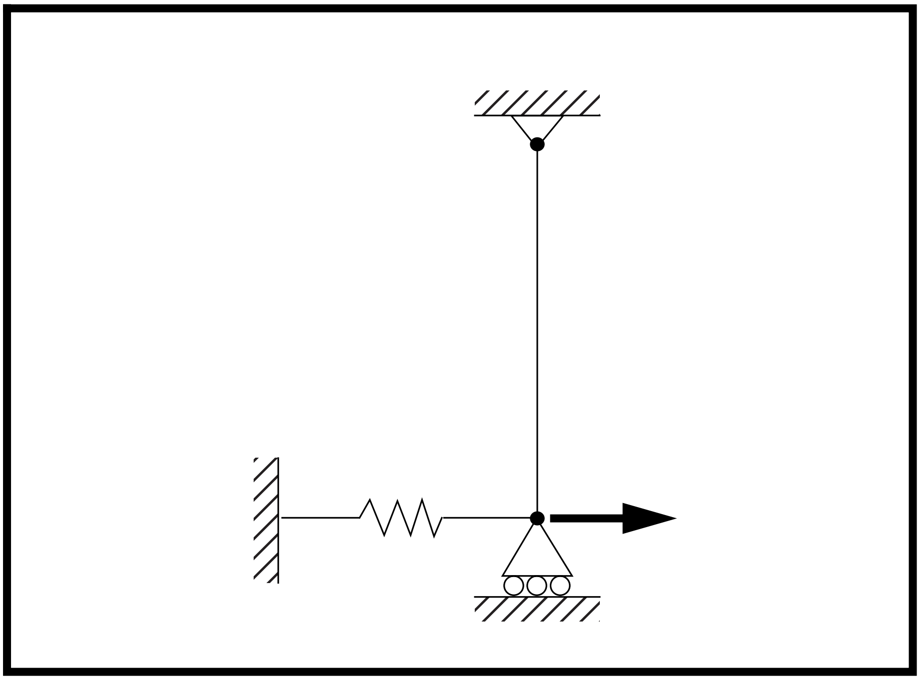
WORKSHOP PROBLEM 1c

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

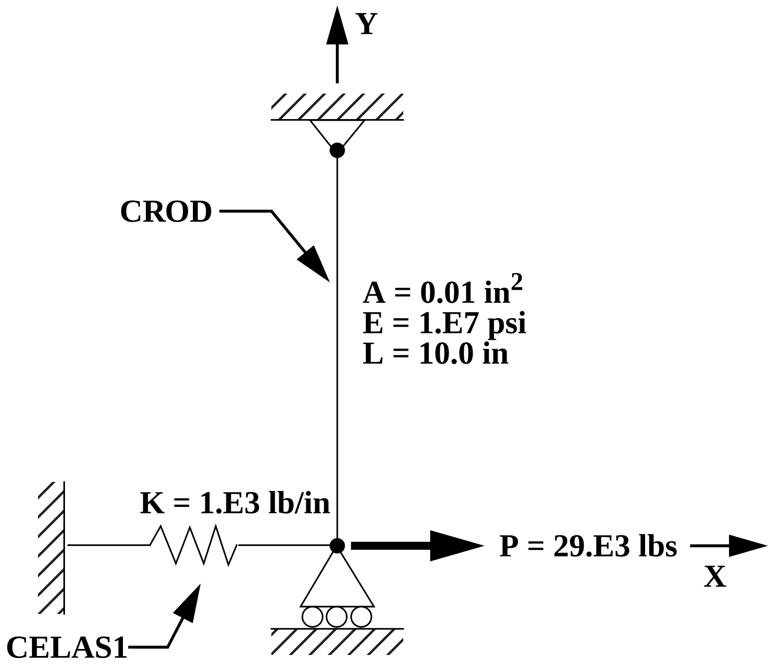


Objectives:

- 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 $16x10^3$ 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 $24x10^3$ 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 $29x10^3$ 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 (NLPARM) in each subcase.
- For Bulk Data, insert all the relevant nonlinear static analysis parameters for each subcase (NLPARM).
- 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
 NLPARM=10
OUTPUT(PLOT)
  SET 1 ALL
  MAXI DEFO 5.
  AXES Z, X, Y
  VIEW  0., 0., 0.
  FIND SCALE ORIGIN 1 SET 1
  PLOT STATIC 0 MAXIMUM DEFORMATION 5. SET 1
BEGIN BULK
PARAM, POST, -1
PARAM, PATVER, 3.0
GRID, 1, 0, 0.0, 0.0, 0.0,  , 23456
GRID, 3, 0, 0.0, 10.0, 0.0,  , 123456
CROD, 3, 3, 3, 1
CELAS1, 2, 2, 1, 1, 0
PROD, 3, 3, .01
PELAS, 2, 1.0E3
```

