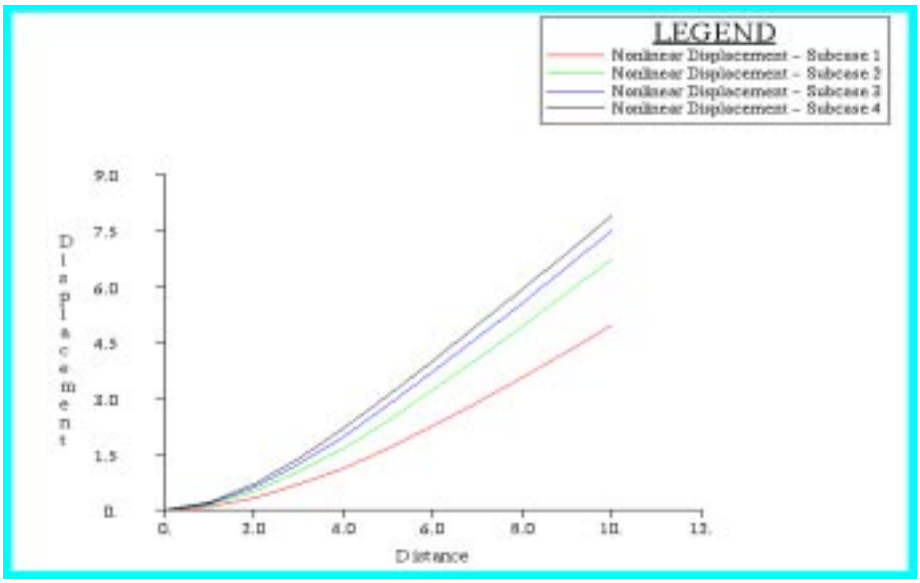
Apply

Cancel

Repeat the above procedure for the remaining subcases with only one difference. Under the **Display Attributes** window in the **Results** form, click *on* the **Append Curves in XY Window** as shown below.

■ **Append Curves in XY Window**

The following XY plot should resemble yours. It contains four curves - one for each of the subcases



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

◆ **XY Plot**

Action: **Post**

Object: **XYWindow**

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

Apply

12. Create an XY plot of Load Factor vs. Displacement.

◆ **Results**

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

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:	**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

❒**Append Curves in XY Window**

Now click on the **Plot Options** icon

 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

Now change the title in the Legend.

◆ **XY Plot**

Action: **Modify**

Object: **Curve**

Curve List **default_GraphResults Graph 0**

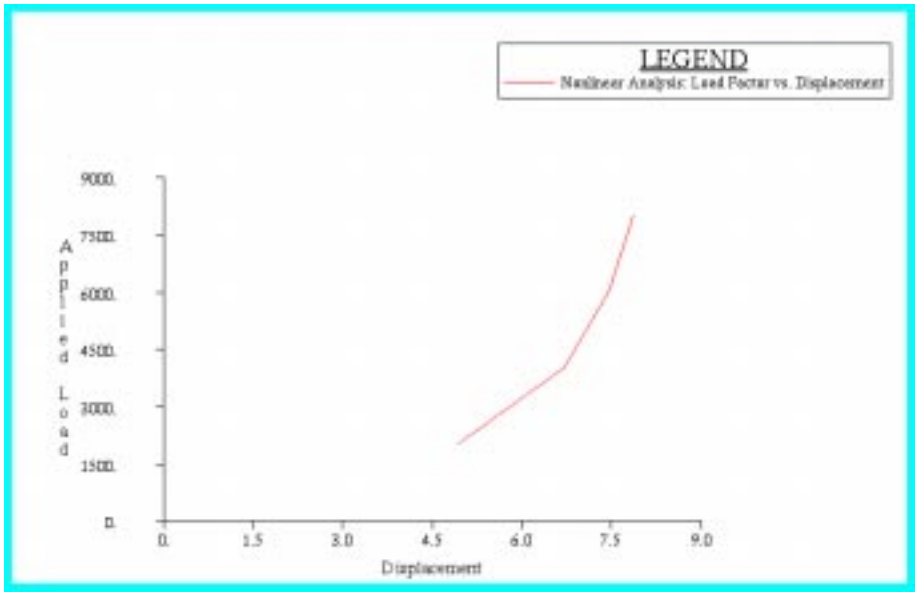
Title...

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

Apply

Cancel

The second XY plot should appear as follows:



Notice that there is no longer a 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 XY Windows:	*(hold* ***<ctrl>*** *click on* **Results Graph** *to deselect it.)*

Apply

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob2b.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 13:21:45.
ASSIGN OUTPUT2 = 'prob2b.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 13:07:43
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : subcase_1
  SUBTITLE=subcase_1
  NLPARM = 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
  NLPARM = 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
  NLPARM = 3
  SPC = 2
  LOAD = 6
  DISPLACEMENT(SORT1,REAL)=ALL
```

```
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 4
$ Subcase name : subcase_4
  SUBTITLE=subcase_4
  NLPARM = 4
  SPC = 2
  LOAD = 8
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,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       10              AUTO    5       25      PW      NO      +       A
+       A       .001    1.-7
NLPARM   2       10              AUTO    5       25      PW      NO      +       B
+       B       .001    1.-7
NLPARM   3       10              AUTO    5       25      PW      NO      +       C
+       C       .001    1.-7
NLPARM   4       10              AUTO    5       25      PW      NO      +       D
+       D       .001    1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : beam
PBEAM    1       1       1.      .01     .01                     +       E
+       E                                               +       F
+       F YES     1.      1.      .01     .01                     +       G
+       G
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 1e925697
```