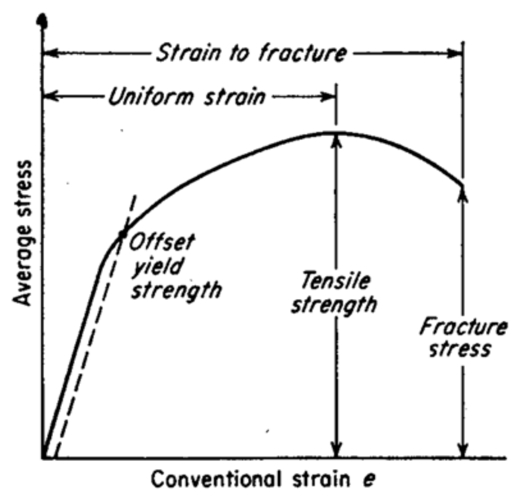
Week 11 Application Activity
ME219 Computer Programming for Engineers

A tensile test is often used to evaluate the mechanical characteristics of different materials. A typical tensile test involves using a machine to pull on a material specimen while measuring its tensile load (F) and deformation (ΔL). The test continues until the specimen breaks, and the resulting load and deformation data is converted to axial stress (σ) and strain (ε):

$$\sigma = \frac{F}{A} = \frac{4F}{\pi d^2} \quad (1) \qquad\qquad \varepsilon = \frac{\Delta L}{L_o} \quad (2)$$

where d is the specimen's diameter and L_o is its original length (called the gauge length), which are 0.25 in and 3 in, respectively. In a (ε, σ) plot, many materials will exhibit the behavior shown below, where the stress-strain relationship is linear at low levels of strain and nonlinear at higher levels. The linear region represents the range of deformations at which the material will return to its original shape if unloaded – this is often called the linear elastic region. The slope of this line is called the modulus of elasticity (E), which is a measure of the material's stiffness.



The yield point represents the (ε, σ) point at which the material will no longer return to its original shape. Yield stress σ_{yield} is usually defined by creating a line parallel to the linear elastic region that is offset by 0.2 percent strain. The stress at which this offset line intersects the original (ε, σ) plot is defined as σ_{yield}. The ultimate tensile stress (σ_{ult}) is the maximum stress experienced by the specimen, and is followed by a decrease in stress that occurs prior to breaking.

In this exercise, you will use information in a (ε, σ) plot to determine E, σ_{yield}, and σ_{ult} for 3 different materials (aluminum, brass, and steel). A file corresponding to each material is provided, and includes 2 columns that contain elongation and load data, respectively.

1. Start MATLAB, navigate to the directory where you want to save your work, and start a new m-file called “W11.m”. Your program should do the following:

2. **PART 1**: Create variables to define d and L_o.

3. **PART 2**: Using the ‘input’ command, prompt the user for the filename of the material he/she wants to analyze (i.e. ‘aluminum.txt’, ‘brass.txt’, or ‘steel.txt’). Then, use the ‘load’ command to load that file and save as a variable called ‘data’. The 2 columns of the resulting ‘data’ matrix can then be saved as separate column vectors representing elongation and load.

4. **PART 3**: Use F, d, ΔL, and L_o to calculate stress and strain according to equations (1) and (2).

5. **PART 4**: Calculate E by doing the following:
   a. Create a plot of strain versus stress
   b. Have the user select two points in the linear elastic region – this can be done with ‘ginput’:

      ```
      [x,y] = ginput(2);
      ```

   c. E can now be calculated by dividing the difference in the resulting two y values by the difference in x values.

6. **PART 5**: Calculate the values of your offset line by doing the following:
   a. Create a row vector containing the slope and intercept of the offset line:

      ```
      c = [E -0.002*E];
      ```

   b. Use the ‘polyval’ command to calculate a vector ‘oline’ containing offset line values for each value of strain:

      ```
      oline = polyval(c,str);
      ```

7. **PART 6**: Determine yield stress by finding the intersection point of ‘oline’ and the stress-strain curve:
   a. Use the ‘find’ command to determine which values of ‘oline’ are greater than their corresponding values of stress. This will return a vector.
   b. Define the first value of this vector as ‘iy’ – this is the location of yield stress in your stress vector.
   c. Define yield stress using ‘iy’, e.g.

      ```
      YS = stress(iy);
      ```

8. **PART 7**: use the 'max' command to find the magnitude ('US') and location ('iUS') of ultimate stress.

9. **PART 8**: Create a new plot of the stress-strain curve that includes the following:
    a. Your stress-strain curve and offset line, along with 'o' markers identifying yield and ultimate stresses. This can be done in a single 'plot' command via:

```
plot(str,stress,str(1:iy+10),oline(1:iyi+10),str(iy),YS,'o',str(iUS),US,'o')
```

    b. Include axes labels using 'xlabel' and 'ylabel' that identify what is plotted on each axis. Be sure to include units.
    c. Place a title on your figure. This can be done with the variable containing the data filename, e.g.

```
title(fname)
```

    d. Include a figure legend using 'legend' to label the 4 items you plotted in part a: stress-strain curve, offset line, and markers for yield and ultimate stresses.
    e. Use the following 'gtext' command to place values of E, YS, and US on your plot:

```
gtext([{'Modulus = ' num2str(E) ' psi'},...
{'Yield Stress = ' num2str(YS) ' psi'},...
{'Ultimate Stress = ' num2str(US) ' psi'}])
```

10. **PART 9**: When your code is complete, run it once for each of the 3 materials. When you obtain the plot for each, save it as a .fig file by clicking file>save as. Then, do the following:
    a. Open all 3 saved figure files for inspection
    b. Compare the modulus, yield stress, and ultimate stress among the 3 materials, and write a brief description of the differences you see in your m-file as a comment.
    c. Comment the rest of your code to identify what each part is doing.

When finished, please do the following:

1. Bundle your m-file and 3 figure files into a single .zip file called W11.zip.
2. Upload your .zip file to Blackboard using the link provided.