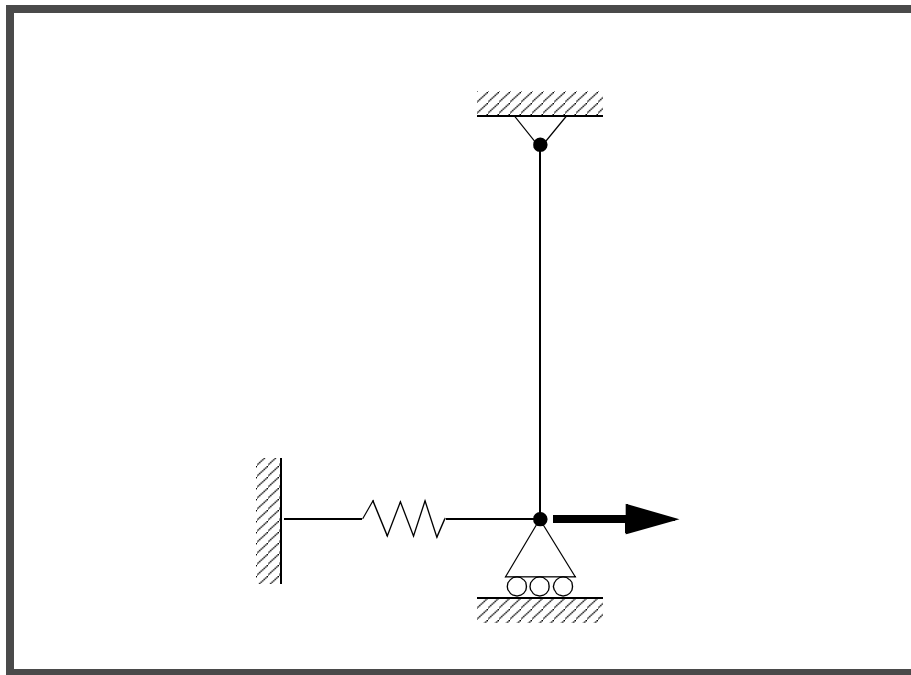

WORKSHOP PROBLEM 1c

Spring Element with Nonlinear Analysis Parameters (Multi-Step Analysis)

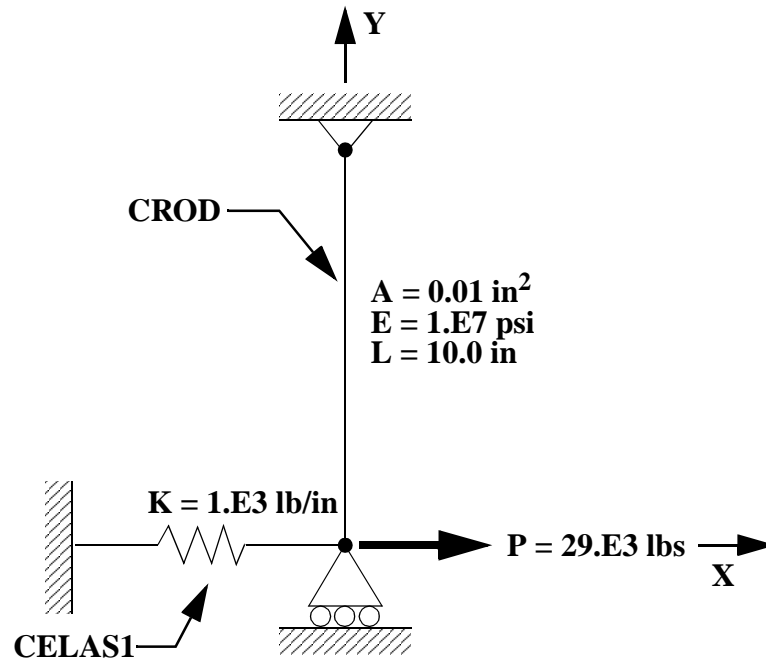


Objectives:

- Demonstrate the effects of geometric nonlinear analysis in SOL 106 (nonlinear statics).
- Apply incremental loads through multiple subcases.
- Demonstrate multiple subcase analysis and output options.

