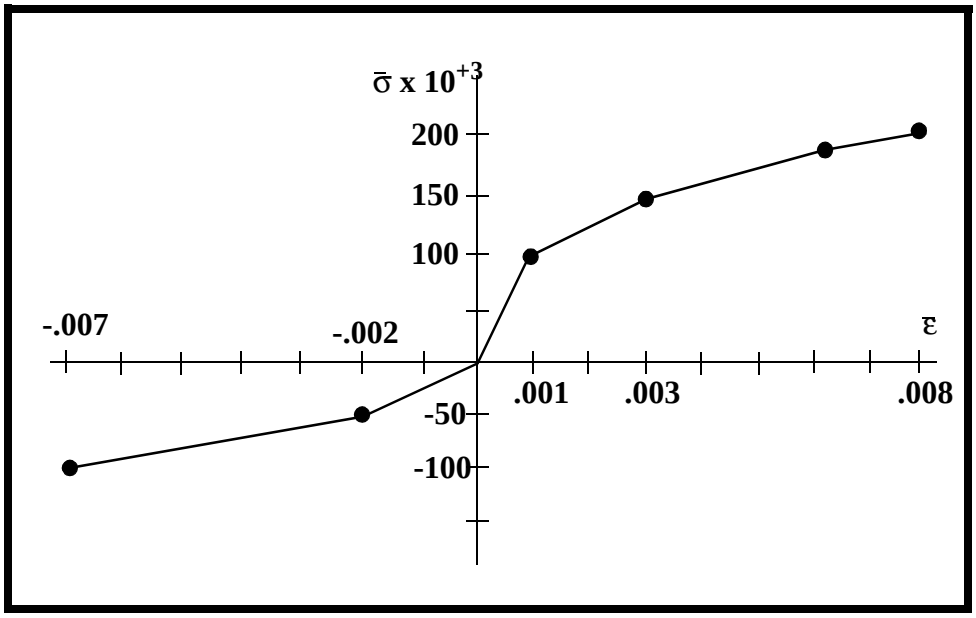
WORKSHOP 15

Nonlinear Elastic Material Properties



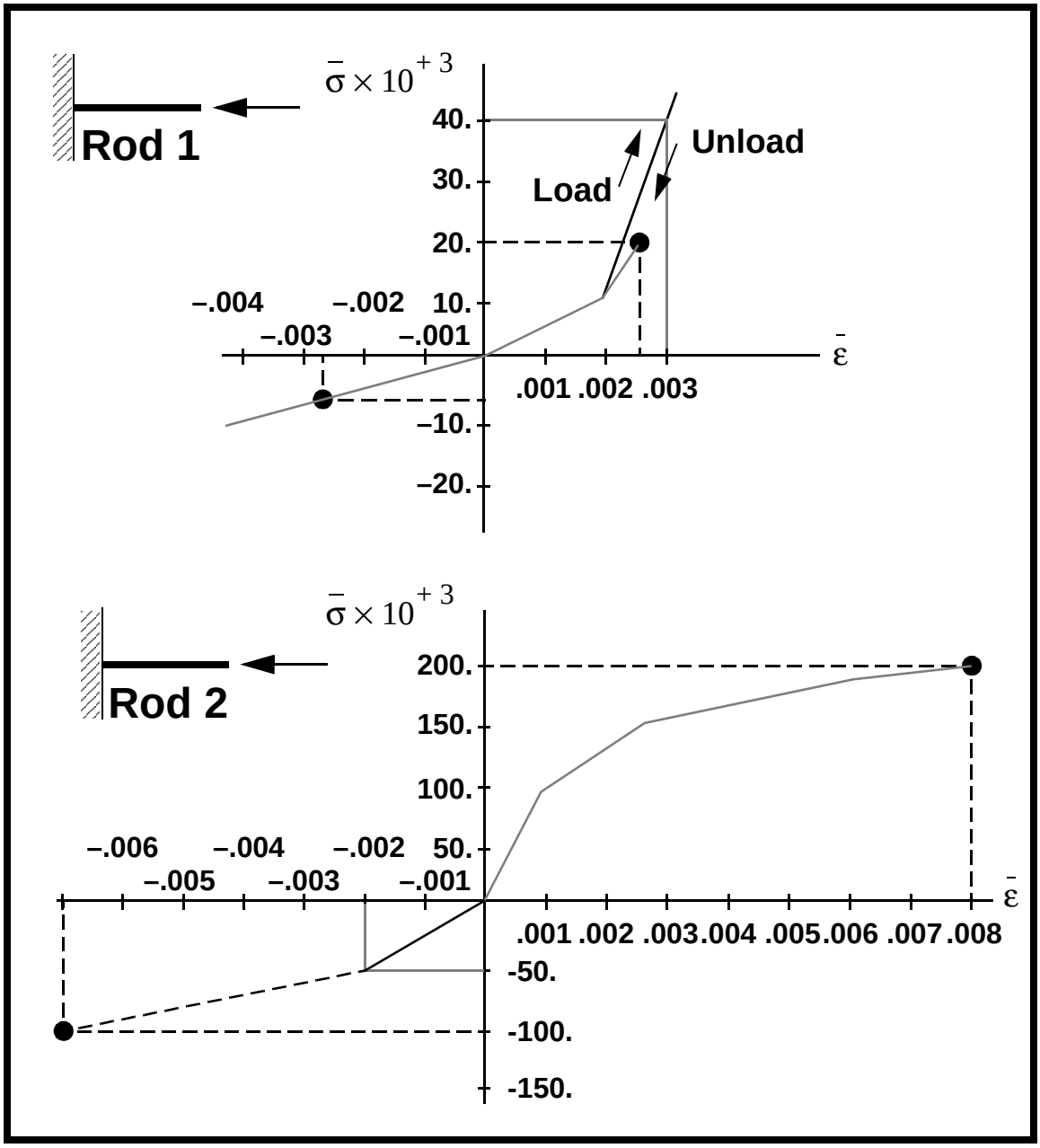
Objectives:

- Demonstrate the use of nonlinear elastic material properties
- Create two nonlinear elastic materials.
- Create two rods, each having one of the material properties.
- Apply appropriate loads and boundary conditions to the rods.
- Run a nonlinear analysis.

Model Description:

Generate the input file to specify the nonlinear elastic material properties for the two rods as shown below.

Figure 15.1



Nonlinear Elastic Material 1:

Young’s Modulus = 5.E6

$\bar{\sigma}$ x 10^3	$\bar{\varepsilon}$
-10E3	-0.004
0	0
10E3	0.002
40E3	0.003

Nonlinear Elastic Material 2:

Young’s Modulus = 100.E6

$\bar{\sigma}$ x 10^3	$\bar{\varepsilon}$
-75E3	-0.0045
-50E3	-0.002
0	0
100E3	0.001
150E3	0.003
175E3	0.0055

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Create functions to represent the nonlinear elastic material properties for the two rods.

Model/Function...

Title:	**nonlinear_1**
Type:	**4..vs Stress**

X:	**-10e3**	*Y:*	**-.004**	**More**
X:	**0**	*Y:*	**0**	**More**
X:	**10e3**	*Y:*	**.002**	**More**
X:	**40e3**	*Y:*	**.003**	**OK**

Title:	**nonlinear_2**
Type:	**4..vs Stress**

X:	**-75e3**	*Y:*	**-.0045**	**More**
X:	**-50e3**	*Y:*	**-.002**	**More**
X:	**0**	*Y:*	**0**	**More**
X:	**100e3**	*Y:*	**.001**	**More**
X:	**150e3**	*Y:*	**.003**	**More**
X:	**175e3**	*Y:*	**.0055**	**OK**

Cancel

3. Now, create the materials.

Model/Material...

