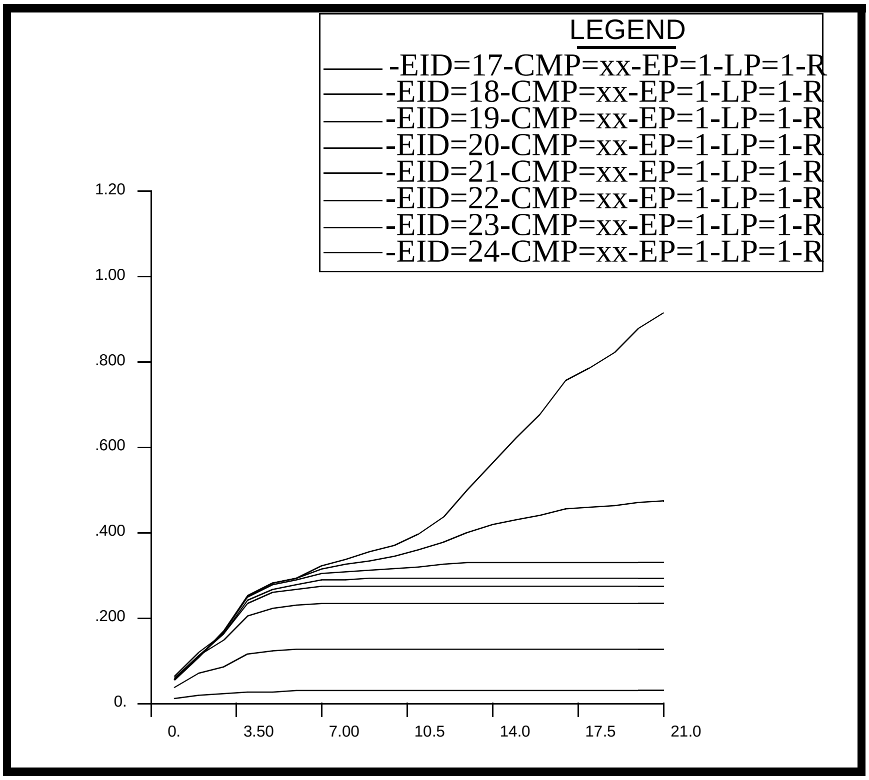
LESSON 8

Necking of 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/ADVANCED_FEA**

OK

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

◆ **Geometry**

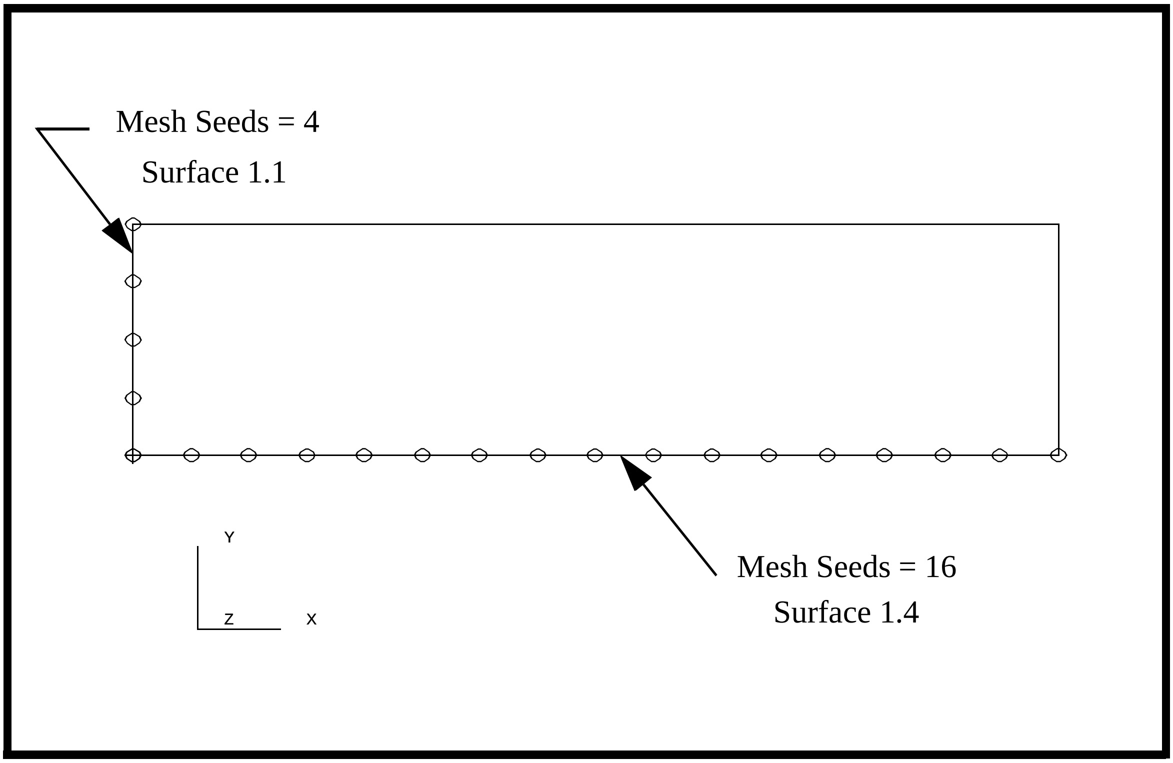
Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinates List:	**<4, 1, 0>**
Origin Coordinates List:	**[0, 0, 0]**

