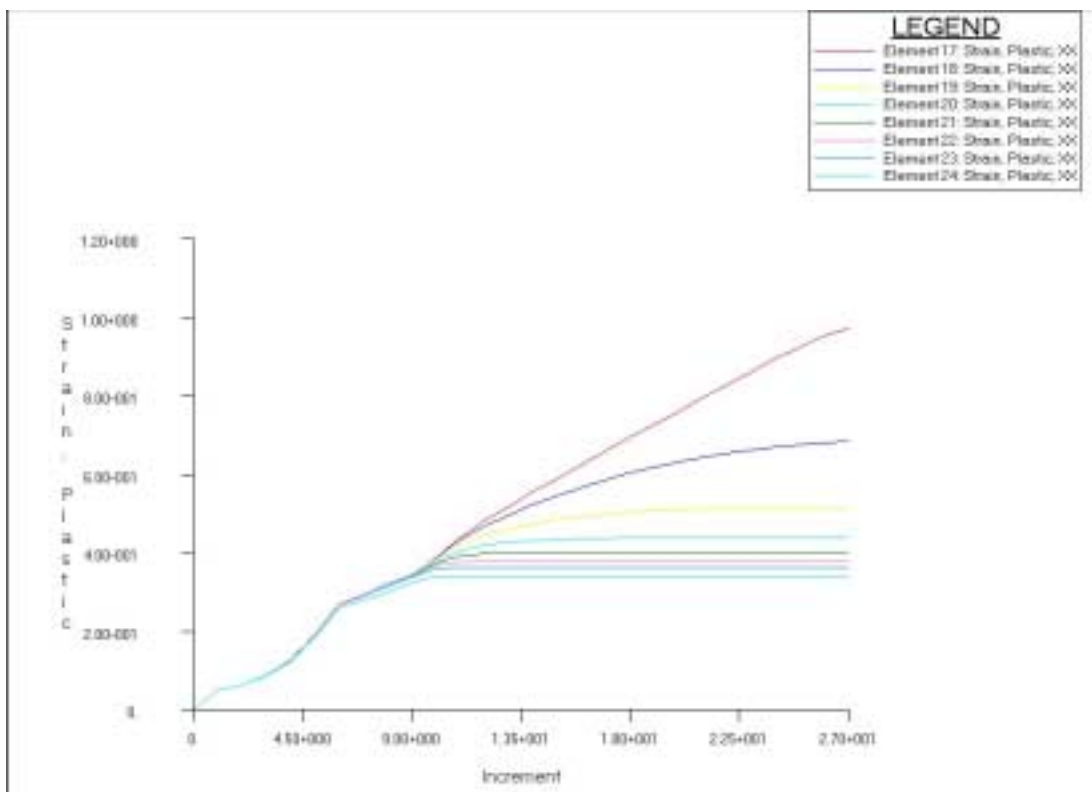


LESSON 11

Necking of a Test Specimen



Objectives:

- Large Deflections/Strains analysis.
- Elastic-Plastic material model using isotropic hardening.



Model Description:

In this lesson, you will stretch an 8 inch long planar steel bar by 2 inches (i.e. 25% of its length). Thus, this elasto-plastic problem will demonstrate the importance of the concept of true stress (or Cauchy stress) in non-linear analysis. This test specimen will be modeled using a quarter symmetry model.

Suggested Exercise Steps:

- Create a 4x1 inch surface in the XY plane.
- Mesh the model with 16x4 mesh of QUAD/4 elements.
- Fix the vertical and horizontal lines of symmetry of the bar and pull the other end by 2 inches.

Exercise Procedure:

1. Create a new database named **necking.db**.

File/New ...

