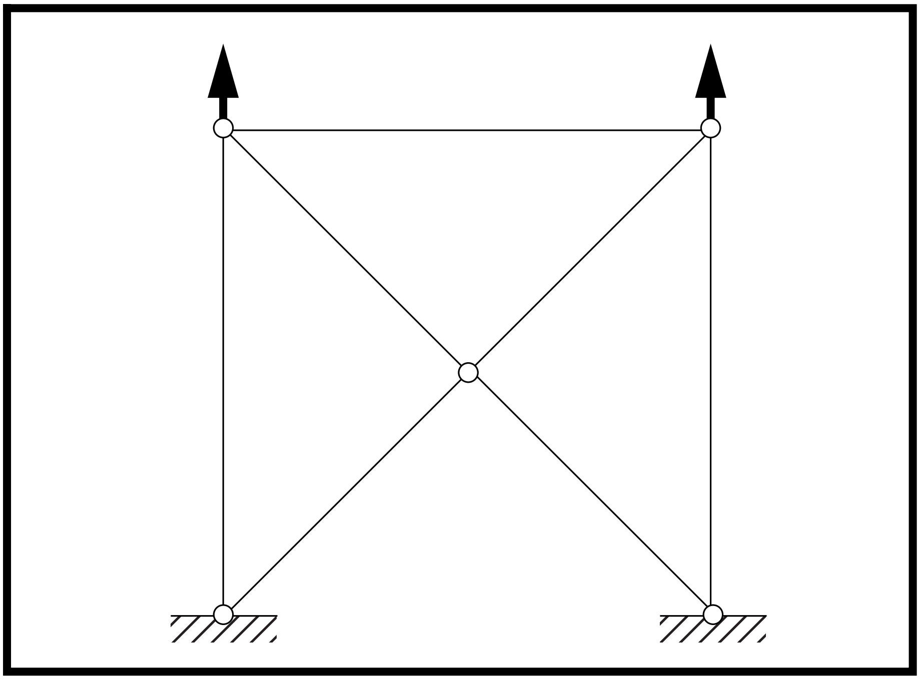
WORKSHOP PROBLEM 8

Elasto-Plastic Deformation of a Truss Structure

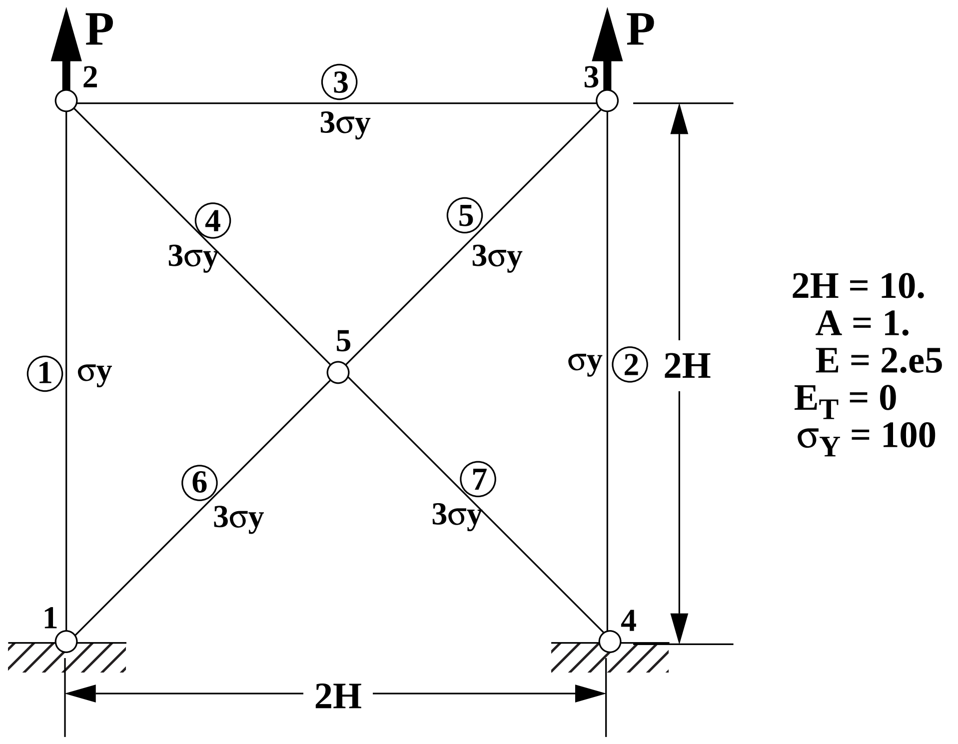


Objectives:

- Demonstrate the use of elasto-plastic material properties.
- Create an enforced displacement on the model.
- Create an XY plot of Stress vs. Percent Load of three elements.

