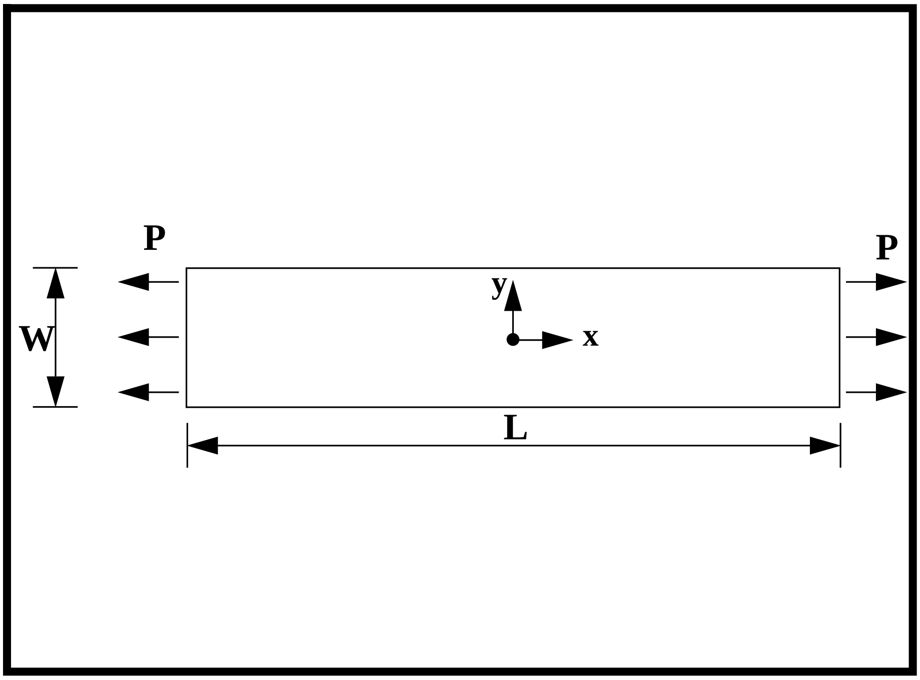
WORKSHOP PROBLEM 6

Elasto-Plastic Deformation of a Thin Plate

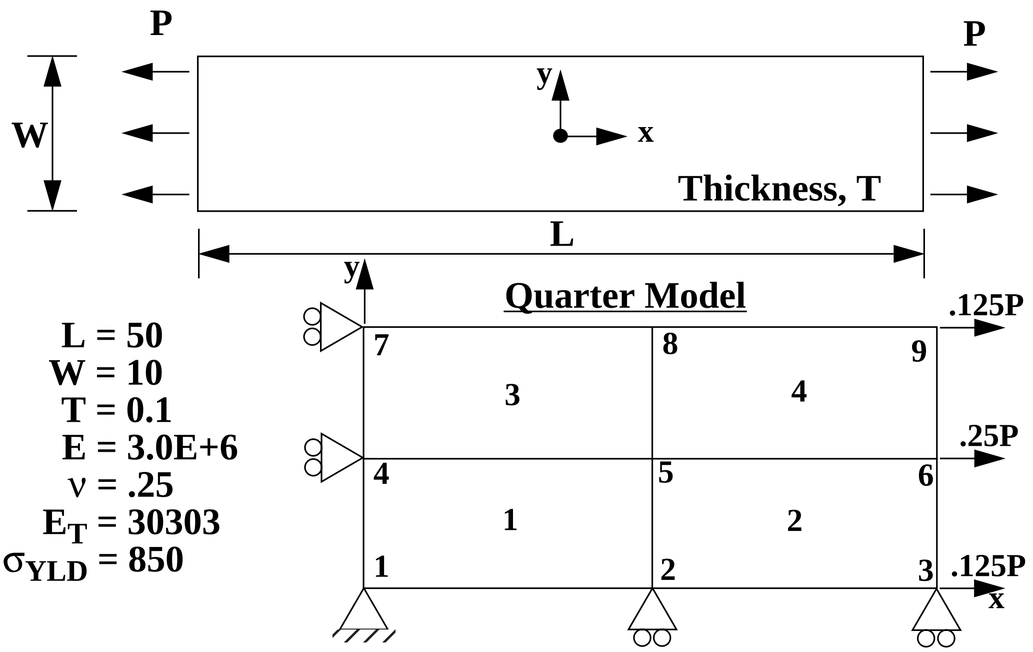


Objectives:

- Demonstrate the use of elasto-plastic material properties.
- Create an accurate deformation plot of the model.
- Create an XY plot of Stress vs. Strain for all the subcases.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Model the elasto-plastic behavior of the material.
2. Analyze the model subjected to the following load history:

   1) Load P = 800
   2) Load P = 1000
   3) Unload P = 950
   4) Unload P = 0

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 stress-dependent material property for the nonlinear material (MATS1)
- Generate an input file and submit it to the MSC/NASTRAN solver for a nonlinear static analysis.
- Review the results.

Input File for Modification:

prob6.dat

```
ID NAS103, WORKSHOP 6
TIME 10
SOL 106
CEND
ECHO=BOTH
STRESS=ALL
DISP=ALL
TITLE=SIMPLE TENSION STRIP ELASTO-PLASTIC ANALYSIS (VON MISES MODEL)
$
  SUBCASE 10
SUBTITLE=ELASTIC--LOAD TO 800. PSI
$
  SUBCASE 20
SUBTITLE=PLASTIC--LOAD TO 1000. PSI
$
  SUBCASE 30
SUBTITLE=ELASTIC--UNLOAD BACK AROUND ELBOW TO 950. PSI
$
  SUBCASE 40
SUBTITLE=ELASTIC--UNLOAD COMPLETLY TO 0. PSI
$
BEGIN BULK
$
$ Geometry
GRID,1, ,0,0,0,,123456
=,*(3),=,=,*(2.5),=,=,13456
=(1)
GRID,2, ,12.5,0,0, ,23456
=,*(3),=,=,*(2.5),=,=,3456
=(1)
GRID,3, ,25.0,0,0, ,23456
=,*(3),=,=,*(2.5),=,=,3456
=(1)
CQUAD4,1,30,1,2,5,4
CQUAD4,2,30,2,3,6,5
CQUAD4,3,30,4,5,8,7
CQUAD4,4,30,5,6,9,8
```

```
$
$ Loading
FORCE, 10, 3, 0, 100., 1.0, 0.0, 0.0
FORCE, 10, 6, 0, 200., 1.0, 0.0, 0.0
FORCE, 10, 9, 0, 100., 1.0, 0.0, 0.0
FORCE, 20, 3, 0, 125., 1.0, 0.0, 0.0
FORCE, 20, 6, 0, 250., 1.0, 0.0, 0.0
FORCE, 20, 9, 0, 125., 1.0, 0.0, 0.0
FORCE, 30, 3, 0, 118.75, 1.0, 0.0, 0.0
FORCE, 30, 6, 0, 237.50, 1.0, 0.0, 0.0
FORCE, 30, 9, 0, 118.75, 1.0, 0.0, 0.0
FORCE, 40, 3, 0, 0., 1.0, 0.0, 0.0
FORCE, 40, 6, 0, 0., 1.0, 0.0, 0.0
FORCE, 40, 9, 0, 0., 1.0, 0.0, 0.0
$
$ Parameters
NLPARM, 10, 1,  , AUTO,  ,  , P
NLPARM, 20, 8,  , AUTO,  ,  , P
NLPARM, 30, 5,  , AUTO,  ,  , P
NLPARM, 40, 2,  , AUTO,  ,  , P
$
$ Properties
PSHELL, 30, 1, 0.1
MAT1, 1, 3.0+6,  , .25
ENDDATA
```

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 12, otherwise, proceed to step 2.

2. Create a new database called **prob6.db**.

File/New...

New Database Name:	**prob6**
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, **prob6.ses**. If you choose to build the model yourself, proceed to step 4.

File/Session/Play...

Session File List:	**prob6.ses**
Apply	

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

4. Create a surface representing a quarter model of a plate.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List	**<25, 5, 0>**

Apply

5. Mesh the model.

First, plant mesh seeds to mesh the quarter plate with 4 elements.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number =	**2**
Curve List:	*(Select top and left edges.)*

Apply

Next, mesh the plate with Quad4 elements.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	**Surface 1** *(Select the surface.)*

Apply

6. Create the boundary conditions for the model.

Create the first constraint for the model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name	**constraint_1**

Input Data...

