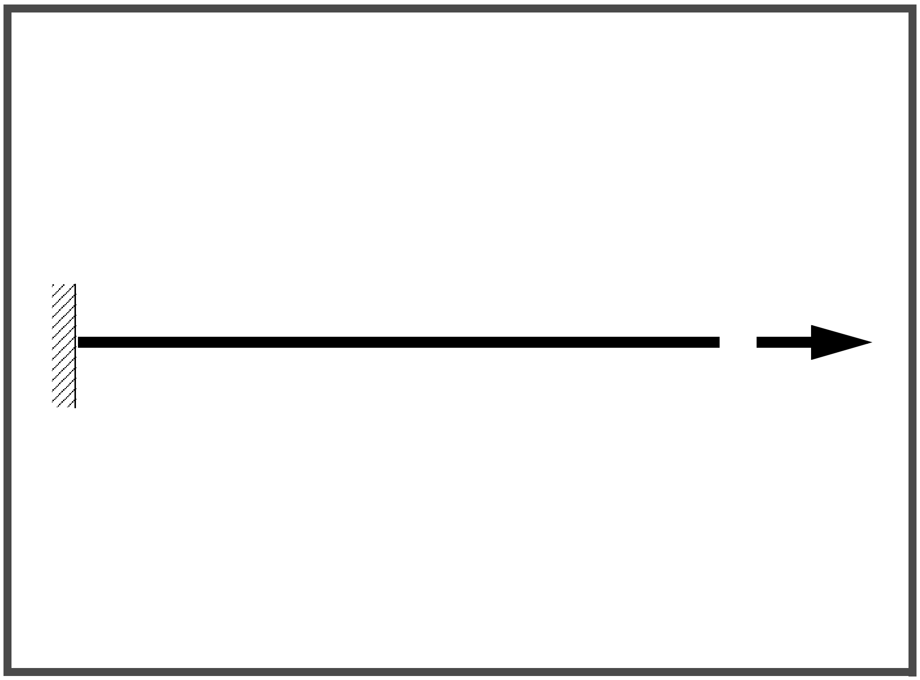
WORKSHOP PROBLEM 7

Nonlinear Creep Analysis

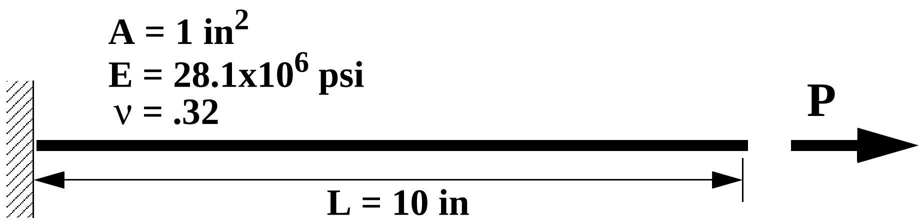


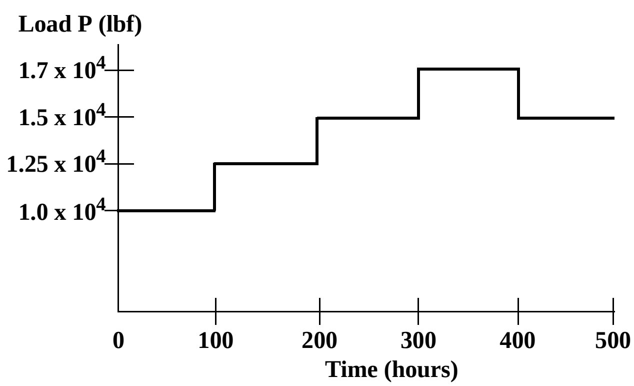
Objectives:

- Demonstrate the use of creep material properties.
- Examine the strain for each subcase.
- Create an XY plot of Load vs. Displacement for all the subcases.

Model Description:

For the structure below:





<u>Creep Strain:</u>

$\varepsilon_c = f(\sigma)[1-e^{-r(\sigma)t}] + g(\sigma)t$

where

$f(\sigma) = 3.476 \times 10^{-4} \exp(0.000208\sigma)$

$r(\sigma) = 3.991 \times 10^{-5} (\sigma/1000)^{2.094}$

$g(\sigma) = 1.02 \times 10^{-11} \exp(0.000743\sigma)$

Add Case Control commands and Bulk Data Entries to:

1. Model the creep behavior of the material.
2. Analyze the model subjected to the given load history.

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).
- Set up the appropriate subcase loading and analysis parameters (LOAD, NLPARM)
- Input the proper creep material property for the nonlinear material (CREEP)
- Generate an input file and submit it to the MSC/NASTRAN solver for nonlinear static analysis.
- Review the results.