Model Description:

For the structure below:

**Add Case Control commands and Bulk Data Entries to:**

1. Perform a geometric nonlinear analysis in SOL 106.
2. For Subcase 1:
 - ◆ Apply a 16×10^3 lbs load in 4 increments.
 - ◆ Do not use line search, quasi-Newton update or bisections. Print output at every load step.
3. For Subcase 2:
 - ◆ Apply a 24×10^3 lbs load in the second subcase in 8 increments.
 - ◆ Use only the work criteria for convergence, and printoutput at every load step.
4. For Subcase 3:
 - ◆ Apply a 29×10^3 lbs load in 5 increments.
 - ◆ Request output at the end of the subcase only.

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file, by adding the appropriate loading conditions and nonlinear static analysis control parameters.
- Request all constraint forces in the output (SPCF=ALL).
- For Case Control, insert the static load set selection (LOAD) and the nonlinear static analysis parameter selection (NLPARAM) in each subcase.
- For Bulk Data, insert all the relevant nonlinear static analysis parameters for each subcase (NLPARAM).
- Prepare the model for a geometric nonlinear analysis.
 - ◆ 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 from Workshop 1b for Modification:**prob1b.dat**

```
ASSIGN OUTPUT2 = 'prob1b.op2' , UNIT=12
ID NAS103, WORKSHOP 1B 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
NLPARAM=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
```

Exercise Procedure:

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 6, otherwise, proceed to step 2.

2. Open the existing database called **prob1a.db**.

File/Open...

Database List:

prob1a

OK

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

3. Create the additional loads for the model.

For this model, the load will be incremented up to the final load through three subcases (created from three load cases). First, create the first value for the load incrementation.

◆ **Loads/BCs**

Action:

Create

Object:

Force

Method:

Nodal

New Set Name

load_1

Input Data...

Force <F1 F2 F3>

<16.E3, 0, 0>

OK

Select Application Region...

Select Geometry Entities

Point 1

(Select point at bottom of beam.)

Add

OK

Apply

Next, create the second value for the loading.

New Set Name

load_2

Input Data...

Force <F1 F2 F3>

<24.E3, 0, 0>

OK

Select Application Region...

Select Geometry Entities

Point 1

(Select point at bottom of beam.)

Add

OK

Apply

The third value for the loading already exists. It is the one we have been applying for the previous two exercises as the final load.

4. Create the three load cases for the three loading conditions.

◆ Load Cases

Action:

Create

Load Case Name:

case_1

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet:

Displ_constraint_1
Displ_constraint_2
Force_load_1

Make sure that the LBC Scale Factor for selected Loads/BCs in spreadsheet have a value of 1.

OK

Apply

Create the second load case.

Load Case Name:

case_2

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet:

**Displ_constraint_1
Displ_constraint_2
Force_load_2**

OK

Apply

Create the third load case.

Load Case Name:

case_3

Assign/Prioritize Loads/BCs

Remove All Rows

Select Loads/BCs to Add to Spreadsheet:

**Displ_constraint_1
Displ_constraint_2
Force_load_3**

OK

Apply

5. 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

proble

Solution Type...

Solution Type

● **NONLINEAR STATIC**

OK

Subcase Create...