Translation < T1 T2 T3 >	**< 0, , >**
OK	
Select Application Region...	
Geometry Filter	● **FEM**
Select Nodes	*(Select nodes on left edge.)*
Add	
OK	
Apply	

Create the second model constraint.

New Set Name:	**constraint_2**
Input Data...	
Translation < T1 T2 T3 >	**< , 0 , >**
OK	
Select Application Region...	
Geometry Filter	● **FEM**
Select Nodes	*(Select nodes on bottom edge.)*
Add	
OK	
Apply	

Create the final constraint.

New Set Name	**constraint_3**
Input Data...	
Translation < T1 T2 T3 >	**< , , 0 >**
Rotation < R1 R2 R3 >	**< 0, 0, 0 >**
OK	
Select Application Region...	

Geometry Filter ● **FEM**

Select Nodes *(Select all nodes.)*

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: **force_1a**

Input Data...

Force <F1 F2 F3> **<100, 0, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes *(Select the two corner nodes and the nodes on the right edge.)*

Add

OK

Apply

And the second...

New Set Name **force_1b**

Input Data...

Force <F1 F2 F3> **<200, 0, 0>**

OK

Select Application Region...

Geometry Filter ● **FEM**

Select Nodes *(Select mid-node on the right edge.)*

Add

OK

Apply

8. Create the load cases for the model.

Instead of creating 8 separate loads for the model and referencing each pair to its respective load case, we will now use the feature of load scaling to create the remaining load cases from the first set.

◆ **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
Displ_constraint_3
Force_force_1a
Force_force_1b

Be sure that the LBC Scale Factor in spreadsheet for all Loads/BCs are 1.

OK

Apply

Create the second load case by scaling the loads in the first case.

Load Case Name: **case_2**

Assign/Prioritize Loads/BCs

Highlight both force cells in the LBC Scale Factor column.

LBC Scale Factor *type* **1.25** *and hit* ***Enter***

OK

Apply

Create the third load case.

Load Case Name:	**case_3**

Assign/Prioritize Loads/BCs

Highlight both force cells in the LBC Scale Factor column.

LBC Scale Factor	*type* **1.1875** *and hit* ***Enter***

OK

Apply

Create the final load case.

Load Case Name:	**case_4**

Assign/Prioritize Loads/BCs

Highlight both force cells in the LBC Scale Factor column.

LBC Scale Factor	*type* **0** *and hit* ***Enter***

OK

Apply

This is where the session file ends.

9. Create the elasto-plastic 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 =	**3.E6**
Poisson's Ratio	**.25**

Apply

Next, create the elasto-plastic properties of the material.

Constitutive Model:	**Elastoplastic**
Nonlinear Data Input:	**Hardening Slope**
Hardening Slope	**30303**
Yield Point	**850**

Apply

Cancel

10. Create the element property for the plate.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**plate**

Input Properties...

Material Name	**m:mat_1**
Thickness	**0.1**

OK

Select Members	**Surface 1** *(Select the surface.)*

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	**prob6**

Solution Type...

Solution Type	● **NONLINEAR STATIC**

OK

Subcase Create...

Available Subcases	**case_1**

Subcase Parameters...

Number of Load Increments	**1**
(Turn off work error criterion.)	❒ **Work Error**

OK

Output Requests...

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

Delete

OK

Apply

Repeat the procedure for the second subcase.

Available Subcases:	**case_2**

Subcase Parameters...

Number of Load Increments:	**8**
(Turn off work error criterion.)	❒ **Work Error**

OK

Output Requests...

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

Delete

OK

Apply

And the third...

Available Subcases:	**case_3**
Subcase Parameters...	
Number of Load Increments:	**5**
(Turn off work error criterion.)	❐ **Work Error**
OK	
Output Requests...	
Output Requests:	**SPCFORCES(SORT1...**
Delete	
OK	
Apply	

And the fourth (and final) subcase.