Input File for Modification:

prob7.dat

```
ID NAS103, WORKSHOP 7
TIME 30
SOL 106
CEND
TITLE=SIMPLE ONE DOF CREEP PROBLEM
LABEL=REF: FOSTER WHEELER REPORT FWR-27, MARCH 1972, P. A-6
ECHO=SORT
DISP=ALL
OLOAD=ALL
STRESS=ALL
SUBCASE 10 $ ELASTIC
  NLPARM=10
SUBCASE 11 $ CREEP
  NLPARM=20
SUBCASE 20 $ ELASTIC
  NLPARM=10
SUBCASE 21 $ CREEP
  NLPARM=20
SUBCASE 30 $ ELASTIC
  NLPARM=10
SUBCASE 31 $ CREEP
  NLPARM=20
SUBCASE 40 $ ELASTIC
  NLPARM=10
SUBCASE 41 $ CREEP
  NLPARM=20
SUBCASE 50 $ ELASTIC
  NLPARM=10
SUBCASE 51 $ CREEP
  NLPARM=20
BEGIN BULK
$
$ GEOMETRY
$
GRID, 1,  , 0., 0., 0.,  , 123456
GRID, 2,  , 10., 0., 0.,  , 23456
$
$ CONNECTIVITY
$
```

```
CROD, 10, 10, 1, 2
$
$ PROPERTIES
$
PROD, 10, 1, 1.
MAT1, 1, 21.8+6,  , .32
$
$ LOADING
$
FORCE, 1, 2,  , 1.+4, 1., 0., 0.
FORCE, 2, 2,  , 1.25+4, 1., 0., 0.
FORCE, 3, 2,  , 1.5+4, 1., 0., 0.
FORCE, 4, 2,  , 1.7+4, 1., 0., 0.
FORCE, 5, 2,  , 1.5+4, 1., 0., 0.
$
$ SOLUTION STRATEGY
$
NLPARM, 10, 1
$
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 **prob7.db**.

File/New...

New Database Name:	**prob7**
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, **prob7.ses**. If you choose to build the model yourself, proceed to the step 4.

File/Session/Play...

Session File List	**prob7.ses**
Apply	

The model has now been created. Skip to **Step 9**.

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

4. Create the geometry of the model.

◆ **Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**

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

Apply

5. Mesh the model.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length	**10**
Element Topology:	**Bar2**
Curve List	**Curve 1**
	(Select the curve.)

Apply

6. Create the boundary conditions for the model.

Create the first constraint for the model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**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...

Select Geometric Entities *(Select point on left.)*

Add

OK

Apply

Create the second model constraint.

New Set Name **constraint_2**

Input Data...

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

Rotation < R1 R2 R3 > **< 0, 0, 0 >**

OK

Select Application Region...

Select Geometric Entities (*Select point on right.)*

Add

OK

Apply

7. Create the loading for the model.

Create the first load as follows:

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name: **load_1**

Input Data...

Force <F1 F2 F3> **<1.0e4, 0, 0>**

OK

Select Application Region...

Select Geometric Entities (*Select point on right.)*

Add

OK

Apply

Now create the second load.

New Set Name: **load_2**

Input Data...

Force <F1 F2 F3> **<1.25e4, 0, 0>**

OK

Select Application Region...

Select Geometric Entities *(Select point on the right.)*

Add

OK

Apply

Create the remaining three loads following the same procedure. The table below summarizes the remaining load attributes:

Name	*Force*	*Apply To*
load_3	**<1.5e4, 0, 0>**	*point on right*
load_4	**<1.7e4, 0, 0>**	*point on right*
load_5	**<1.5e4, 0, 0>**	*point on right*

8. Create the load cases for the model.

◆ **Load Cases**

Action: **Create**

Load Case Name: **case_1**

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet: **Displ_constraint_1**
Displ_constraint_2
Force_load_1

OK

Apply

Load Case Name: **case_2**

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet:	**Force_load_2**
	*(Select row containing **Force_load_1**)*
Remove Selected Row	
OK	
Apply	
Load Case Name:	**case_3**
Assign/Prioritize Loads/BCs	
Select Loads/BCs to Add to Spreadsheet:	**Force_load_3**
	*(Select row containing **Force_load_2**)*
Remove Selected Row	
OK	
Apply	
Load Case Name:	**case_4**
Assign/Prioritize Loads/BCs	
Select Loads/BCs to Add to Spreadsheet:	**Force_load_4**
	*(Select row containing **Force_load_3**)*
Remove Selected Row	
OK	
Apply	
Load Case Name:	**case_5**
Assign/Prioritize Loads/BCs	
Select Loads/BCs to Add to Spreadsheet:	**Force_load_5**
	*(Select row containing **Force_load_4**)*
Remove Selected Row	
OK	
Apply	

