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

◆ Fields

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Object:</td>
<td>Material Property</td>
</tr>
<tr>
<td>Method:</td>
<td>Tabular Input</td>
</tr>
<tr>
<td>Field Name:</td>
<td>plastic_s_e</td>
</tr>
<tr>
<td>Active Independent Variable:</td>
<td>Temperature (T)</td>
</tr>
<tr>
<td></td>
<td>Strain (e)</td>
</tr>
</tbody>
</table>

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

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

<table>
<thead>
<tr>
<th>Strain</th>
<th>Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>2842</td>
</tr>
<tr>
<td>0.002</td>
<td>4897</td>
</tr>
<tr>
<td>0.005</td>
<td>6096</td>
</tr>
<tr>
<td>0.015</td>
<td>8045</td>
</tr>
<tr>
<td>0.045</td>
<td>10682</td>
</tr>
<tr>
<td>0.125</td>
<td>13934</td>
</tr>
<tr>
<td>0.350</td>
<td>18225</td>
</tr>
<tr>
<td>1.000</td>
<td>23969</td>
</tr>
</tbody>
</table>

**1D Material Scalar Table Data**

<table>
<thead>
<tr>
<th>Data</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>e</td>
<td>Value</td>
</tr>
<tr>
<td>1</td>
<td>2.84200E+03</td>
</tr>
<tr>
<td>2</td>
<td>4.89700E+03</td>
</tr>
<tr>
<td>3</td>
<td>6.09600E+03</td>
</tr>
<tr>
<td>4</td>
<td>8.04500E+03</td>
</tr>
<tr>
<td>5</td>
<td>1.06820E+04</td>
</tr>
<tr>
<td>6</td>
<td>1.39340E+04</td>
</tr>
<tr>
<td>7</td>
<td>1.82250E+04</td>
</tr>
</tbody>
</table>
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

Now to create the plastic material properties.

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

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
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, , >  

*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:
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...**
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*

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

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...**
12. Select the analysis step and submit it.
Be sure to deselect the Default static step.

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...
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.

15. Change the display for postprocessing.

◆ Results

Select the **Deformation Attributes** icon

![Deformation Attributes 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.

![Select Results icon]

**Action:** Create

**Object:** Quick Plot

**Select Result Case:** pick the last result case

**Select Fringe Result:** (none)
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.

**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

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 Entities icon]

<table>
<thead>
<tr>
<th>Target Entity</th>
<th>Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select Elements</td>
<td>Elm 17:24</td>
</tr>
</tbody>
</table>

Your plot should look like the one in Figure 4.9:

**Figure 4.9 - XY plot of plastic strain components for selected elements**

![Graph showing plastic strain components]

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.