```
MAT1, 3, 1.0E7
FORCE, 1, 1, 0, 1.6E4, 1.0
FORCE, 2, 1, 0, 2.4E4, 1.0
FORCE, 3, 1, 0, 2.9E4, 1.0
PARAM, LGDISP,1
NLPARM, 10, 4
ENDDATA
```

Exercise Procedure:

1. Users who are not utilitizing 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 **prob1c**

Solution Type...

Solution Type ● **NONLINEAR STATIC**

OK

Subcase Create...

Available Subcases	**case_1**
Subcase Parameters...	
Number of Load Increments	**4**
Matrix Update Method:	**Semi-Automatic**
OK	
Output Requests...	
Form Type:	**Advanced**
Output Requests	**STRESS(SORT1...**
Delete	
Select Result Type	**Element Forces**
Create	
Intermediate Output Option:	**Yes**
OK	
Apply	

For the second subcase, enter the following:

Available Subcases	**case_2**
Subcase Parameters...	
Number of Load Increments	**8**
Matrix Update Method:	**Automatic**
(Turn off Load Error convergence criteria.)	❐ **Load Error**
OK	
Output Requests...	
Form Type:	**Advanced**
Output Requests	**STRESS(SORT1...**
Delete	
Select Result Type	**Element Forces**
Create	
Intermediate Output Option:	**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
 NLPARM=10
SUBCASE 20 $ LOAD=24.E03
 LABEL=APPLY LOAD P IN X DIRECTION = 24E+03
 LOAD=2
 NLPARM=20
SUBCASE 30 $ LOAD=29.E03
 LABEL=APPLY LOAD P IN X DIRECTION = 29E+03
 LOAD=3
 NLPARM=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 prob1c.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 prob1c.dat**. Monitor the analysis using the UNIX **ps** command.

9. When the analysis is completed, edit the **prob1c.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 **prob1c.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 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	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 SUBCASE
2
LOAD STEP = 2.00000E+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	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 SUBCASE
3
LOAD STEP = 3.00000E+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	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**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File:	**prob1c.op2**
OK	
Apply	

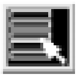
When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

Now we will generate the fringe plot of the model.

◆ **Results**

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

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**
Quantity:	**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**

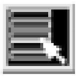
Apply

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**

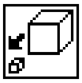
Coordinate Transformation: **None**

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:

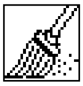
 **Zoom Out**

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**

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

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

Select Result Case(s)	**case_1, PW Linear, 25.% of Load ...** *(Highlight from the above on down.)*
Y:	**Result**
Select Y Result	**Constraint Forces, Translational**
Quantity:	**Magnitude**
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 bottom 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: — **Constraint Forces**

Y Axis Scale — ● **Linear**

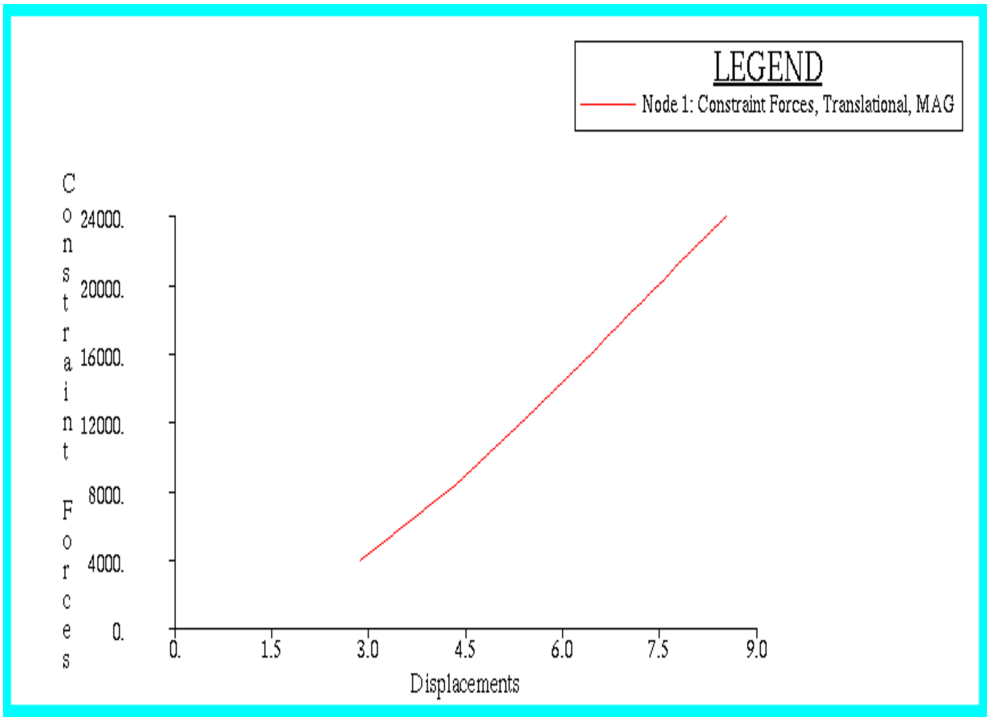
Y Axis Format...

Label Format: — **Fixed**

OK

Apply

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
  NLPARM = 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
  NLPARM = 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
  NLPARM = 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
```