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 4.1:

Figure 4.1 - Mesh seed locations



◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**Uniform**

◆ **Number of Elements**

Number:	**4**
Curve List:	left edge, see Figure 4.1

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.

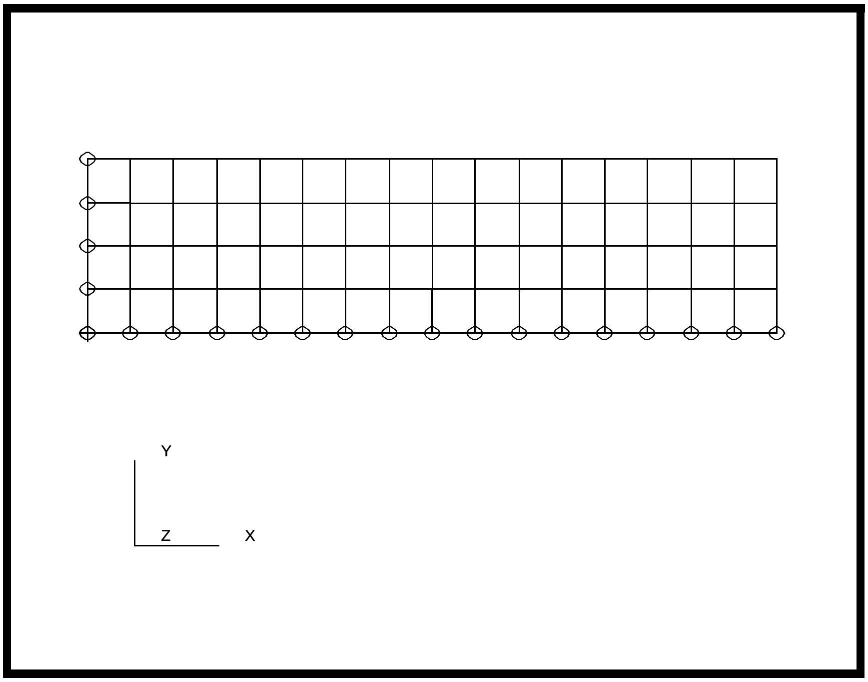
◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Element Topology:	**Quad4**
Surface List:	**Surface 1**

Apply

Your model should now appear as shown in Figure 4.2:

Figure 4.2 - 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)**

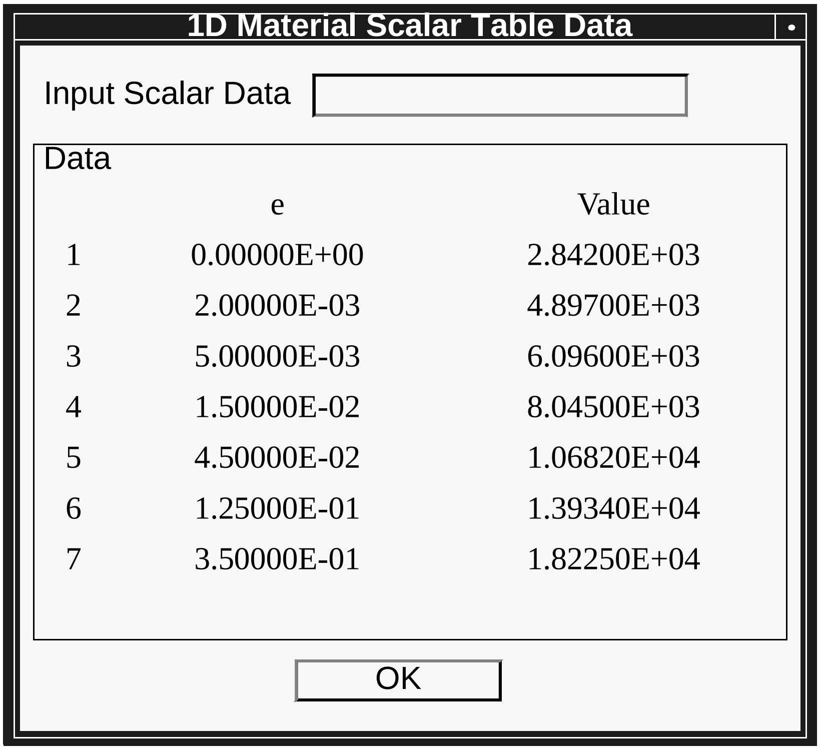
Input Data...

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	Value
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

OK

OK

Apply

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

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**aluminum_1100**

Input Properties...

Constitutive Model:	**Elastic**
Elastic Modulus:	**1E7**
Poisson's Ratio:	**0.33**

Apply

Now to create the plastic material properties.

Constitutive Model:	**Plastic**
Hardening Rule:	**Isotropic**
Stress vs. Plastic Strain:	**plastic_s_e**

Apply

Cancel

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

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**test_specimen**
Options:	
	Plane Stress

	Standard Formulation
Input Properties...	
Material Name:	**aluminum_1100**
Thickness:	**1.0**
OK	
Select Members:	**Surface 1**
Add	
Apply	

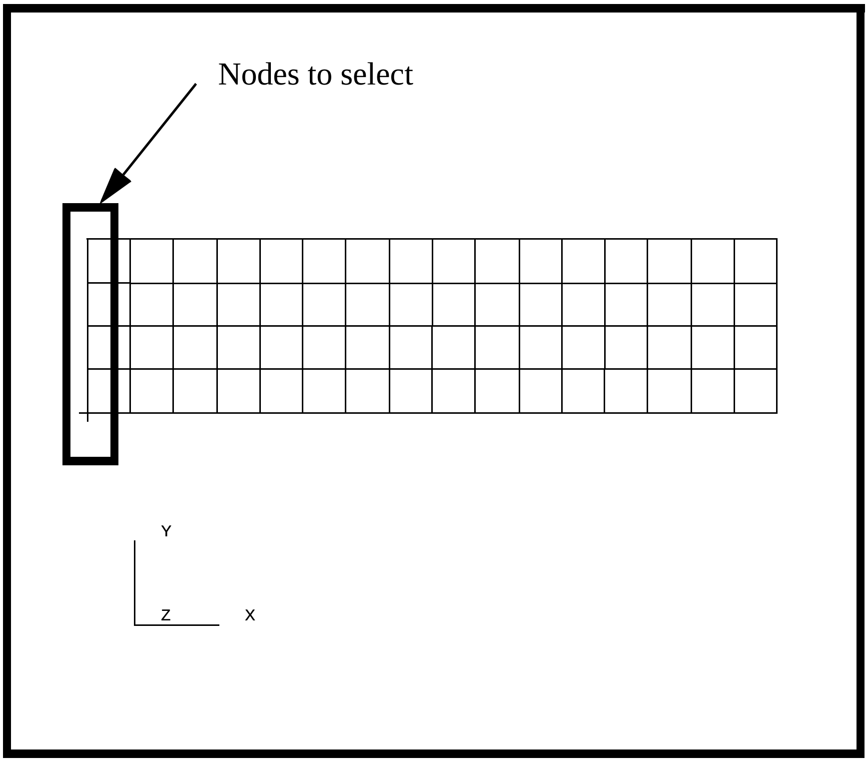
9. Create the load to fix the nodes on the left edge of the surface in the x direction.