Model Description:

For the structure below:



Add Case Control commands and Bulk Data Entries to:

1. Model the elasto-plastic behavior of the materials.

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 analysis parameter (NLPARM).
- Input the proper stress-dependent material properties for the nonlinear materials (MATS1).
- Input the enforced displacement on the model (SPC).
- Generate an input file and submit it to the MSC/NASTRAN solver for nonlinear static analysis.
- Review the results.

Input File for Modification:

prob8.dat

```
ID NAS103 WORKSHOP 8
TIME 5
SOL 106
CEND
TITLE=MATERIAL NONLINEAR PROBLEM - LIMIT ANALYSIS
SUBTITLE=REF: WHITE AND HODGE; COMP. AND STRUCT.; 12:769-776 (1980)
  DISP=ALL
  SPCF=ALL
  STRESS=ALL
  SPC=12
  NLPARM=10
BEGING BULK
PARAM,POST,-1
$ GEOMETRY
GRID,1,,0.,0.
GRID,2,,0.,10.
GRID,3,,10.,10.
GRID,4,,10.,0.
GRID,5,,5.,5.
GRDSET,,,,,,3
$ CONNECTIVITY
CONROD,1,1,2,10,1.
CONROD,2,3,4,10,1.
CONROD,3,2,3,11,1.
CONROD,4,2,5,11,1.
CONROD,5,3,5,11,1.
CONROD,6,1,5,11,1.
CONROD,7,4,5,11,1.
$ PROPERTIES
MAT1,10,2.+5
MAT1,11,2.+5
$ CONSTRAINTS
SPC1,12,12,1,4
$ LOADING
$ PARAMETERS
$ SOLUTION STRATEGY
$
ENDDATA
```

Exercise Procedure:

1. Users who are not utilitizing MSC/PATRAN for generating an input file should go to Step 10, otherwise, proceed to step 2.

2. Create a new database called **prob8.db**.

File/New...

New Database Name: **prob8**

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, **prob8.ses**. If you choose to build the model yourself, proceed to the next step.

File/Session/Play...

Session File List: **prob8.ses**

Apply

The model has now been created. Skip to **Step 6**.

4. Create the finite element model as shown in the description.

First, create the nodes of the model.

◆ **Finite Elements**

Action: **Create**

Object: **Node**

Method: **Edit**

❐ **Associate With Geometry**

Node Location List: **[0, 0, 0]**

Apply

Node Location List: **[0, 10, 0]**

Apply

Node Location List: **[10, 10, 0]**

Apply

Node Location List: **[10, 0, 0]**

Apply

Node Location List: **[5, 5, 0]**

Apply

To better view the nodes of the model, click on the following toolbar icons:

Node Size **Show Labels**

Next, create the elements of the model by connecting the pairs of nodes together.

◆ **Finite Elements**

Action:	**Create**
Object:	**Element**
Method:	**Edit**
Shape:	**Bar**
Node 1:	**Node 1**
Node 2:	**Node 2**

Apply

Node 1: **Node 3**

Node 2:	**Node 4**

Apply

Node 1:	**Node 2**
Node 2:	**Node 3**

Apply

Node 1:	**Node 2**
Node 2:	**Node 5**

Apply

Node 1:	**Node 3**
Node 2:	**Node 5**

Apply

Node 1:	**Node 1**
Node 2:	**Node 5**

Apply

Node 1:	**Node 4**
Node 2:	**Node 5**

Apply

5. Create the boundary conditions for the model.

Create the base constraint for the model.

◆ **Loads/BCs**

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

Input Data...

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

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Node 1, 4**

(Select nodes at bottom.)

Add

OK

Apply

This is where the session file ends.

6. Create the enforced displacement for the model.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **pull**

Input Data...

Translation < T1 T2 T3 >: **< , .05, >**

OK

Select Application Region...

Geometry Filter: ● **FEM**

Select Nodes: **Nodes 2, 3**

(Select nodes at top.)

Add

OK

Apply

7. Create the elasto-plastic materials for the model.

First, define the linear elastic properties of the first material.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**2e5**

Apply

Next, define the elasto-plastic properties of the first material.

Constitutive Model:	**Elastoplastic**
Nonlinear Data Input:	**Hardening Slope**
Hardening Slope:	**0**
Yield Point:	**100**

Apply

Cancel

Now, define the linear elastic properties of the second material.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_2**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**2e5**

Apply

Finally, define the elasto-plastic properties of the second material.

Constitutive Model: **Elastoplastic**

Nonlinear Data Input: **Hardening Slope**

Hardening Slope: **0**

Yield Point: **300**

Apply

Cancel

8. Create the element properties for the model.

Create the first element property set.