This is where the session file ends.

9. Create the material for the model.

First, create the linear elastic properties of the material.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**
Input Properties...	
Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**21.8e6**
Poisson's Ratio:	**.32**
Apply	

Next, define the creep properties of the material.

Constitutive Model:	**Creep**
Creep Data Input:	**Creep Law 222**
Coefficient A:	**3.476e-4**
Coefficient B:	**2.08e-4**
Coefficient C:	**2.085e-11**
Coefficient D:	**2.094**
Coefficient E:	**1.02e-11**
Coefficient F:	**7.43e-4**
Apply	
Cancel	

10. Create the element property for the bar.

◆ **Properties**

Action:	**Create**

Dimension:	**1D**
Type:	**Rod**
Property Set Name:	**bar**
Input Properties...	
Material Name:	**m:mat_1**
Area:	**1**
OK	
Select Members:	**Curve 1** *(Select the curve.)*
Add	
Apply	

11. Generate an input file for the 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	**prob7**
Solution Type...	
Solution Type:	● **NONLINEAR STATIC**
OK	
Subcase Create...	
Available Subcases:	**case_1**
Subcase Parameters...	
Number of Load Increments:	**1**
OK	

Output Requests...

Form Type: **Basic**

Output Requests: **SPCFORCES(SORT1...**

Delete

Select Result Type: **Applied Loads**

OK

Apply

Repeat the procedure to create the second subcase.

Available Subcases: **case_2**

Subcase Parameters...

Number of Load Increments: **1**

OK

Output Requests...

Output Requests: **SPCFORCES(SORT1...**

Delete

Select Result Type: **Applied Loads**

OK

Apply

Now create the third subcase.

Available Subcases: **case_3**

Subcase Parameters...

Number of Load Increments **1**

OK

Output Requests...

Output Requests **SPCFORCES(SORT1...**

Delete

Select Result Type **Applied Loads**

OK

Apply

Now create the fourth.

Available Subcases:	**case_4**
Subcase Parameters...	
Number of Load Increments	**1**
OK	
Output Requests...	
Output Request	**SPCFORCES(SORT1...**
Delete	
Select Result Type	**Applied Loads**
OK	
Apply	

Finally the fifth subcase.

Available Subcases:	**case_5**
Subcase Parameters...	
Number of Load Increments:	**1**
OK	
Output Requests...	
Output Requests:	**SPCFORCES(SORT1...**
Delete	
Select Result Type	**Applied Loads**
OK	
Apply	

Follow the following procedure to create 5 additional subcases which mirror the output requests of the first 5.

Available Subcases	**case_1**
Subcase Name	**case_1b**
Direct Text Input...	
(enter in blank area)	**NLPARM = 99**
OK	
Apply	
Available Subcases	**case_2**
Subcase Name	**case_2b**
Direct Text Input...	
(enter in blank area)	**NLPARM = 99**
OK	
Apply	
Available Subcases	**case_3**
Subcase Name	**case_3b**
Direct Text Input...	
(enter in blank area)	**NLPARM = 99**
OK	
Apply	
Available Subcases	**case_4**
Subcase Name	**case_4b**
Direct Text Input...	
(enter in blank area)	**NLPARM = 99**
OK	
Apply	
Available Subcases	**case_5**

Subcase Name **case_5b**

Direct Text Input...

(enter in blank area) **NLPARM = 99**

OK

Apply

Cancel

Input the following text in the Bulk Data section:

Direct Text Input...

● **Bulk Data Section**

(enter in blank area) **NLPARM,99,5,20., , , , ,YES**

OK

Finally, select all the subcases before submitting the analysis.

Subcase Select...

Subcases for Solution Sequence:

case_1, case_1b
case_2, case_2b
case_3, case_3b
case_4, case_4b
case_5, case_5b

Subcases Selected: *(Deselect* ***Default)***

OK

Apply

An input file called **prob7.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 FOR AN ANALYSIS!!!**

12. Edit the input file and remove all the NLPARM entries generated for the even numbered subcases.

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

In the subcases section, delete all lines that say

NLPARM = *[even #]*

In the Bulk Data section, delete all lines that say

NLPARM *[even #]* **1** **...**

☞ NOTE: Be sure to delete the continuation lines as well !!!

Save the file and exit the text editor when you have made these changes.

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 (**prob7.dat**):