◆ **Load/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**symmetry_vertical**
Input Data...	
Translations <T1 T2 T3>:	**<0, , >**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	see Figure 4.3

Click in the *Select Nodes* databox and select the five nodes on the left edge of the surface as shown Figure 4.3:

Figure 4.3 - Nodes to select for symmetry BC



Add

OK

Apply

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

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**pull_at_end**

Input Data...

Translations <T1 T2 T3>:	**<2.0, 0, >**
Rotational <R1 R2 R3>:	**< >**

OK

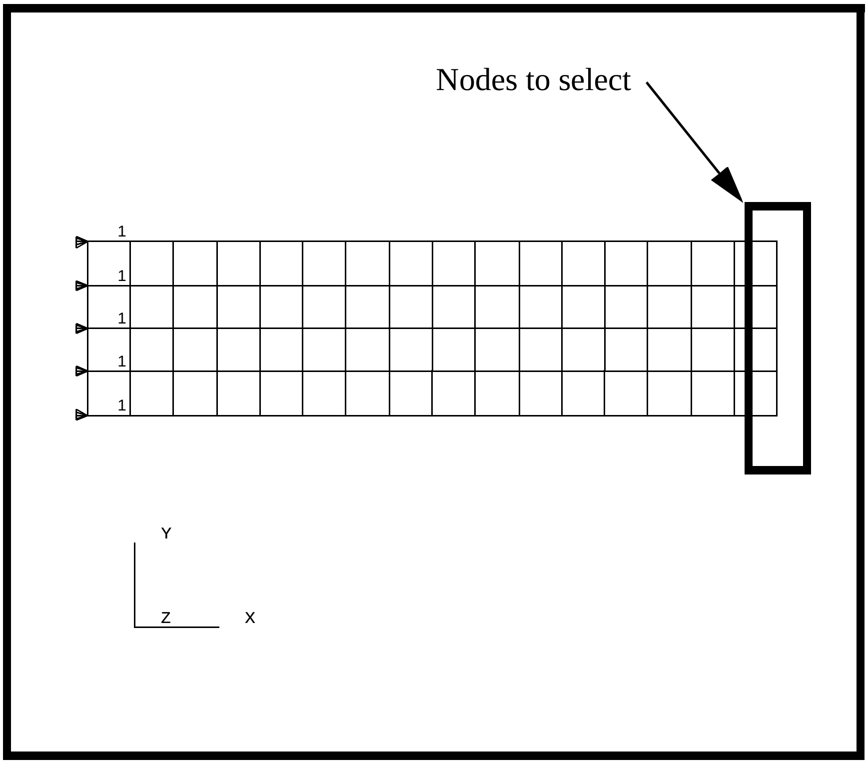
Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: see Figure 4.4