<i>Available Subcases</i>	<input type="text" value="case_1"/>
Subcase Parameters...	
<i>Number of Load Increments</i>	<input type="text" value="4"/>
<i>Matrix Update Method:</i>	<input type="text" value="Semi-Automatic"/>
OK	
Output Requests...	
<i>Form Type:</i>	<input type="text" value="Advanced"/>
<i>Output Requests</i>	<input type="text" value="STRESS(SORT1..."/>
Delete	
<i>Select Result Type</i>	<input type="text" value="Element Forces"/>
Create	
<i>Intermediate Output Option:</i>	<input type="text" value="Yes"/>
OK	
Apply	

For the second subcase, enter the following:

<i>Available Subcases</i>	<input type="text" value="case_2"/>
Subcase Parameters...	
<i>Number of Load Increments</i>	<input type="text" value="8"/>
<i>Matrix Update Method:</i>	<input type="text" value="Automatic"/>
<i>(Turn off Load Error convergence criteria.)</i>	<input type="checkbox"/> Load Error
OK	
Output Requests...	
<i>Form Type:</i>	<input type="text" value="Advanced"/>
<i>Output Requests</i>	<input type="text" value="STRESS(SORT1..."/>
Delete	
<i>Select Result Type</i>	<input type="text" value="Element Forces"/>
Create	
<i>Intermediate Output Option:</i>	<input type="text" value="Yes"/>

OK

Apply

For the third subcase, enter the following:

Available Subcases

case_3

Subcase Parameters...

Number of Load Increments

5

Matrix Update Method:

Automatic

OK

Output Requests...

Form Type:

Advanced

Output Requests

STRESS(SORT1...

Delete

Select Result Type

Element Forces

Create

Intermediate Output Option:

No

OK

Apply

Cancel

Subcase Select...

Subcases for Solution Sequence

**case_1
case_2
case_3**

Subcases Selected

(Deselect Default.)

OK

Apply

An input file called **prob1c.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.

☞ **NOTE: YOU MUST EDIT THE INPUT FILE BEFORE SUBMITTING IT TO MSC/NASTRAN!!! DO NOT RUN ANALYSIS YET!!!**

6. Edit the input file to de-activate quasi-Newton updates, line searches, and bisections.

Enter a text editor and make the following changes to the input file:

In the Bulk Data section, look for the **NLPARM** entry referenced by subcase 1 (**NLPARM 1**) and edit it so it looks like the following:

```
NLPARM 1      4          SEMI  5      25      PW      YES  +   A
+   A          .001  1.-7          0      0          +   Z
+   Z  0
```

☞ **NOTE:** Be sure to maintain the column format!!!

Exit the text editor when you have made these changes.

MSC/PATRAN users should now proceed to **Step 8**.

Generating an input file for MSC/NASTRAN Users:

7. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob1c.dat**):

```
ASSIGN OUTPUT2 = 'prob1c.op2' , UNIT=12
ID NAS103, WORKSHOP 1C 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$
SPCF=ALL
$ APPLY X LOAD
$
SUBCASE 10 $ LOAD=16.E03
LABEL=APPLY LOAD P IN X DIRECTION = 16E+03
LOAD=1
NLPARAM=10
SUBCASE 20 $ LOAD=24.E03
LABEL=APPLY LOAD P IN X DIRECTION = 24E+03
LOAD=2
NLPARAM=20
SUBCASE 30 $ LOAD=29.E03
LABEL=APPLY LOAD P IN X DIRECTION = 29E+03
LOAD=3
NLPARAM=30
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, , SEMI, , , , YES, +
+, , , , 0, 0, , , +
+, 0
NLPARM, 20, 8, , AUTO, , , W, YES
NLPARM, 30, 2
ENDDATA

Submitting the input file for analysis:

8. Submit the input file to MSC/NASTRAN for analysis.
 - 8a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran problc.bdf scr=no**. Monitor the analysis using the UNIX **ps** command.
 - 8b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran problc.dat**. Monitor the analysis using the UNIX **ps** command.
9. When the analysis is completed, edit the **problc.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.
 - 9a. While still editing **problc.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 first step?

T1 = _____

What is the x-displacement of the guided end at the end of the second step?

T1 = _____

What is the x-displacement of the guided end at the end of the last step?

T1 = _____

Comparison of Results:

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