```
ASSIGN OUTPUT2 = ‘prob7.op2’, UNIT = 12
ID NAS103, WORKSHOP 7 SOLUTION
TIME 30
SOL 106
CEND
TITLE=SIMPLE ONE DOF CREEP PROBLEM
LABEL=REF: FOSTER WHEELER REPORT FWR-27, MARCH 1972, P. A-6
ECHO=SORT
DISP=ALL
OLOAD=ALL
STRESS=ALL
SUBCASE 10 $ ELASTIC
  LOAD=1
  NLPARM=10
SUBCASE 11 $ CREEP
  LOAD=1
  NLPARM=20
SUBCASE 20 $ ELASTIC
  LOAD=2
  NLPARM=10
SUBCASE 21 $ CREEP
  LOAD=2
  NLPARM=20
SUBCASE 30 $ ELASTIC
  LOAD=3
  NLPARM=10
SUBCASE 31 $ CREEP
  LOAD=3
  NLPARM=20
SUBCASE 40 $ ELASTIC
  LOAD=4
  NLPARM=10
SUBCASE 41 $ CREEP
  LOAD=4
  NLPARM=20
```

```
SUBCASE 50 $ ELASTIC
  LOAD=5
  NLPARM=10
SUBCASE 51 $ CREEP
  LOAD=5
  NLPARM=20
BEGIN BULK
$
$ GEOMETRY
$
GRID, 1,  , 0., 0., 0.,  , 123456
GRID, 2,  , 10., 0., 0.,  , 23456
$
$ CONNECTIVITY
$
CROD, 10, 10, 1, 2
$
$ PROPERTIES
$
PROD, 10, 1, 1.
MAT1, 1, 21.8+6,  , .32
CREEP, 1,  ,  , CRLAW,  ,  ,  , 1.-9,
+, 222, 3.476-4, 2.08-4, 2.085-11, 2.094, 1.02-11, 7.43-4
$
$ LOADING
$
FORCE, 1, 2,  , 1.+4, 1., 0., 0.
FORCE, 2, 2,  , 1.25+4, 1., 0., 0.
FORCE, 3, 2,  , 1.5+4, 1., 0., 0.
FORCE, 4, 2,  , 1.7+4, 1., 0., 0.
FORCE, 5, 2,  , 1.5+4, 1., 0., 0.
$
$ SOLUTION STRATEGY
$
NLPARM, 10, 1
NLPARM, 20, 5, 20.,  ,  ,  ,  , YES
$
ENDDATA
```

Submit the input file for analysis:

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

    14a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob7.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 prob7.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

15. When the analysis is completed, edit the **prob7.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether the existing WARNING messages indicate any modeling errors.

15a. While still editing **prob7.f06**, search for the word:

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

What is the x-displacement of Node 2 for the first subcase?

T1= ____________________

What is the x-displacement of Node 2 for the second subcase?

T1 = ____________________

What is the x-displacement of Node 2 for the third subcase?

T1= ____________________

What is the x-displacement of Node 2 for the fourth subcase?

T1 = ____________________

What is the x-displacement of Node 2 for the final subcase?

T1 = ____________________

Comparison of Results:

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

```
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
        2      G      4.587156E-03   0.0            0.0            0.0            0.0            0.0


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
        2      G      1.548405E-02   0.0            0.0            0.0            0.0            0.0


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
        2      G      1.663084E-02   0.0            0.0            0.0            0.0            0.0


LOAD STEP =  4.00000E+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
        2      G      3.639811E-02   0.0            0.0            0.0            0.0            0.0

   .
   .
   .
   .