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

Figure 4.4 - Nodes at pulled end of bar



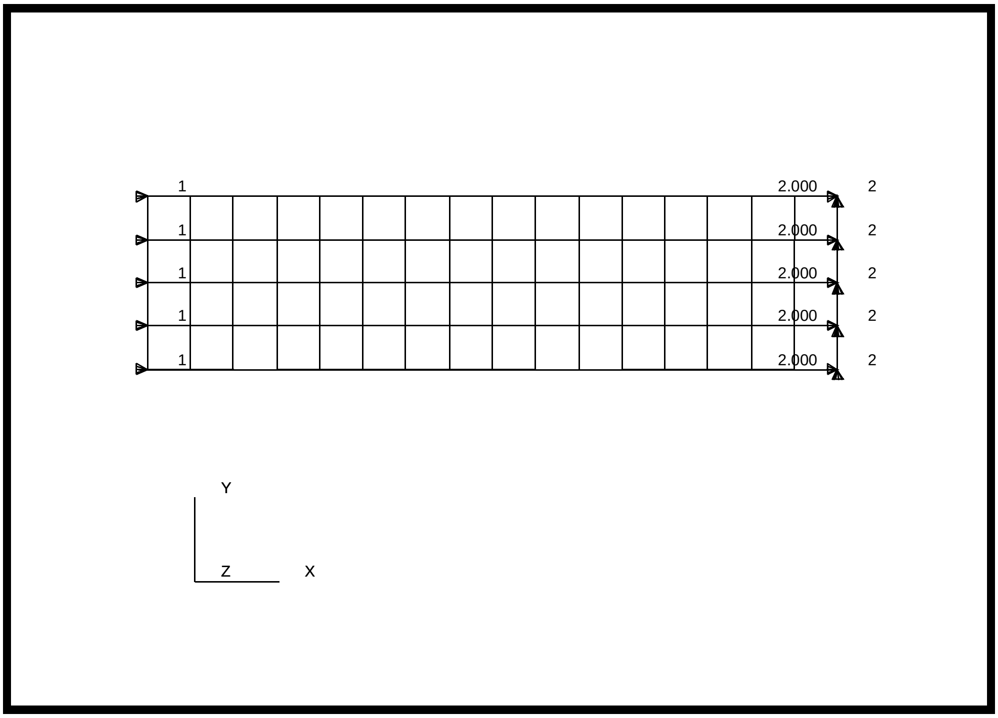
Add

OK

Apply

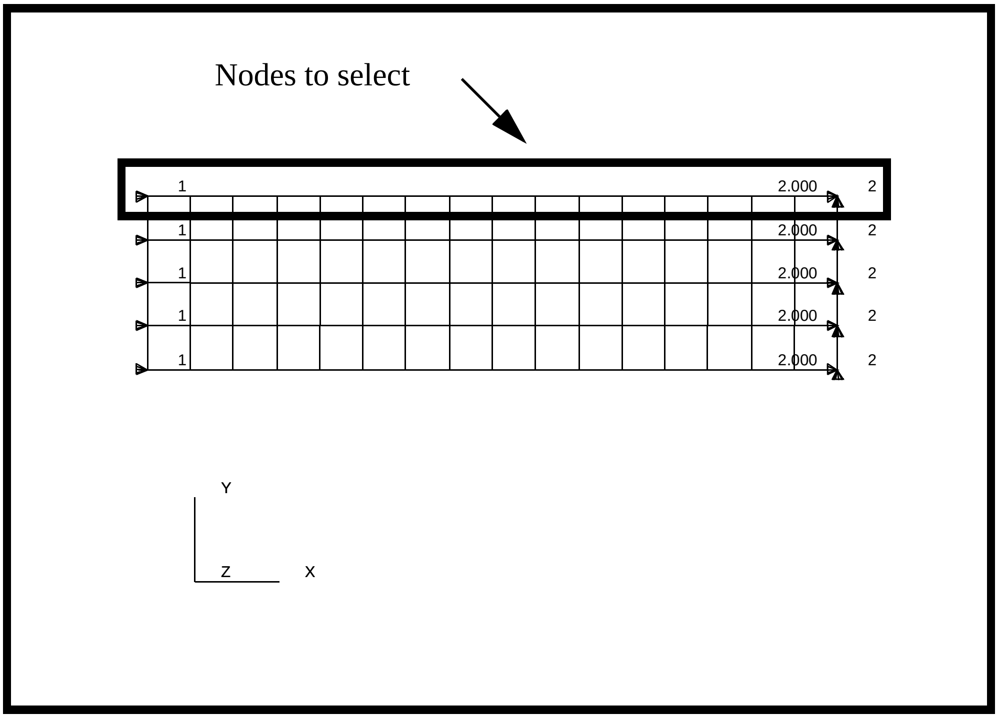
Your model should now appear as shown in Figure 4.5:

Figure 4.5 - 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 , >**
Rotational <R1 R2 R3>:	**< >**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	see Figure 4.6

Figure 4.6 - 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

Step Creation...

Job Step Name:	necking
Solution Type:	**Nonlinear Static**

Solution Parameters...

Large Deflection/Strains:	**ON**
Max No of Increments Allowed:	**100**
Automatic Load Increment:	**ON**
Time Duration of Step:	**1.0**

OK

Output Request...

Stress Invariants:	**Integ Point**
Plastic Strains:	**Integ Point**
Elastic Strains:	**Integ Point**

OK

Apply

Cancel

12. Select the analysis step and submit it.

Be sure to deselect the Default static step.