Database Name:

necking.db

OK

Analysis Code:

MSC.Marc

OK

2. Use the XYZ method to create a 4 x 1 surface.

■ **Geometry**

Action:

Create

Object:

Surface

Method:

XYZ

Vector Coordinates List:

<4, 1, 0>

Origin Coordinates List:

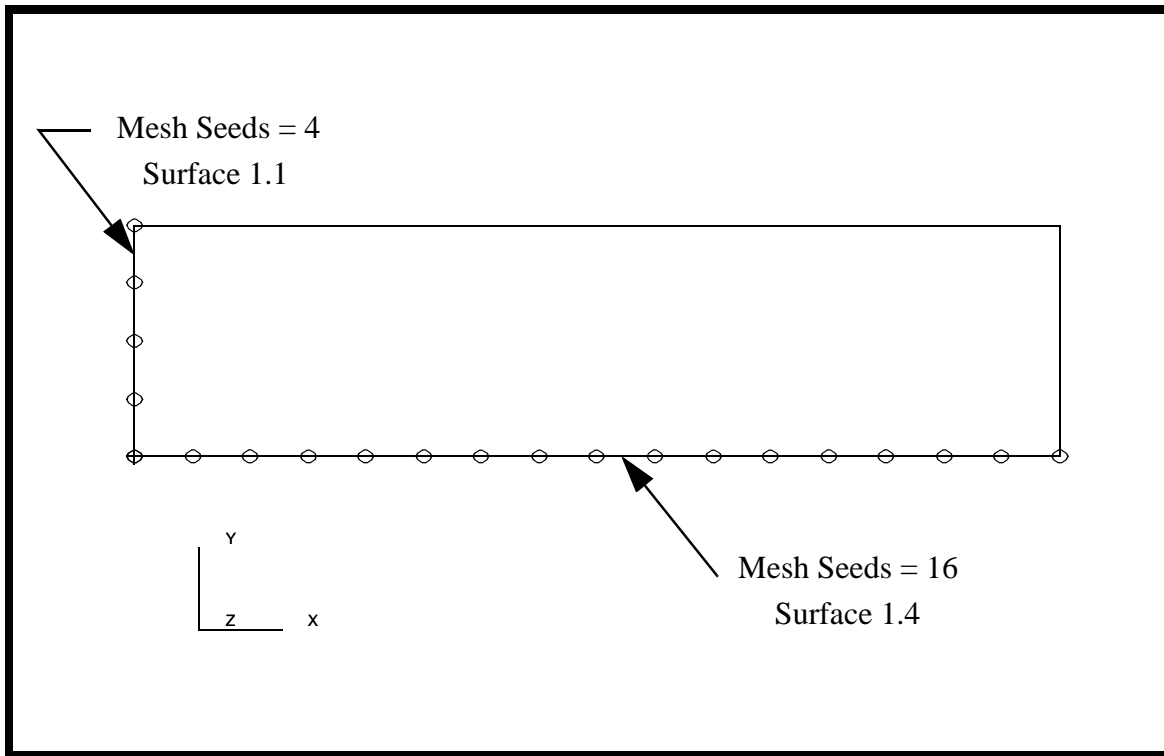
[0, 0, 0]

Apply

3. Create mesh seeds on the surface, 4 on the vertical and 16 on the horizontal.

To create the mesh seeds, first click on the **Finite Elements** toggle in the *Main Window*. You will be defining a mesh seed of 4 on the left edge of the surface and a mesh seed of 16 on the lower edge of the surface as shown in Figure 11.1:

Figure 11.1 - Mesh seed locations



■ Elements

Action:

Object:

Method:

■ Number of Elements

Number:

Curve List:

The next edge to be seeded is the bottom, **Surface 1.4**. It will have 16 elements.

4. Create a group **fem** and make it current. This group will contain all of the finite elements.

Group/Create ...