LOAD STEP =  1.00000E+01
                                        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
        2      G      9.911608E-02   0.0            0.0            0.0            0.0            0.0
```

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

18. Proceed with the Reverse Translation process, that is, importing the **prob7a.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:	**prob7.op2**
OK	
Apply	

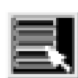
19. When the translation is complete bring up the **Results** form.

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)	*(Select the first case.)*
Select Fringe Result	**Displacements, Translational**
Quantity:	**Magnitude**

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

 **Target Entities**

Target Entity:	**Current Viewport**

Note: This feature allows you to view fringe plots of specific elements of your choice.

Click on the **Display Attributes** icon.

 **Display Attributes**

Style:	**Discrete/Smooth**
Display:	**Free Edges**

For better visual quality of the fringe plot, change the width of the line.

Width:	*(Select the third line from top.)*

Note: The **Display Attributes** form allows you the ability to change the displayed graphics of fringe plots.

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

 **Plot Options**

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

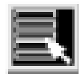
Apply

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

◆ **Results**

Action:	**Create**
Object:	**Deformation**

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

 **Select Results**

Select Result Case(s)	*(Select the first case.)*
Select Fringe Result	**Displacements, Translational**
Show As:	**Resultant**

Click on the **Display Attributes** icon.

 **Display Attributes**

■ *Show Undeformed*

Line Width:	*(Select the third line from top.)*

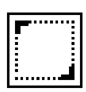
Now click on the **Plot Options** icon .

 **Plot Options**

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

Apply

Use the **View Corners** icon to zoom on to the tip.

 **View Corners**

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

Repeat this process for the other load cases.

20. Create an XY plot of Load vs Displacement for all the subcases.Create an XY plot of Element Force vs Displacement.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**
Select Result Case(s)	*(Select all the results from the first four cases*
Y:	**Result**
Select Y Result	**Applied Loads, Translational**
Quantity:	**X Component**
X:	**Result**

Select X Result...

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

OK

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

 **Target Entities**

Target Entity:	**Nodes**
Select Nodes	**Node 2** *(Select node on the right.)*

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Show X Axis Label**

X Axis Label: **Displacement**

X Axis Scale ● **Linear**

X Axis Format...

Label Format: **Fixed**

OK

■ **Show Y Axis Label**

Y Axis Label: **Applied Load**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

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

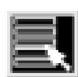
 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

Apply

Now click on the **Select Results** icon and create a second curve.

 **Select Results**

Action: **Create**

Object: **Graph**

Method: **Y vs X**

Select Result Case(s) *(Select the last result of Case 4 and all of the results of Case 5*

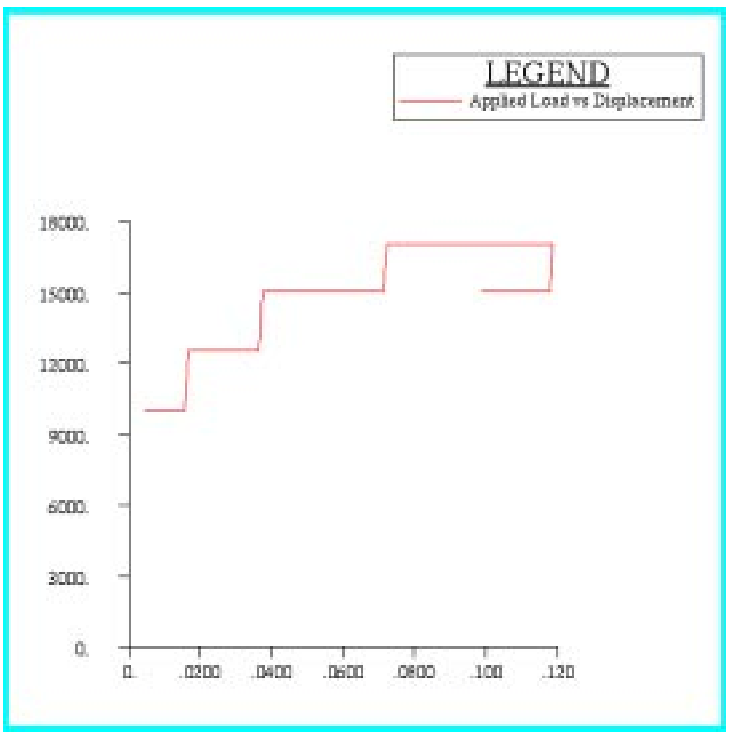
Apply

To change the title, do the following:

◆ **XY Plot**

Action:	**Modify**
Object:	**Curve**
Curve List	**default_GraphResults Graph 0**
Title...	
Curve Title Text	**Applied Load vs. Displacement**
Apply	
Cancel	

You should now see an XY plot which appears like the following:



The nearly vertical lines represent the initial static deformation. The horizontal lines represent creep deformation.

At the end of the load cycle where the load has been decreased, the displacement @ node 2 also decreases which illustrate a creep relaxation.