Step Selection...

Selected Job Steps:	**necking**

Apply

Apply

The analysis job will take (on average) about 2 to 5 minutes to run. When the job is done there will be a results file titled **necking.fil** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at **necking.msg** and **necking.sta** as well as using the UNIX command ***ps -a*** and ***tail necking.msg***.

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

◆ **Analysis**

Action:	**Read Results**

Select Results File...

Available Files: **necking.fil**

OK

Apply

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: **fem**

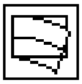
Apply

Cancel

15. Change the display for postprocessing.

◆ **Results**

Select the **Deformation Atrributes** icon

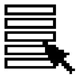


Scale Factor: **1.0**

Scale Interpretation ◆ **True Scale**

❐ **Show Undeformed**

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



Action: **Create**

Object: **Quick Plot**

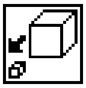
Select Result Case: pick the last result case

Select Fringe Result: (none)

Select Deformation: **Deformation, Displacement**

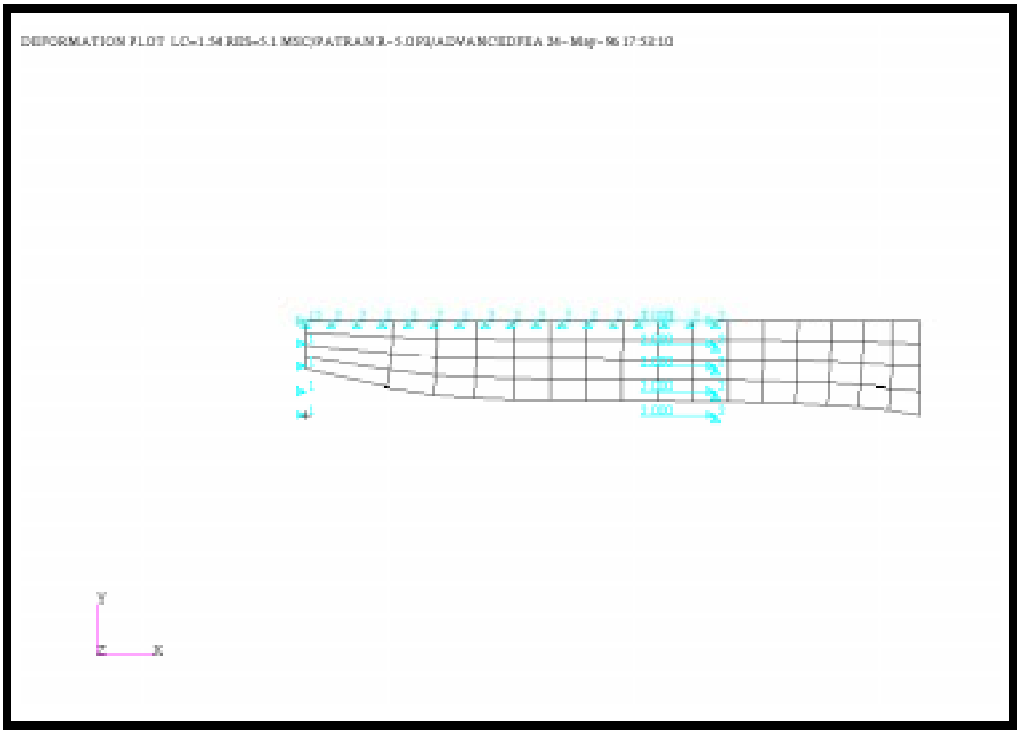
Apply

Your plot should look like the one shown in Figure 4.7. You may need to zoom out to see the whole model. Use the toolbar **Zoomout** Icon.