New Group Name:

fem

■ **Make Current**

Apply

Cancel

5. Mesh the surface.

■ **Elements**

Action:

Create

Object:

Mesh

Type:

Surface

Element Topology:

Quad4

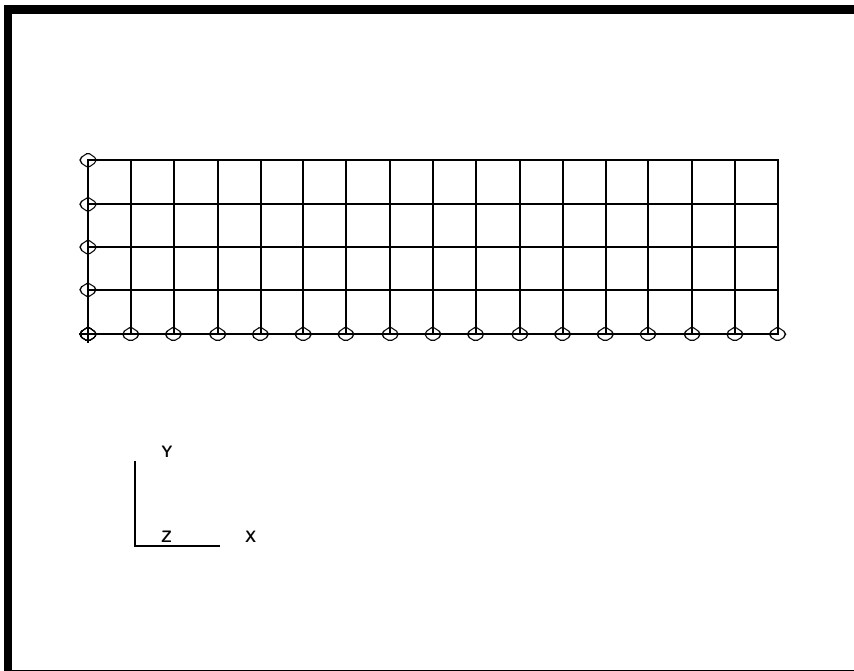
Surface List:

Surface 1

Apply

Your model should now appear as shown in Figure 11.2:

Figure 11.2 - Meshed bar



6. Create a field for the elasto-plastic material data.

■ **Fields**

Action:

Create

Object:

Material Property

Method:

Tabular Input

Field Name:

plastic_s_e

Active Independent Variable:

Temperature (T)

Strain (e)

Input Data...

The *1D Material Scalar Table Data* form needs to be filled out as shown in Table 1. To fill in the table, click on the cell you wish to

Table 1: Stress vs. Strain data for Wrought aluminum 1100

Strain	Stress
0	2842
0.002	4897
0.005	6096
0.015	8045
0.045	10682
0.125	13934
0.350	18225
1.000	23969

edit, enter the value in the *Input Scalar Data* databox and then press <Return>. The table will automatically tab down.

1D Material Scalar Table Data

Input Scalar Data

Data	e	Data
1	0.00000E+00	2.84200E+03
2	2.00000E-03	4.89700E+03
3	5.00000E-03	6.09600E+03
4	1.50000E-02	8.04500E+03
5	4.50000E-02	1.06820E+04
6	1.25000E-01	1.39340E+04
7	3.50000E-01	1.82250E+04
8	1.00000E+00	2.39690E+04

7. Create the material **aluminum_1100**, with elastic and plastic properties.

■ Materials

<i>Action:</i>	Create
<i>Object:</i>	Isotropic
<i>Method:</i>	Manual Input
<i>Material Name:</i>	aluminum_1100
Input Properties...	
<i>Constitutive Model:</i>	Elastic
<i>Elastic Modulus:</i>	1E7
<i>Poisson's Ratio:</i>	0.33
OK	
Apply	

Now to create the plastic material properties.

Input Properties...	
<i>Constitutive Model:</i>	Plastic
<i>Stress vs. Plastic Strain:</i>	plastic_s_e
OK	
Apply	

8. Create the element properties, apply the material **aluminum** to all the elements.

■ Properties

<i>Action:</i>	Create
<i>Dimension:</i>	2D
<i>Type:</i>	2D Solid
<i>Property Set Name:</i>	test_specimen

Options:

Material Name:

Thickness:

Select Members:

9. Create the load to fix the nodes on the left edge of the surface in the x direction.

■ **Load/BCs**

Action:

Object:

Type:

New Set Name:

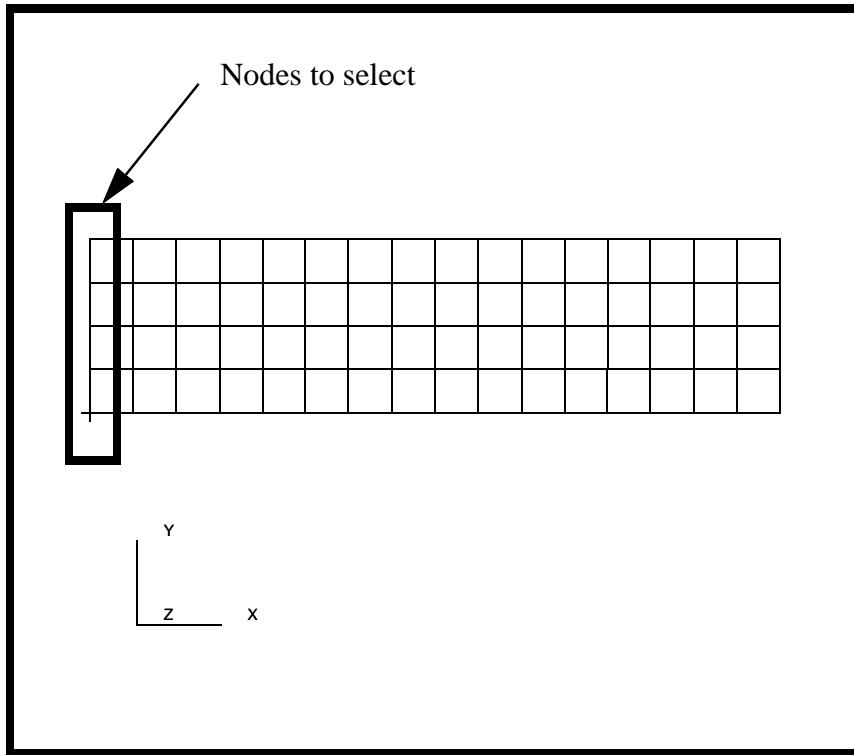
Translations <T1 T2 T3>:

Geometry Filter: **FEM**

Select Nodes:

Click in the *Select Nodes* databox and select the five nodes on the left edge of the surface as shown Figure 11.3:

Figure 11.3 - Nodes to select for symmetry BC



-
-
-

10. Create a displacement set to move the nodes on the right edge of the surface 2 inches in the +X direction.

<i>Action:</i>	<input type="button" value="Create"/>
<i>Object:</i>	<input type="button" value="Displacement"/>
<i>Type:</i>	<input type="button" value="Nodal"/>
<i>New Set Name:</i>	<input type="button" value="pull_at_end"/>
<input type="button" value="Input Data..."/>	
<i>Translations <T1 T2 T3>:</i>	<input type="button" value="<2.0, 0, >"/>
<input type="button" value="OK"/>	

Select Application Region...

Geometry Filter:

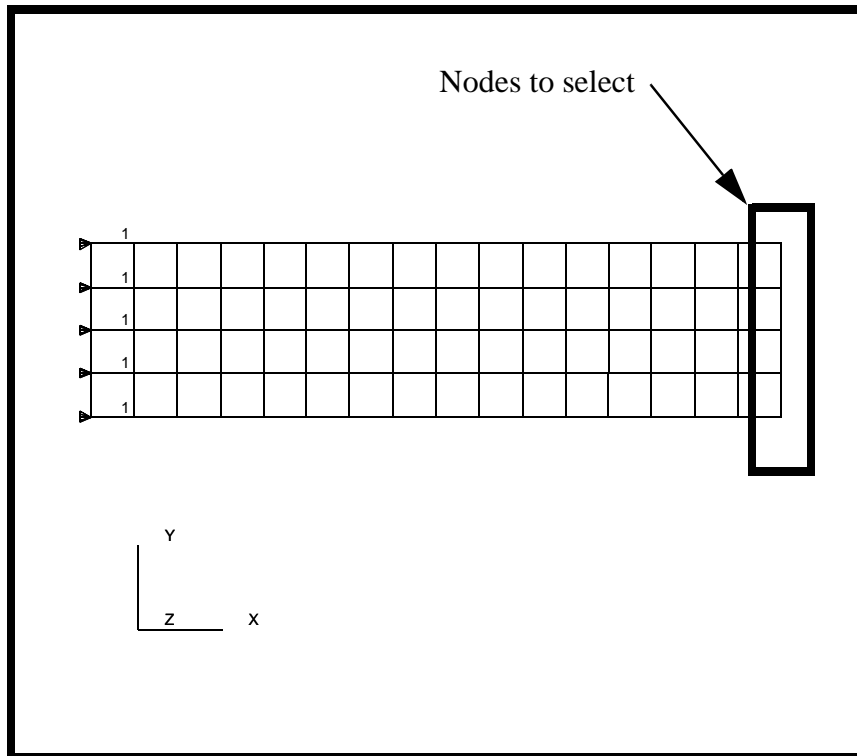
● **FEM**

Select Nodes:

see Figure 11.4

Click in the *Select Nodes* databox and select the nodes on the right edge of the model:

Figure 11.4 - Nodes at pulled end of bar



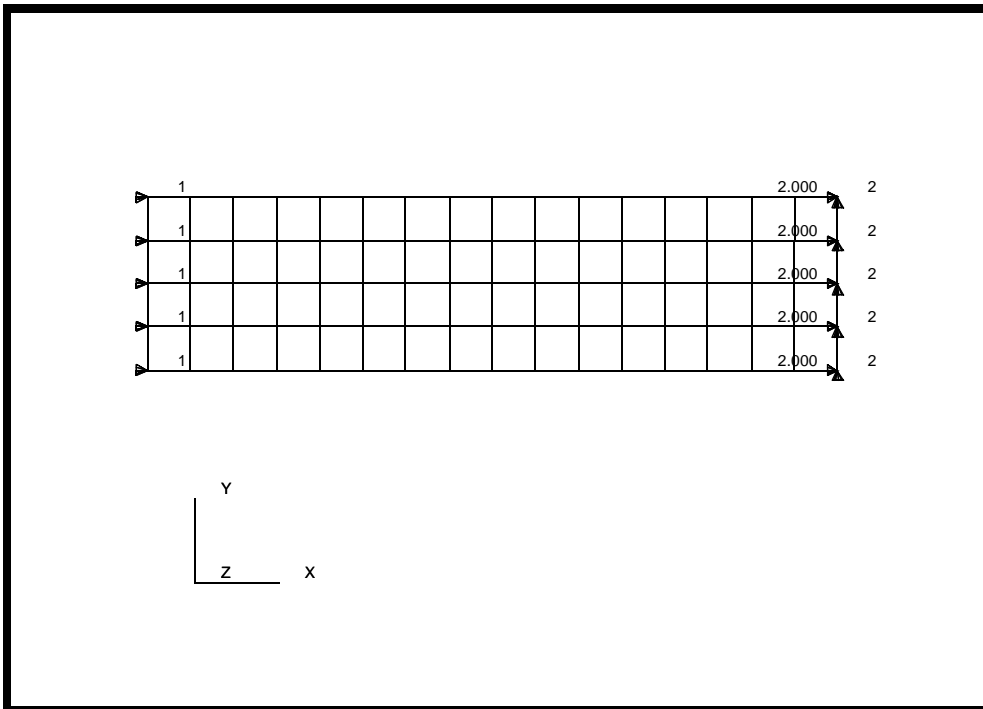
Add

OK

Apply

Your model should now appear as shown in Figure 11.5:

Figure 11.5 - Bar with pull_at_end BC applied



Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

symmetry_horiz

Input Data...

Translations <T1 T2 T3>:

<, 0, >

OK

Select Application Region...