Available Subcases:	**case_4**
Subcase Parameters...	
Number of Load Increments:	**2**
(Turn off work error criterion.)	❐ **Work Error**
OK	
Output Requests...	
Output Requests:	**SPCFORCES(SORT1...**
Delete	
OK	
Apply	

Cancel

Finally, select all the subcases before submitting the analysis

Subcase Select...

Subcases for Solution Sequence	**case_1** **case_2** **case_3** **case_4**
Subcases Selected	*(Deselect **Default)***

OK

Apply

An input file called **prob6.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 13**.

Generating an input file for MSC/NASTRAN Users:

12. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob6.dat**):

```
ASSIGN OUTPUT2 = 'prob6.op2' , UNIT=12
ID NAS103, WORKSHOP 6 SOLUTION
TIME 10
SOL 106
CEND
ECHO=BOTH
STRESS=ALL
DISP=ALL
TITLE=SIMPLE TENSION STRIP ELASTO-PLASTIC ANALYSIS (VON MISES MODEL)
$
 SUBCASE 10
SUBTITLE=ELASTIC--LOAD TO 800. PSI
LOAD=10     $ LOAD UP TO 800. PSI
NLPARM=10   $ IN 1 STEP
$
 SUBCASE 20
SUBTITLE=PLASTIC--LOAD TO 1000. PSI
LOAD=20     $ LOAD UP TO 1000. PSI
NLPARM=20   $ IN 8 STEPS
$
 SUBCASE 30
SUBTITLE=ELASTIC--UNLOAD BACK AROUND ELBOW TO 950. PSI
LOAD=30     $ UNLOAD TO 950. PSI
NLPARM=30   $ IN 5 STEPS
$
 SUBCASE 40
SUBTITLE=ELASTIC--UNLOAD COMPLETLY TO 0. PSI
LOAD=40     $ UNLOAD TO 0. PSI
NLPARM=40   $ IN 2 STEPS
$
BEGIN BULK
$
$ Geometry
GRID,1, ,0,0,0,,123456
=,*(3),=,=,*(2.5),=,=,13456
=(1)
```

```
GRID,2, ,12.5,0,0, ,23456
=,*(3),=,=,*(2.5),=,=,3456
=(1)
GRID,3, ,25.0,0,0, ,23456
=,*(3),=,=,*(2.5),=,=,3456
=(1)
CQUAD4,1,30,1,2,5,4
CQUAD4,2,30,2,3,6,5
CQUAD4,3,30,4,5,8,7
CQUAD4,4,30,5,6,9,8
$
$ Loading
FORCE, 10, 3, 0, 100., 1.0, 0.0, 0.0
FORCE, 10, 6, 0, 200., 1.0, 0.0, 0.0
FORCE, 10, 9, 0, 100., 1.0, 0.0, 0.0
FORCE, 20, 3, 0, 125., 1.0, 0.0, 0.0
FORCE, 20, 6, 0, 250., 1.0, 0.0, 0.0
FORCE, 20, 9, 0, 125., 1.0, 0.0, 0.0
FORCE, 30, 3, 0, 118.75, 1.0, 0.0, 0.0
FORCE, 30, 6, 0, 237.50, 1.0, 0.0, 0.0
FORCE, 30, 9, 0, 118.75, 1.0, 0.0, 0.0
FORCE, 40, 3, 0, 0., 1.0, 0.0, 0.0
FORCE, 40, 6, 0, 0., 1.0, 0.0, 0.0
FORCE, 40, 9, 0, 0., 1.0, 0.0, 0.0
$
$ Parameters
NLPARM, 10, 1,  , AUTO,  ,  , P
NLPARM, 20, 8,  , AUTO,  ,  , P
NLPARM, 30, 5,  , AUTO,  ,  , P
NLPARM, 40, 2,  , AUTO,  ,  , P
$
$ Properties
PSHELL, 30, 1, 0.1
MAT1, 1, 3.0+6,  , .25
MATS1, 1,  , PLASTIC, 30303., 1, 1, 850.
ENDDATA
```

Submit the input file for analysis:

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

    13a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob6.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

    13b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob6.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

14. When the analysis is completed, edit the **prob6.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.

14a. While still editing **prob6.f06**, search for the word:

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

What is the x-displacement of Node 3 for the first subcase?

T1= ____________________

What is the x-displacement of Node 3 for the second subcase?