Zoom Out

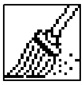
Figure 4.7 - Deformation result of pulled bar

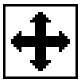


17. Plot the plastic strains as a function of increment.

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 4.8:

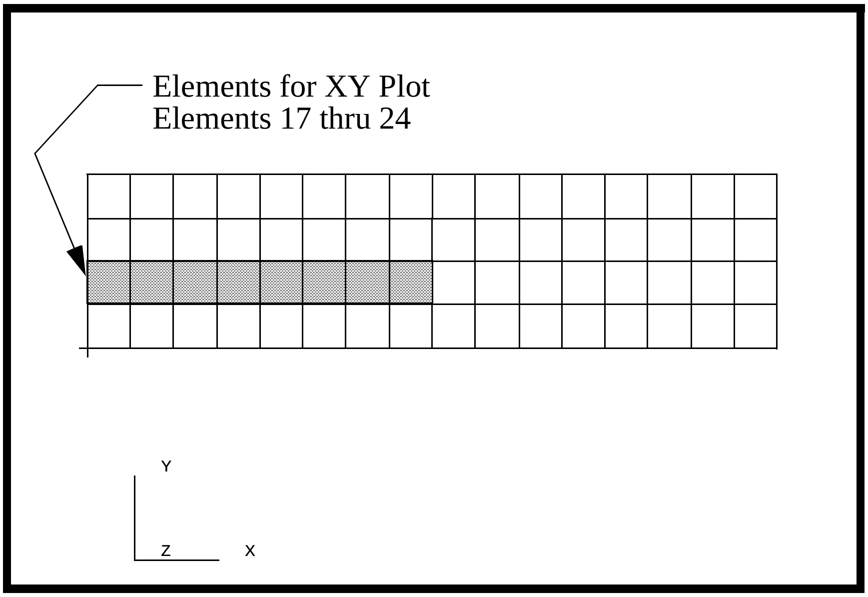
First, clean up the display using the following toolbar icons:





Reset Graphics **Fit View**

Figure 4.8 - Elements to select for XY plot of plastic strains



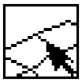
Action:	**Create**
Object:	**Graph**
Method:	**Y vs X**

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

Filter Method	**All**
Filter	
Apply	
Close	
Y:	**Result**
Select Y Result	**Plastic Strain, Components**
Quantity:	**X Component**
X:	**Global Variable**
Variable	**Increment**

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

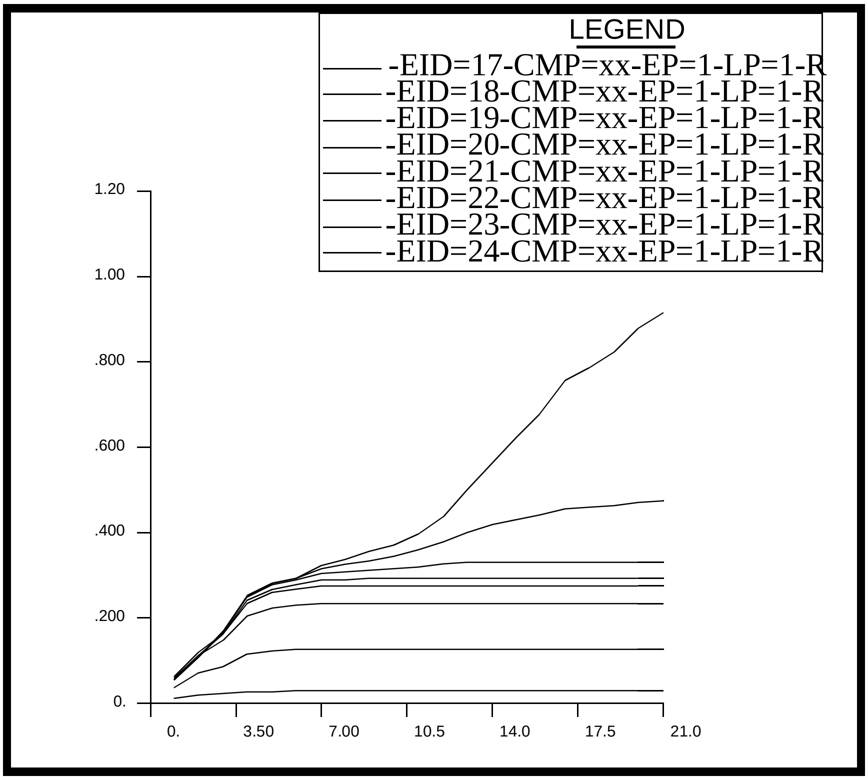


Target Entity	**Elements**
Select Elements	**Elm 17:24**

Apply

Your plot should look like the one in Figure 4.9:

Figure 4.9 - 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.