◆ **Properties**

Action: **Create**

Dimension: **1D**

Type: **Rod**

Property Set Name: **bar_1**

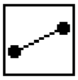
Input Properties...

Material Name: **m:mat_1**

Area: **1.0**

OK

In order to select the beam elements of the model, first click on the following entity select icon:

 **Beam Elements**

Select Members: **Elm 1, 2**

(Select the vertical elements.)

Add

Apply

Now create the second element property set.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Rod**
Property Set Name:	**bar_2**
Input Properties...	
Material Name:	**mat_2**
Area:	**1.0**
OK	
Select Members:	**Elm 3:7** *(Select the rest of the elements.)*
Add	
Apply	

9. 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	**prob8**
Solution Type...	
Solution Type:	● **NONLINEAR STATIC**
OK	
Subcase Create...	
Available Subcases:	**Default**
Subcase Parameters...	

Number of Load Increments:	**20**

OK

Output Requests...

Form Type:	**Advanced**
Intermediate Output Option:	**Yes**

OK

Apply

Cancel

Apply

An input file called **`prob8.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 11**.

Generating an input file for MSC/NASTRAN Users:

10. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob8.dat**):

```
ASSIGN OUTPUT2 = ‘prob8.op2’, UNIT = 12
ID NAS103 WORKSHOP 8 SOLUTION
TIME 5
SOL 106
CEND
TITLE=MATERIAL NONLINEAR PROBLEM - LIMIT ANALYSIS
SUBTITLE=REF: WHITE AND HODGE; COMP. AND STRUCT.; 12:769-776 (1980)
  DISP=ALL
  SPCF=ALL
  STRESS=ALL
  SPC=12
  NLPARM=10
BEGING BULK
PARAM,POST,-1
$ GEOMETRY
GRID,1,,0.,0.
GRID,2,,0.,10.
GRID,3,,10.,10.
GRID,4,,10.,0.
GRID,5,,5.,5.
GRDSET,,,,,,,3
$ CONNECTIVITY
CONROD,1,1,2,10,1.
CONROD,2,3,4,10,1.
CONROD,3,2,3,11,1.
CONROD,4,2,5,11,1.
CONROD,5,3,5,11,1.
CONROD,6,1,5,11,1.
CONROD,7,4,5,11,1.
$ PROPERTIES
MAT1,10,2.+5
MATS1,10,,PLASTIC,0.,,,100.
MAT1,11,2.+5
MATS1,11,,PLASTIC,0.,,,300.
$ CONSTRAINTS
SPC1,12,12,1,4
$ LOADING
SPC,12,2,2,.05
SPC,12,3,2,.05
```

```
$ PARAMETERS
$ SOLUTION STRATEGY
NLPARM,10,20,,,,,,YES
$
ENDDATA
```

Submit the input file for analysis:

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

11a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob8.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.

11b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob8.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.

12. When the analysis is completed, edit the **prob8.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.

12a. While still editing **prob8.f06**, search for the word:

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

What are the x and y displacements of Node 3 at the end of the subcase?

T1= ____________________

T2= ____________________

What is the stress in Element 1 at the end of the subcase?

Stress = ____________________

What is the stress in Element 3at the end of the subcase?

Stress = ____________________

What is the stress in Element 4 at the end of the subcase?

Stress = ____________________

Comparison of Results:

13. 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	5.287338E-03	5.000000E-02	0.0	0.0	0.0	0.0
3	G	-5.287338E-03	5.000000E-02	0.0	0.0	0.0	0.0
4	G	0.0	0.0	0.0	0.0	0.0	0.0
5	G	-9.606084E-17	2.765829E-02	0.0	0.0	0.0	0.0

0 NONLINEAR

SUBCASE 1

LOAD STEP = 1.00000E+00

S T R E S S E S   I N   R O D   E L E M E N T S   (C R O D)

ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN	TORSIONAL STRESS	SAFETY MARGIN	ELEMENT ID.	AXIAL STRESS	SAFETY MARGIN
1	1.000000E+02		0.0		2	1.000000E+02	
3	-2.114935E+02		0.0		4	3.000000E+02	
5	3.000000E+02		0.0		6	3.000000E+02	
7	3.000000E+02		0.0				

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

15. Proceed with the Reverse Translation process, that is, importing the **prob8.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

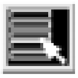
Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File:	**prob8.op2**
OK	
Apply	

16. Create an XY plot of Stress vs Percent Load for elements 1, 3, and 4.

◆ **Results**

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

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



Select Results

Select Result Case(s)	*(Select all cases.)*
Y:	**Result**
Select Y Result	**Nonlinear Stresses, Stress Tensor**
Quantity:	**von Mises**

X: **Global Variable**

Variable: **Percent of Load**

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

 **Target Entities**

Target Entity: **Elements**

Select Elements **Elm 1**

Click on the **Display Attributes** icon.