T1 = ____________________

What is the x-displacement of Node 3 for the third subcase?

T1= ____________________

What is the x-displacement of Node 3 for the fourth subcase?

T1 = ____________________

Comparison of Results:

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

```
                                                                                          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      0.0            0.0            0.0            0.0            0.0            0.0
        2      G      3.333333E-03   0.0            0.0            0.0            0.0            0.0
        3      G      6.666666E-03   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      0.0            0.0            0.0            0.0            0.0            0.0
        2      G      6.604432E-02   0.0            0.0            0.0            0.0            0.0
        3      G      1.320886E-01   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      0.0            0.0            0.0            0.0            0.0            0.0
        2      G      6.583558E-02   0.0            0.0            0.0            0.0            0.0
        3      G      1.316710E-01   0.0            0.0            0.0            0.0            0.0
      .
      .
                                                                                          SUBCASE 4
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
        1      G      0.0            0.0            0.0            0.0            0.0            0.0
        2      G      6.187789E-02   0.0            0.0            0.0            0.0            0.0
        3      G      1.237553E-01   0.0            0.0            0.0            0.0            0.0
      .
      .
```

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

17. Proceed with the Reverse Translation process, that is, importing the **prob6.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: **prob6.op2**

OK

Apply

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

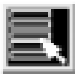
Now we will generate the fringe plot of the model.

◆ **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**

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**

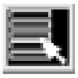
Apply

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**

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*

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

 **Plot Options**

Coordinate Transformation: **None**

Scale Factor **1.0**

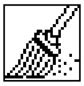
Apply

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

Repeat this process for the other three load cases. For the last load case, change the fringe result to **Nonlinear Strains, Strain Tensor** in order to view the remaining plastic strain of the model

To better fit the results on the screen, zoom out a couple times using
To clear the post-processing results and obtain the original model

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

 **Reset Graphics**

19. Create an XY plot of Stress vs. Strain for all four subcases.

◆ **Results**

Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**
Select Result Case(s)	*(Select all cases.)*
Y:	**Result**
Select Y Result	**Nonlinear Stresses, Stress Tensor**
Quantity:	**von Mises**
X:	**Result**
Select X Result...	
Select X Result	**Nonlinear Strains, Plastic Strain**
OK	

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

 **Target Entities**

Target Entity:	**Elements**
Select Nodes	**Elm 1** *(Select the bottom left element.)*

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Show X Axis Label**

X Axis Label: **Plastic Strain**

X Axis Scale ● **Linear**

X Axis Format...

Label Format: **Exponential**

OK

■ **Show Y Axis Label**

Y Axis Label: **Stress Tensor**

Y Axis Scale ● **Linear**

Y Axis Format...

Label Format: **Fixed**

OK

Apply

To change the title, do the following:

◆ **XY Plot**

Action: **Modify**

Object: **Curve**

Curve List **default_GraphResults Graph 0**

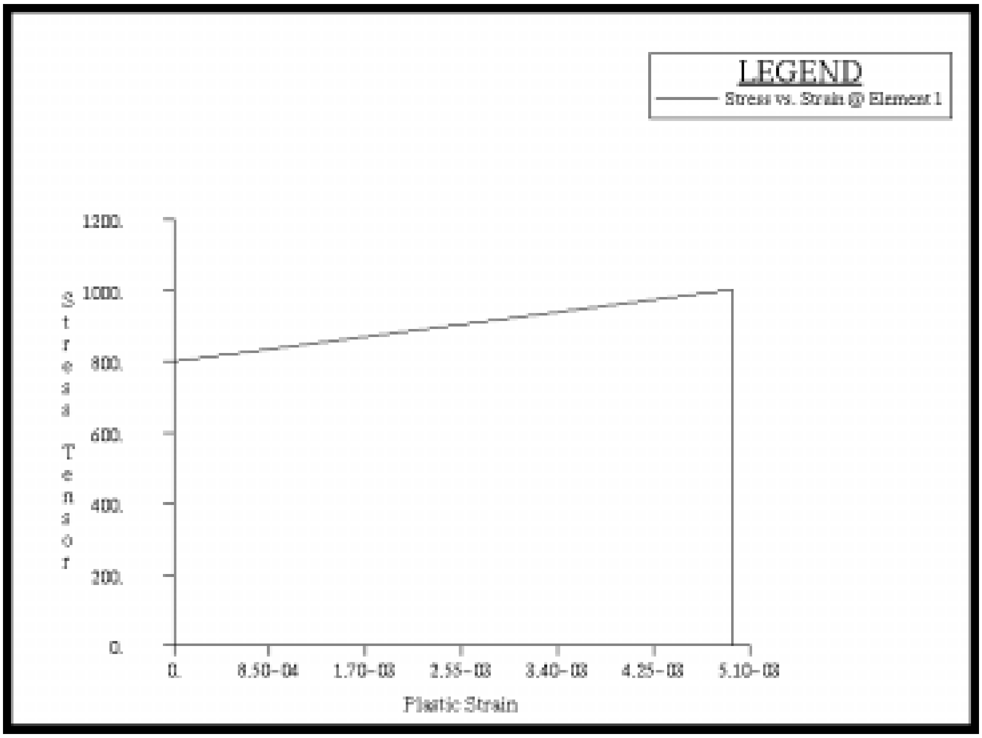
Title...

