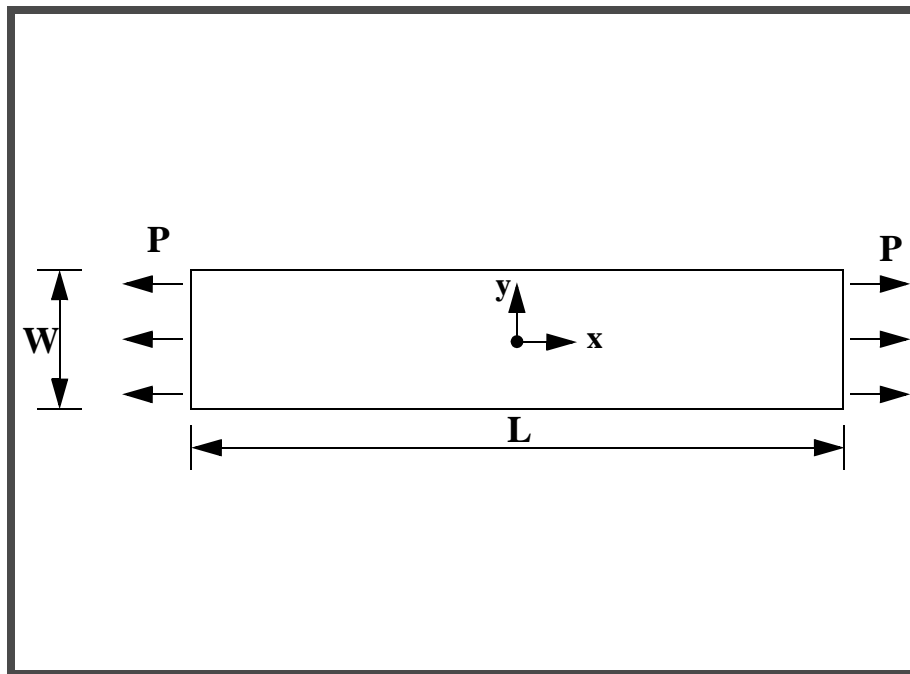


---

## WORKSHOP 29

# *Elasto-Plastic Deformation of a Thin Plate*



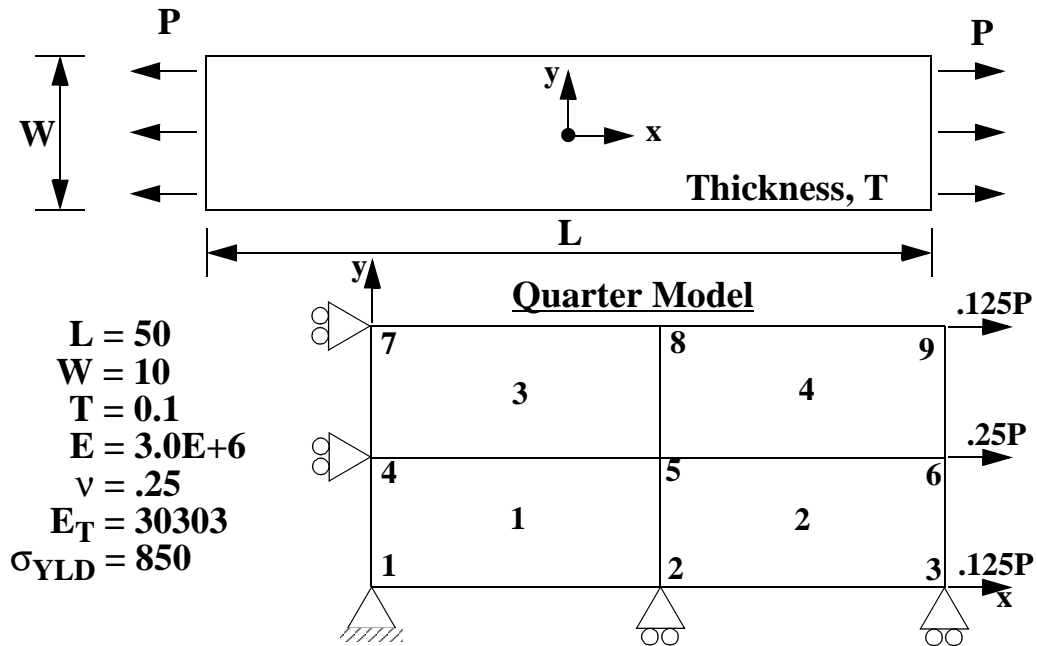
### Objectives:

- 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 (MAT51)
- 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 COMPLETELY 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 utilizing MSC.Patran for generating an input file should go to Step 12, otherwise, proceed to step 2.

2. Create a new database called **prob29.db**.

**File/New...**

*New Database Name:*

**prob29**

**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, **prob29.ses**. If you choose to build the model yourself, proceed to step 4.