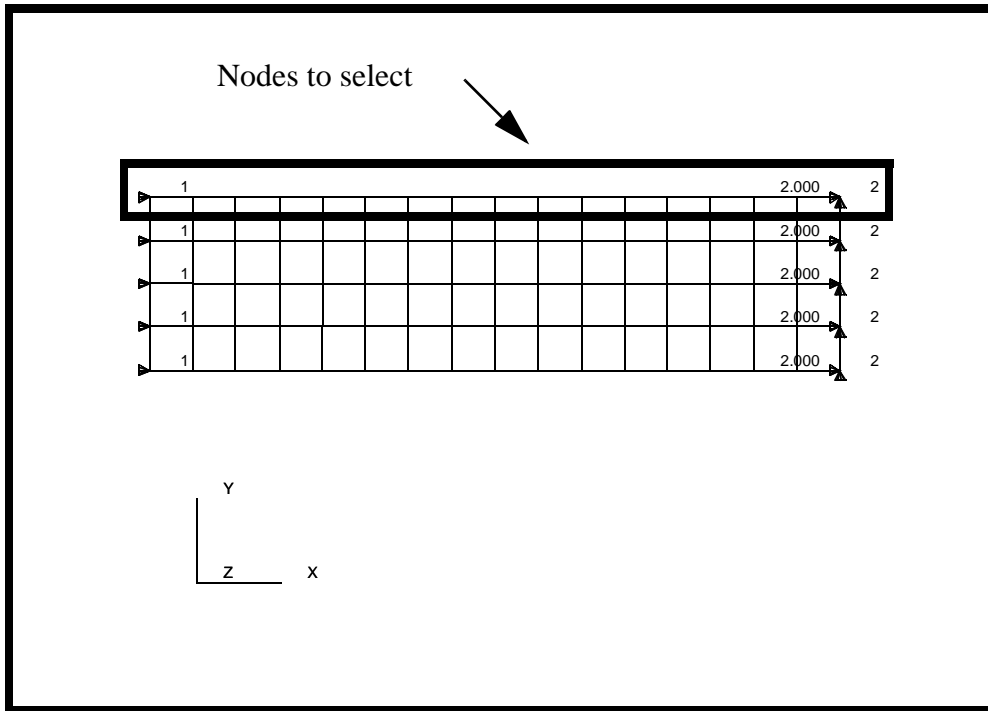
Geometry Filter:

● FEM

Select Nodes:

see Figure 11.6

Figure 11.6 - Nodes to select for symmetry BC



Add

OK

Apply

11. Create the analysis step.

■ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Full Run

Job Name:

necking

Load Step Creation...

Job Step Name:

necking

Solution Type:

Static

Solution Parameters...

NonLinear

**Load Increment
Parameters...***Arclength Method:***Modified Riks/Ramm** **Automatic Cutback***Total Time:***1.0****OK****Iteration Parameters...***Max # of Iterations per
Increment:***40****OK****OK****Output Requests...****Select Element Results...****STRAIN, PLASTIC
COMPONENTS (321)****OK****OK****Apply****Cancel**

12. Select the analysis step and submit it.

Be sure to deselect the Default static step.

Load Step Selection...*Selected Job Steps:***necking****OK****Apply**

The analysis job will take (on average) about 1 to 2 minutes to run. When the job is done there will be a results file titled **necking.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at *necking.log* and *necking.sts* as well as using the *necking.out* file or the **Analysis** option, **Monitor**.

Action:

Object:

A successful run should give the job exit number: **3004**.

13. When the job is finished, import the results.

■ Analysis

Action:

Available Files:

The database will close to allow data transfer to the PATRAN database. When it opens again the results will be in.

14. Post the group **fem** only.

Group/Post ...

Select Groups to Post:

15. Change the display for postprocessing.

■ Results

Select the **Deformation Attributes** icon



Scale Interpretation

● **True Scale**

Scale Factor:

1.0

Show Undeformed (OFF)

16. Create a deformed plot of the last analysis step. Start by clicking on the **Select Results** icon.



Action:

Create

Object:

Quick Plot

Select Result Case:

pick the last result case

Select Fringe Result:

(none)

Select Deformation:

Displacement, Translation

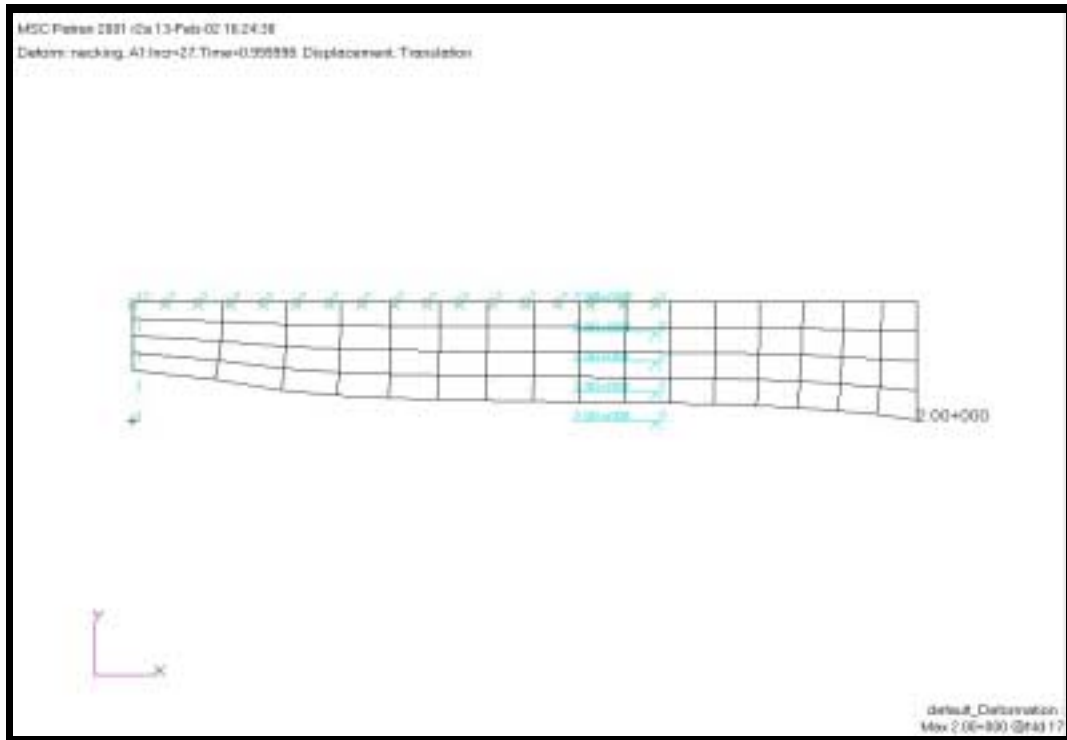
Apply

Your plot should look like the one shown in Figure 11.7. You may need to zoom out to see the whole model. Use the toolbar **Zoom Out** Icon.



Zoom Out

Figure 11.7 - Deformation result of pulled bar



17. Plot the plastic strains as a function of increment.

In this step, you will select 8 elements to plot their plastic strains as a function of load increment. The eight elements will start at the necking point and go toward the fixed end as shown in Figure 11.8:

First, clean up the display using the following toolbar icons:

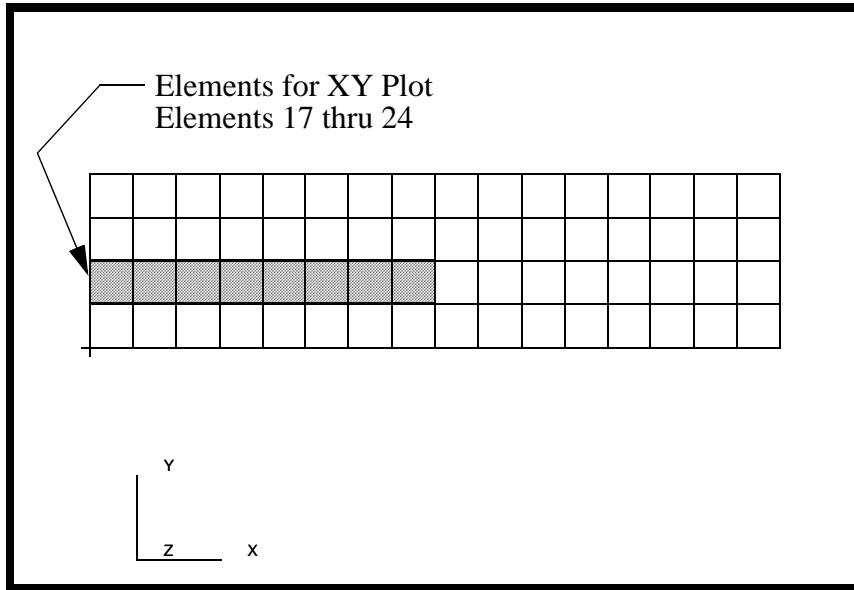


Reset Graphics



Fit View

Figure 11.8 - Elements to select for XY plot of plastic strains



Action:

Create

Object:

Graph

Method:

Y vs X

Click on **View Results** and then **Select Subcase**



Filter Method

All

Filter

Apply

Close

Y:

Result

Select Y Result

Strain, Plastic

Quantity:

X Component

X:

Global Variable

Variable

Increment

Click on the **Target Entities** icon and select



Target Entity

Elements

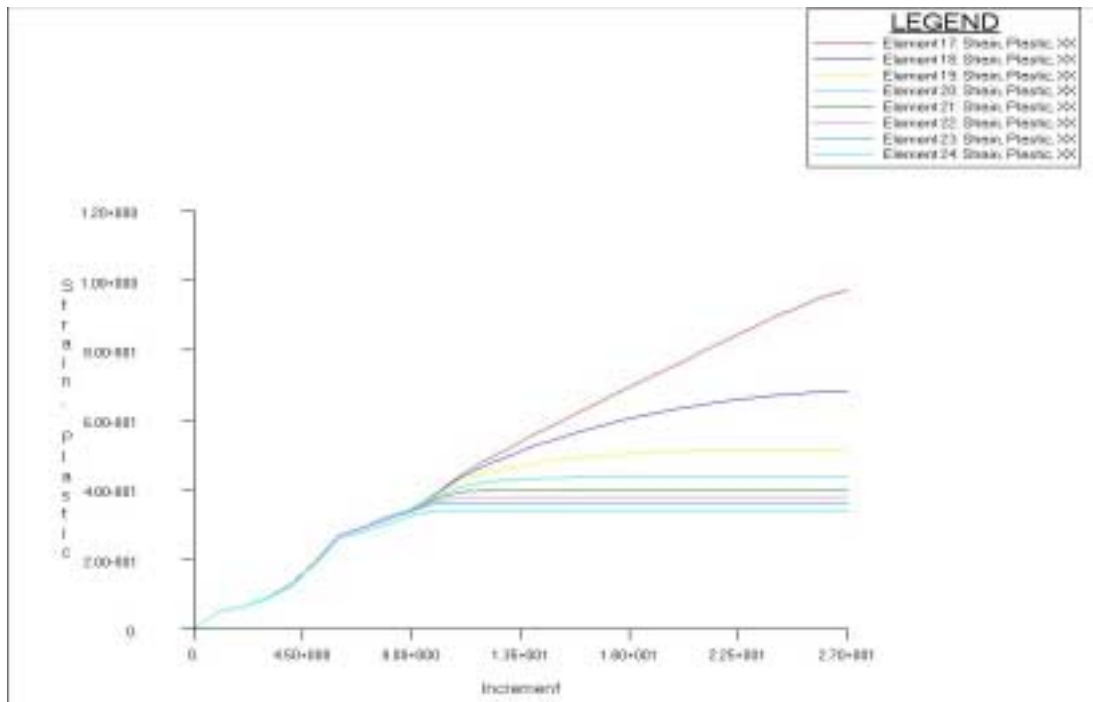
Select Elements

Elm 17:24

Apply

Your plot should look like the one in Figure 11.9:

Figure 11.9 - XY plot of plastic strain components for selected elements



Note that most of the plastic strain occurs at the middle element. This element acts as a “load fuse” to absorb all of the model deformation and hence its area reduces (i.e. necking).

Close the database and quit PATRAN.

This concludes this exercise.