Curve Title Text **Stress vs. Strain @ Element 1**

Apply

Cancel

The following XY plot should appear on your screen:



Notice the plastic strain still remain as the stresses in the plate relax to zero (complete unloading), the plastic strain still remains.

When you are done viewing, unpost 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: prob6.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January   15, 1998 at
$ 20:10:47.
ASSIGN OUTPUT2 = 'prob6.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 20:05:03
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
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_2
  SUBTITLE=case_2
  NLPARM = 2
  SPC = 2
  LOAD = 5
  DISPLACEMENT(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 3
$ Subcase name : case_3
  SUBTITLE=case_3
  NLPARM = 3
  SPC = 2
  LOAD = 8
  DISPLACEMENT(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
SUBCASE 4
```

```
$ Subcase name : case_4
   SUBTITLE=case_4
   NLPARM = 4
   SPC = 2
   LOAD = 11
   DISPLACEMENT(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 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     P      NO     +     A
+      A        .001
NLPARM   2      8            AUTO    5      25     P      NO     +     B
+      B        .001
NLPARM   3      5            AUTO    5      25     P      NO     +     C
+      C        .001
NLPARM   4      2            AUTO    5      25     P      NO     +     D
+      D        .001
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : plate
PSHELL   1      1      .1     1             1
CQUAD4   1      1      1      2      5      4
CQUAD4   2      1      2      3      6      5
CQUAD4   3      1      4      5      8      7
CQUAD4   4      1      5      6      9      8
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 15-Jan-98          Time: 20:03:34
MATS1    1              PLASTIC 30303.  1      1      850.
MAT1     1      3.+6           .25
$ Nodes of the Entire Model
GRID     1              0.      0.      0.
GRID     2              12.5    0.      0.
GRID     3              25.     0.      0.
GRID     4              0.      2.5     0.
```

```
GRID    5           12.5   2.5    0.
GRID    6           25.    2.5    0.
GRID    7           0.     5.     0.
GRID    8           12.5   5.     0.
GRID    9           25.    5.     0.
$ Loads for Load Case : case_1
SPCADD  2      13     15     16
LOAD    2      1.     1.     10     1.     12
$ Loads for Load Case : case_2
LOAD    5      1.     1.25   10     1.25   12
$ Loads for Load Case : case_3
LOAD    8      1.     1.1875 10     1.1875 12
$ Loads for Load Case : case_4
LOAD    11     1.     0.     10     0.     12
$ Displacement Constraints of Load Set : constraint_1
SPC1    13     1      1      4      7
$ Displacement Constraints of Load Set : constraint_2
SPC1    15     2      1      2      3
$ Displacement Constraints of Load Set : constraint_3
SPC1    16     3456   1      THRU   9
$ Nodal Forces of Load Set : force_1a
FORCE   10     3      0      100.   1.     0.     0.
FORCE   10     9      0      100.   1.     0.     0.
$ Nodal Forces of Load Set : force_1b
FORCE   12     6      0      200.   1.     0.     0.
$ Referenced Coordinate Frames
ENDDATA f588c106
```