 **Display Attributes**

■ **Append Curves in XY Win...**

Apply

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

 **Target Entities**

Target Entity: **Elements**

Select Elements **Elm 3**

Apply

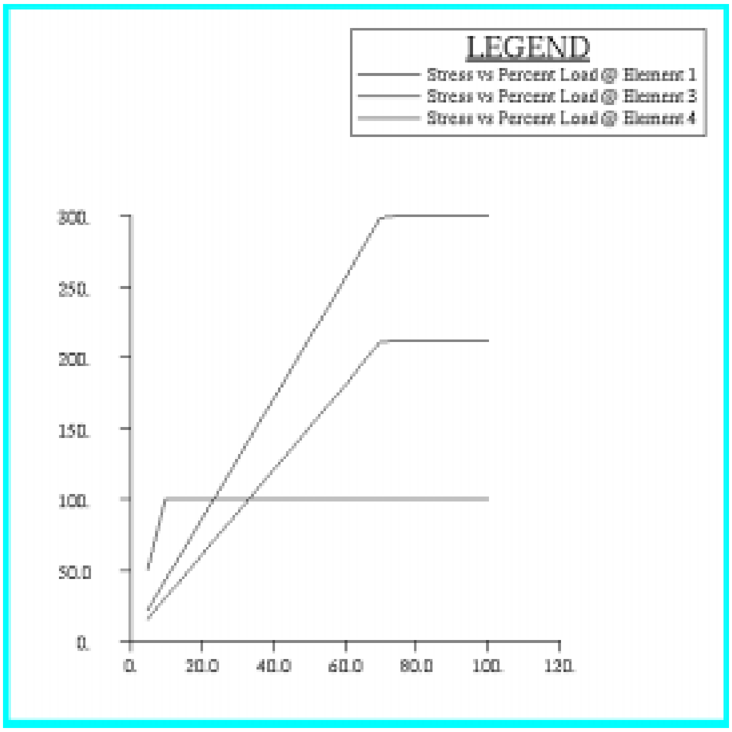
Finally select Element 4.

Target Entity: **Elements**

Select Elements **Elm 4**

Apply

The following XY plot should appear on your screen.



Notice how the stress in Element 1 increases linearly to the yield point and then maintains, just as the material properties defined the stress-strain relationship.

Notice that the same behavior exists for Element 4, only with a higher yield point.

Since the cross-sectional area remains constant in this model, force is directly proportional to stress. When the stresses for all the elements reach their maximum (except Element 3), the forces also reach their maximum and remain constant as well. This is why the stresses on Element 3 do not increase after the other elements have reached their limits.

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob8.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.0 ) on January   15, 1998 at
$ 19:34:25.
ASSIGN OUTPUT2 = 'prob8.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 19:29:30
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
   SUBTITLE=Default
   NLPARM = 1
   SPC = 2
   LOAD = 1
   DISPLACEMENT(SORT1,REAL)=ALL
   SPCFORCES(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 0
PARAM    K6ROT   100.
PARAM    WTMASS  1.
PARAM    LGDISP  1
PARAM,NOCOMPS,-1
NLPARM   1       20              AUTO    5       25      PW      YES     +       A
+       A        .001    1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : bar_1
PROD     1       1       1.
CROD     1       1       1       2
CROD     2       1       3       4
$ Elements and Element Properties for region : bar_2
PROD     2       2       1.
```

```
CROD    3   2   2   3
CROD    4   2   2   5
CROD    5   2   3   5
CROD    6   2   1   5
CROD    7   2   4   5
$ Referenced Material Records
$ Material Record : mat_1
$ Description of Material : Date: 15-Jan-98      Time: 19:26:18
MATS1   1       PLASTIC 0.  1   1   100.
MAT1    1   200000.
$ Material Record : mat_2
$ Description of Material : Date: 15-Jan-98      Time: 19:26:18
MATS1   2       PLASTIC 0.  1   1   300.
MAT1    2   200000.
$ Nodes of the Entire Model
GRID    1       0.  0.  0.
GRID    2       0.  10. 0.
GRID    3       10. 10. 0.
GRID    4       10. 0.  0.
GRID    5       5.  5.  0.
$ Loads for Load Case : Default
SPCADD  2   1   3
$ Enforced Displacements for Load Set : pull
$ Dummy Force Required to Activate the Following Enforced Displacements
FORCE   1   2       0.  .57735  .57735  .57735
SPCD    1   2   2   .05 3   2   .05
$ Displacement Constraints of Load Set : fixity
SPC1    3   123 1   4
$ Displacement Constraints of Load Set : pull
SPC1    1   2   2   3
ENDDATA b7073e6e
```