```

0
LOAD STEP = 1.00000E+00
SUBCASE 1
DISPLACEMENT VECTOR
POINT ID. TYPE T1 T2 T3 R1 R2 R3
1 G 6.300765E+00 0.0 0.0 0.0 0.0 0.0
2 G 0.0 0.0 0.0 0.0 0.0 0.0
    
```

```

0
LOAD STEP = 2.00000E+00
SUBCASE
DISPLACEMENT VECTOR
POINT ID. TYPE T1 T2 T3 R1 R2 R3
1 G 7.751148E+00 0.0 0.0 0.0 0.0 0.0
2 G 0.0 0.0 0.0 0.0 0.0 0.0
    
```

```

0
LOAD STEP = 3.00000E+00
SUBCASE
DISPLACEMENT VECTOR
POINT ID. TYPE T1 T2 T3 R1 R2 R3
1 G 8.540173E+00 0.0 0.0 0.0 0.0 0.0
2 G 0.0 0.0 0.0 0.0 0.0 0.0
    
```

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

12. Proceed with the Reverse Translation process, that is, importing the **prob1c.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	Read Output2
<i>Object:</i>	Result Entities
<i>Method:</i>	Translate
Select Results File...	
<i>Selected Results File:</i>	prob1c.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**

<i>Action:</i>	Create
<i>Object:</i>	Fringe

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



Select Results

<i>Select Result Case(s)</i>	case_1, PW Linear: 100.% of Load
<i>Select Fringe Result</i>	Displacements, Translational
<i>Quantity:</i>	Magnitude

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

Now create the deformation plot.

◆ Results

Action:

Create

Object:

Deformation

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

**Select Results**

Select Result Case(s)

case_1, 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.

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



Zoom Out

Create other fringe plots by picking the last increment of each load case (when % of load=100) for your Result Case..

For each of the load cases, you can see the physical deformation of the model as well as the amount of deformation from the fringe.

Notice that the deformation at the end of the last step is identical to the result obtained when directly applying the final load with a nonlinear geometric analysis (previous exercise).

To clear the post-processing results and obtain the original model

in the viewport, select the **Reset Graphics** icon.



Reset Graphics

13. Create an XY plot of Element Force vs Displacement.

◆ **Analysis**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Graph"/>
<i>Method:</i>	<input type="text" value="Y vs X"/>

Select all the Result Cases by highlighting them except for the first two.

<i>Select Result Case(s)</i>	<input type="text" value="case_1, PW Linear, 25.% of Load"/>
	<input type="text" value="..."/>

(Highlight from the above on down.)

<i>Y:</i>	<input type="text" value="Result"/>
-----------	-------------------------------------

<i>Select Y Result</i>	<input type="text" value="Constraint Forces, Translational"/>