Title: **mat_1**

Youngs Modulus: **5e6**

Nonlinear > >

Nonlinearity Type: ● **Nonlinear-Elastic**

Function Dependence **1.. nonlinear_1**

OK

OK

Title: **mat_2**

Youngs Modulus: **100e6**

Nonlinear > >

Nonlinearity Type: ● **Nonlinear-Elastic**

Function Dependence **2.. nonlinear_2**

OK

OK

Cancel

4. Create the rod properties.

Model/Property...

Title: **rod_1**

Material: **1..mat_1**

Elem/Property Type...

● **Rod**

OK

Area: **1**

OK

Title: **rod_2**

Material: **2..mat_2**

Area: **1**

OK

Cancel

5. Create the relevant NASTRAN geometry.

Create the first node of the model by doing the following:

Model/Node...

X:	**0**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**100**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**5**	*Z:*	**0**	**OK**
X:	**100**	*Y:*	**5**	*Z:*	**0**	**OK**

Cancel

To fit the display onto the screen, use the autoscale feature.

View/Autoscale...

Now, connect the nodes to create the rod elements.

Model/Element...

Property: **1..rod_1**

Nodes: **1**

2

OK

Property: **2..rod_2**

Nodes: **3**

4

OK

Cancel

6. Create the relevant constraints.

Model/Constraint/Set...

Title: **constraint_1**

OK

Model/Constraint/Nodal...

Select **Nodes 1 & 3**.

OK

Fixed

OK

Select **Nodes 2 & 4**.

OK

☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

7. Create the load set.

Model/Load/Set...

Title: **load_1**

OK

8. Define the options for a nonlinear analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Basic / Number of Increments: **5**

Stiffness Updates/ Method: **1..AUTO**

OK

9. Create the point loads.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force**.

FX ☒ **20e3**

OK

Now select **Node 4**.

OK

FX ☒ **200e3**

OK

Cancel

10. Create the second load set.

Model/Load/Set...

ID: **2**

Title: **load_2**

OK

11. Define the options for a nonlinear analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Defaults...

Basic / Number of Increments: **5**

Stiffness Updates/ Method: **1..AUTO**

OK

12. Create the second set of point loads.

Model/Load/Nodal...

Select **Node 2**.

OK

Highlight **Force.**

FX ☒ **-6.667e3**

OK

Now select **Node 4**.

OK

FX ☒ **-100e3**

OK

Cancel

Redraw the viewport by selecting:

View/Redraw

13. Submit the job for analysis.

File/Export/Analysis Model...

Type: **10..Nonlinear Static**

OK

Change the directory to **C:\temp**.

File Name: **prob15**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Nonlinear Elastic Sample Problem**

OK

Under *Output Requests*, unselect the following:

☐ **Applied Load**

☐ **Element Force**

Also, change output to:

2..Print and PostProcess

Under *Analysis Case Requests*, enter the following:

Subcase ID: **1**

☒ **Loads =** **1..load_1**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

☒ **Loads =** **2..load_2**

OK

When you get confirmation that the subcase was written, click **OK**.

OK

OK

When asked if you wish to save the model, respond **Yes**.