**File/Session/Play...**

*Session File List:*

**prob29.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

prob29

**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 **prob29.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 (**prob29.dat**):

```
ASSIGN OUTPUT2 = 'prob29.op2' , UNIT=12
ID NAS103, WORKSHOP 29 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 COMPLETELY 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)
```



## 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 prob29.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 prob29.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.
14. When the analysis is completed, edit the **prob29.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 **prob29.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 **prob29.op2** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	<b>Read Output2</b>
<i>Object:</i>	<b>Result Entities</b>
<i>Method:</i>	<b>Translate</b>
<b>Select Results File...</b>	
<i>Selected Results File:</i>	<b>prob29.op2</b>
<b>OK</b>	
<b>Apply</b>	

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

Now we will generate the fringe plot of the model.

◆ **Results**

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

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



**Select Results**

<i>Select Result Case(s)</i>	<b>(Select the first case.)</b>
<i>Select Fringe Result</i>	<b>Displacements, Translational</b>
<i>Quantity:</i>	<b>Magnitude</b>

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**

<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"/>

<i>Select Result Case(s)</i>	<input type="text" value="(Select all cases.)"/>
<i>Y:</i>	<input type="text" value="Result"/>
<i>Select Y Result</i>	<input type="text" value="Nonlinear Stresses, Stress Tensor"/>
<i>Quantity:</i>	<input type="text" value="von Mises"/>
<i>X:</i>	<input type="text" value="Result"/>
<input type="text" value="Select X Result..."/>	
<i>Select X Result</i>	<input type="text" value="Nonlinear Strains, Plastic Strain"/>
<input type="text" value="OK"/>	

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



**Target Entities**

<i>Target Entity:</i>	<input type="text" value="Elements"/>
<i>Select Nodes</i>	<input type="text" value="Elm 1"/>
	<i>(Select the bottom left element.)</i>

---

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...

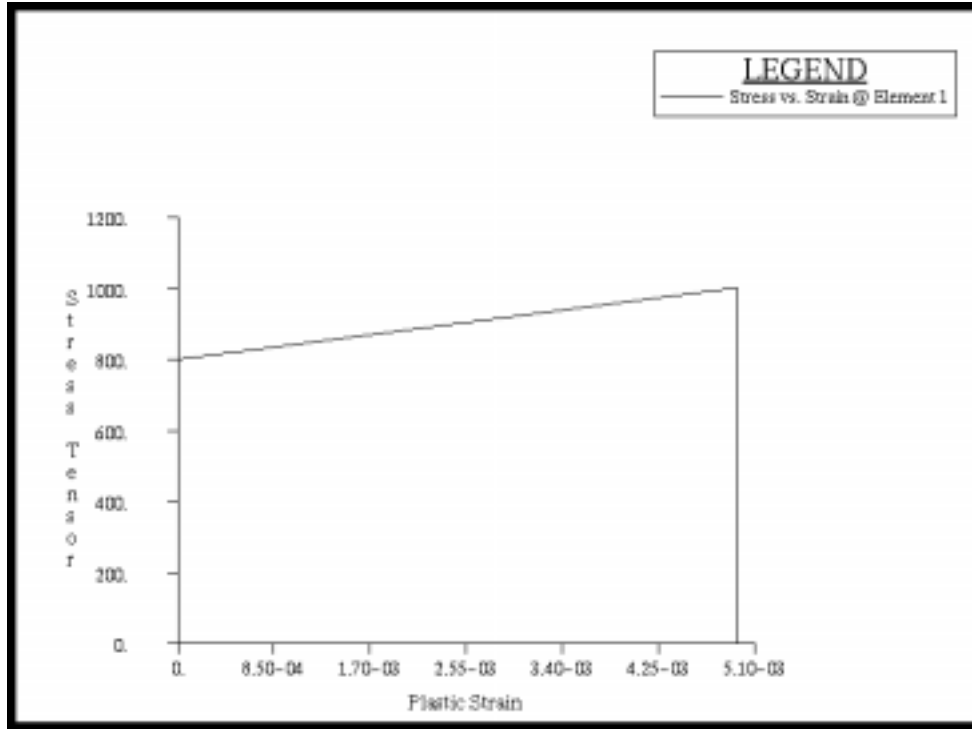
*Curve Title Text*

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: prob29.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 = 'prob29.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
  NLPARAM = 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
  NLPARAM = 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
  NLPARAM = 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
  NLPARAM = 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
NLPARAM 1  1      AUTO  5  25  P  NO  +  A
+  A      .001
NLPARAM 2  8      AUTO  5  25  P  NO  +  B
+  B      .001
NLPARAM 3  5      AUTO  5  25  P  NO  +  C
+  C      .001
NLPARAM 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

```