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob7.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 18:01:01.
ASSIGN OUTPUT2 = 'prob7.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 17:34:04
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
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_1b
  SUBTITLE=case_1
  NLPARM = 2
  SPC = 2
  LOAD = 4
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
NLPARM=99
SUBCASE 3
$ Subcase name : case_2
  SUBTITLE=case_2
  NLPARM = 3
```

```
  SPC = 2
  LOAD = 6
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 4
$ Subcase name : case_2b
  SUBTITLE=case_2
  NLPARM = 4
  SPC = 2
  LOAD = 8
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
NLPARM=99
SUBCASE 5
$ Subcase name : case_3
  SUBTITLE=case_3
  NLPARM = 5
  SPC = 2
  LOAD = 10
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 6
$ Subcase name : case_3b
  SUBTITLE=case_3
  NLPARM = 6
  SPC = 2
  LOAD = 12
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
NLPARM=99
SUBCASE 7
$ Subcase name : case_4
  SUBTITLE=case_4
  NLPARM = 7
  SPC = 2
  LOAD = 14
  DISPLACEMENT(SORT1,REAL)=ALL
```

```
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 8
$ Subcase name : case_4b
  SUBTITLE=case_4
  NLPARM = 8
  SPC = 2
  LOAD = 16
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
NLPARM=99
SUBCASE 9
$ Subcase name : case_5
  SUBTITLE=case_5
  NLPARM = 9
  SPC = 2
  LOAD = 18
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 10
$ Subcase name : case_5b
  SUBTITLE=case_5
  NLPARM = 10
  SPC = 2
  LOAD = 20
  DISPLACEMENT(SORT1,REAL)=ALL
  OLOAD(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
NLPARM=99
BEGIN BULK
PARAM    POST    -1
PARAM    PATVER  3.
PARAM    AUTOSPC YES
PARAM   COUPMASS -1
PARAM    K6ROT   100.
PARAM    WTMASS  1.
PARAM    LGDISP  1
```

```
PARAM,NOCOMPS,-1
PARAM   PRTMAXIM YES
NLPARM  1    1        AUTO   5    25   PW   NO   +   A
+   A      .001   1.-7
NLPARM  3    1        AUTO   5    25   PW   NO   +   C
+   C      .001   1.-7
NLPARM  5    1        AUTO   5    25   PW   NO   +   E
+   E      .001   1.-7
NLPARM  7    1        AUTO   5    25   PW   NO   +   G
+   G      .001   1.-7
NLPARM  9    1        AUTO   5    25   PW   NO   +   I
+   I      .001   1.-7
$ Direct Text Input for Bulk Data
NLPARM,99,5,20.,,,,,YES
$ Elements and Element Properties for region : bar
PROD    1    1    1.
CROD    1    1    1    2
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 15-Jan-98         Time: 17:30:47
CREEP   1              CRLAW                     +   K
*   K 222        3.476-4      2.08-4       2.085-11      *   L
*   L 2.094      1.02-11      7.43-4
MAT1    1    2.18+7       .32
$ Nodes of the Entire Model
GRID    1          0.    0.    0.
GRID    2          10.   0.    0.
$ Loads for Load Case : case_1
SPCADD  2    28    30
LOAD    2    1.    1.    3
$ Loads for Load Case : case_1
LOAD    4    1.    1.    3
$ Loads for Load Case : case_2
LOAD    6    1.    1.    7
$ Loads for Load Case : case_2
LOAD    8    1.    1.    7
$ Loads for Load Case : case_3
LOAD    10   1.    1.    11
$ Loads for Load Case : case_3
LOAD    12   1.    1.    11
$ Loads for Load Case : case_4
LOAD    14   1.    1.    15
$ Loads for Load Case : case_4
LOAD    16   1.    1.    15
$ Loads for Load Case : case_5
```

```
LOAD     18      1.      1.      19
$ Loads for Load Case : case_5
LOAD     20      1.      1.      19
$ Displacement Constraints of Load Set : constraint_1
SPC1     28      123456  1
$ Displacement Constraints of Load Set : constraint_2
SPC1     30      23456   2
$ Nodal Forces of Load Set : load_1
FORCE    3       2       0       10000.  1.      0.      0.
$ Nodal Forces of Load Set : load_2
FORCE    7       2       0       12500.  1.      0.      0.
$ Nodal Forces of Load Set : load_3
FORCE    11      2       0       15000.  1.      0.      0.
$ Nodal Forces of Load Set : load_4
FORCE    15      2       0       17000.  1.      0.      0.
$ Nodal Forces of Load Set : load_5
FORCE    19      2       0       15000.  1.      0.      0.
$ Referenced Coordinate Frames
ENDDATA b204d01b
```