Yes

File Name: **prob15**

Save

The following is the NASTRAN input file created by FEMAP.

```
INIT MASTER(S)
ID Untitled,MSC/N
SOL NLSTATIC
TIME 10000
CEND
SUBCASE 1
  ECHO = NONE
  DISPLACEMENT = ALL
  SPCFORCE = ALL
  STRESS(CORNER) = ALL
  SPC  = 1
  LOAD = 1
  NLPARM = 1
SUBCASE 2
  DISPLACEMENT = ALL
  SPCFORCE = ALL
  STRESS(CORNER) = ALL
  SPC  = 1
  LOAD = 2
  NLPARM = 2
BEGIN BULK
$ ***************************************************************************
$   Written by : MSC/NASTRAN for Windows
$   Version    : 6.00
$   Translator : MSC/NASTRAN
$   From Model :
$   Date       : Thu Jul 01 18:40:16 1999
$   Output To  : C:\TEMP\manguy\n4w_files\nonlin_mat
$ ***************************************************************************
$
NLPARM         1       5            AUTO       5      25       PW      NO+
+          0.001   0.001   1.E-7       3      25       4     0.2     0.5+
+              5                           20.             20.
NLPARM         2       5            AUTO       5      25       PW      NO+
+          0.001   0.001   1.E-7       3      25       4     0.2     0.5+
+              5                           20.             20.
PARAM,LGDISP,1
PARAM,POST,-1
PARAM,OGEOM,NO
PARAM,AUTOSPC,YES
PARAM,K6ROT,100.
PARAM,MAXRATIO,1.E+8
PARAM,GRDPNT,0
CORD2C         1       0      0.      0.      0.      0.      0.      1.+MSC/NC1
+MSC/NC1      1.      0.      1.
CORD2S         2       0      0.      0.      0.      0.      0.      1.+MSC/NC2
+MSC/NC2      1.      0.      1.
$ MSC/NASTRAN for Windows Load Set 1 : load_1
FORCE          1       2       0      1.  20000.      0.      0.
FORCE          1       4       0      1. 200000.      0.      0.
$ MSC/NASTRAN for Windows Load Set 2 : load_2
FORCE          2       2       0      1.  -6667.      0.      0.
FORCE          2       4       0      1.-100000.      0.      0.
$ MSC/NASTRAN for Windows Constraint Set 1 : constraint_1
SPC            1       1  123456      0.
SPC            1       2   23456      0.
SPC            1       3  123456      0.
SPC            1       4   23456      0.
$ MSC/NASTRAN for Windows Property 1 : rod_1
PROD           1       1      1.      0.      0.      0.
$ MSC/NASTRAN for Windows Property 2 : rod_2
PROD           2       2      1.      0.      0.      0.
$ MSC/NASTRAN for Windows Material 1 : mat_1
MATS1          1       1 NLELAST
$ MSC/NASTRAN for Windows Function 1 : nonlinear_1
TABLES1        1                                                       +
+         -4.E-3 -10000.      0.      0.   0.002  10000.   0.003  40000.+
+       ENDT
MAT1           15000000.                       0.      0.      0.
$ MSC/NASTRAN for Windows Material 2 : mat_2
MATS1          2       2 NLELAST
$ MSC/NASTRAN for Windows Function 2 : nonlinear_2
TABLES1        2                                                       +
+        -4.5E-3 -75000.  -2.E-3 -50000.      0.      0.   0.001 100000.+
+          0.003 150000.  0.0055 175000.ENDT
MAT1           2   1.E+8                       0.      0.      0.
GRID           1       0      0.      0.      0.       0
GRID           2       0    100.      0.      0.       0
GRID           3       0      0.      5.      0.       0
GRID           4       0    100.      5.      0.       0
CROD           1       1       1       2
CROD           2       2       3       4
ENDDATA f02211c3
```

When the MSC.Nastran manager is through running, MSC.Nastran will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

14. List the results of the analysis.

To list the results, select the following:

List/Output/Unformatted...

Select All

OK

Unselect **All Vectors** and instead select **T1 Translation**.

☐ **All Vectors, or**

2..T1 Translation

OK

NOTE: You may want to expand the message box in order to view the results.

Answer the following questions using the results. The answers are listed at the end of the exercise.

What is the maximum displacement in the X direction for each load step?

Step 1 Max X Disp @ Node 2 = _____
Step 1 Max X Disp @ Node 4 = _____
Step 2 Max X Disp @ Node 2 = _____
Step 2 Max X Disp @ Node 4 = _____

This concludes the exercise.

Step 1 Max X Disp @ Node 2:	**0.233**
Step 1 Max X Disp @ Node 4:	**0.8**
Step 2 Max X Disp @ Node 2:	**-0.2667**
Step 2 Max X Disp @ Node 4:	**-0.7**