Uniaxial Tension Testing

Introduction

Uniaxial tension is used, like cantilever beam bending, to determine the Young’s Modulus of a material. Uniaxial tension testing is most effective when performed on a homogeneous material, such as steel or aluminum. Young’s Modulus is the slope of the line created by stress-strain coordinate pairs (see the inflation experiment for a review of the definitions of stress and strain). The uniaxial test is performed by clamping a long, thin specimen of a material at either end and pulling it apart until failure. The machine will track both the force applied and the displacement at any given time.

Test Procedure

1. Obtain your testing specimen
2. Measure the cross-sectional area of your specimen
   a. If it is a rectangular bar, use width and thickness near the center and use $A = w \times t$
   b. If it is a circular rod, measure diameter near the center and use $A = \pi r^2$
3. Measure the length of the thin segment (excluding the thicker end portions)
4. Mount your specimen in the jaws of the UTM
5. Test until failure, print out results (should look like below)

| Plastic response region / failure |
|-------------------|-----------------|-----------------
| Force (N)          | Jaw slippage region |
|                   | Elastic / Linear response region |
|                   | Displacement (mm) |

6. Use a ruler and calculator to determine two force-displacement pairs in the linear region of the graph
7. Convert force into stress by dividing by the calculated cross-sectional area (stress = force / area)
8. Convert displacement into strain by strain = displacement / original length
9. Find the Young’s Modulus by $YM = (stress \#2 – stress \#1)/(strain \#2 – strain \#1)$
10. Look up a published Young’s Modulus and Poisson Ratio for this material online

<table>
<thead>
<tr>
<th>Surface Area</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td></td>
</tr>
<tr>
<td>Force #1</td>
<td></td>
</tr>
<tr>
<td>Force #2</td>
<td></td>
</tr>
<tr>
<td>Displacement #1</td>
<td></td>
</tr>
<tr>
<td>Displacement #2</td>
<td></td>
</tr>
<tr>
<td>Stress #1</td>
<td></td>
</tr>
<tr>
<td>Stress #2</td>
<td></td>
</tr>
<tr>
<td>Strain #1</td>
<td></td>
</tr>
<tr>
<td>Strain #2</td>
<td></td>
</tr>
<tr>
<td>Calculated Young’s Modulus</td>
<td></td>
</tr>
<tr>
<td>Published Young’s Modulus</td>
<td></td>
</tr>
<tr>
<td>Published Poisson Ratio</td>
<td></td>
</tr>
</tbody>
</table>

**Simulation**

1. Open Abaqus CAE (it takes a minute to load, just be patient)
2. Select Create Model Database
3. In the model tree on the left side of the screen, right click on PARTS and select CREATE
   a. Choose 3D
   b. Choose Deformable
   c. Choose Solid
   d. Choose Extrusion
4. You will now sketch the part
   a. Choose the rectangle shape
   b. Enter 0,0 as the starting corner and hit ENTER
   c. In the same box you will be prompted for the opposite corner point of your rectangle
      i. You are re-creating your specimen that you broke in the tension testing machine, so enter the point as (width, thickness)
      ii. Hit ENTER again
      iii. Click the red X and then DONE to say you are finished sketching the rectangle
      iv. You will be prompted for the extrusion depth, enter in the value you recorded for length. Click OK.
5. Again in the Model Tree on the left, right click on MATERIALS and select CREATE
   a. At the top of the new box, replace Material-1 with a new name that you will remember
   b. Select MECHANICAL then ELASTICITY then ELASTIC
      i. Enter in the value for Young’s Modulus for the specimen your group tested
      ii. Enter in a Poisson Ratio from published values online
      iii. Click OK
6. Back to the Model Tree, right click on SECTIONS and choose CREATE
   a. Again name the section something you will remember
b. Choose SOLID

c. Choose HOMOGENEOUS

d. Click CONTINUE

e. Make sure that the material name that you created is in the drop-down box and then click OK

7. In the Model Tree, click on the “+” next to ASSEMBLY, right click on INSTANCES and select CREATE
   a. Leaving everything the way it is by default, Click OK

8. In the Model Tree click on the “+” next to PARTS and again on the “+” next to your part’s name (Part-1 if you never changed it)
   a. Right click on SECTION ASSIGNMENTS and select CREATE
   b. Move the mouse cursor over the part you created and left click
   c. The part should become highlighted, select DONE at the bottom of the screen
   d. In the box that pops up make sure that the section name that you created is in the drop-down box and Click OK

9. In the Model Tree right click on BC’s and select CREATE
   a. Accept all default values and Click CONTINUE
   b. Move the mouse cursor over one of the two small faces of the object
   c. Select DONE
   d. In the box that pops up Choose PINNED and then Click OK

10. In the Model Tree right click on STEPS and select CREATE
    a. Accept the defaults and select CONTINUE
    b. Under NLGEOM select ON
    c. Change the TIME PERIOD to 10
    d. Click OK

11. In the Model Tree right click on LOADS and select CREATE
    a. MECHANICAL should already be selected
    b. Change the highlighted text on the right to PRESSURE
    c. Click CONTINUE
    d. You will be prompted to choose the face that the pressure acts on, select the other small face that was not chosen as the BC face and Click DONE
    e. Under magnitude enter a negative value load somewhere between the two values you used for calculating the Young’s Modulus (make sure it is negative) and Click OK

12. In the Model Tree under PARTS double click on MESH (the part should change color, if it doesn’t double click again)
    a. On the top menu, select SEED and then PART
    b. Click OK
    c. Also on the top menu select MESH and then PART
    d. On the bottom of the screen Click YES

13. Save your file now (if you have made mistakes it often crashes the program on the next step)

14. Back in the Model Tree right click on JOBS and select CREATE
    a. Give the job a name you will recognize and select CONTINUE
    b. Accept all the defaults and Click OK
    c. Now click on the “+” beside JOBS, right click on your job name and select SUBMIT
d. After a few minutes of processing (you may get errors, but it should be ok) right click on your job name again and select RESULTS

e. Once in Results, the icon that is Colored and Bent

f. On the top menu click RESULTS and then FIELD OUTPUT
   i. Select MAX PRINCIPLE Stress
   ii. You are now looking at the stress distribution of the simulated root at the max pressure

You can watch the tension test through all the time-steps by selecting the ANIMATE: SCALE FACTOR icon (you may want to go to the ANIMATION OPTIONS first and slow down the speed)