------------------------	---

<i>Quantity:</i>	<input type="text" value="Magnitude"/>
------------------	--

<i>X:</i>	<input type="text" value="Result"/>
-----------	-------------------------------------

<input type="text" value="Select X Result..."/>

<i>Select X Result</i>	<input type="text" value="Displacements, Translational"/>
------------------------	---

<i>Quantity:</i>	<input type="text" value="X Component"/>
------------------	--

<input type="text" value="OK"/>

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



Target Entities

<i>Target Entity:</i>	<input type="text" value="Nodes"/>
-----------------------	------------------------------------

Select Nodes

Node 1

(Select node at bottom of beam.)

Click on the **Display Attributes** icon.



Display Attributes

■ **Show X Axis Label**

X Axis Label:

X Axis Scale

X Axis Format...

Label Format:

OK

■ **Show Y Axis Label**

Y Axis Label:

Y Axis Scale

Y Axis Format...

Label Format:

OK

Apply

Displacements

● **Linear**

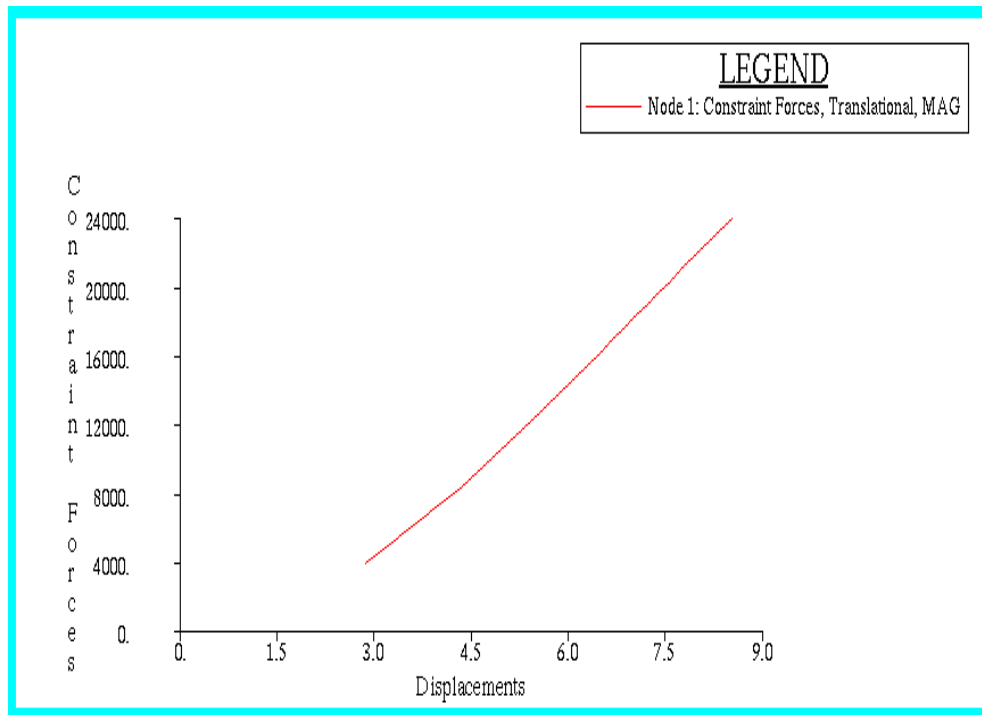
Fixed

Constraint Forces

● **Linear**

Fixed

The following XY plot should appear on your screen:



When you are 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: prob1c.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January 16, 1998 at
$ 09:25:15.
ASSIGN OUTPUT2 = 'prob1c.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 : case_1
  SUBTITLE=case_1
  NLPARAM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  FORCE(SORT1,REAL,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_2
  SUBTITLE=case_2
  NLPARAM = 2
  SPC = 2
  LOAD = 4
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  FORCE(SORT1,REAL,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 3
$ Subcase name : case_3
  SUBTITLE=case_3
  NLPARAM = 3
  SPC = 2
  LOAD = 6
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(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 SEMI 5 25 PW YES + A
+ A .001 1.-7 0 0 + Z
+ Z 0
NLPARM 2 8 AUTO 5 25 W YES + B
+ B 1.-7
NLPARM 3 5 AUTO 5 25 PW NO + C
+ C .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 : case_1
SPCADD 2 7 9
LOAD 2 1. 1. 1
$ Loads for Load Case : case_2
LOAD 4 1. 1. 3
$ Loads for Load Case : case_3
LOAD 6 1. 1. 5
$ Displacement Constraints of Load Set : constraint_1
SPC1 7 123456 2
$ Displacement Constraints of Load Set : constraint_2

```

```
SPC1 9 23456 1
$ Nodal Forces of Load Set : load_3
FORCE 5 1 0 29000. 1. 0. 0.
$ Nodal Forces of Load Set : load_1
FORCE 1 1 0 16000. 1. 0. 0.
$ Nodal Forces of Load Set : load_2
FORCE 3 1 0 24000. 1. 0. 0.
$ Referenced Coordinate Frames
ENDDATA cdcf0464
```