Lab 2: Tapered Bar

**Keywords:** 1D elasticity, convergence

### 2.1 Problem Statement and Objectives

A tapered bar subjected to an axial load will be analyzed in order to predict the distributions of stress and displacement in the bar. The geometrical, material, and loading specifications for the bar are given in Figure 2.1. The thickness of the bar is $2h$ inches, where $h(x)$ is described by the equation: $h(x) = 4 - 0.6x + 0.03x^2$

- **Geometry:**
  - Length: $L=10”$
  - Width: $b=1”$ (uniform)
  - Thickness: $2h$ (a function of $x$)

- **Material:**
  - Steel
  - Yield Strength: 36 ksi
  - Modulus of Elasticity: 29 Msi
  - Poisson’s Ratio: 0.3
  - Density = 0.0088 slugs/in$^3$

- **Loading:**
  - Axial Load: $P=10,000$ lbs

![Image of a tapered bar with dimensions and load](Image)

*Figure 2.1 Geometry, material, and loading specifications for a tapered bar.*

### 2.2 Analysis Assumptions

- The bar is thin in the width (out-of-plane) direction and all loading is in the plane, so a state of plane stress can be assumed.
- Because the bar is tapered, the stress and strain state is not uniform along the length of the bar.
- The taper in the bar also gives rise to non-zero stresses in the thickness direction, but these stress components should be small compared to the axial stresses. Therefore, stresses in the thickness direction will be neglected.
2.3 Mathematical Idealization

Based on the assumptions above, a 1D elasticity analysis will be performed. In this model, the main axis of the bar is discretized using linear, two-noded, 1D bar/truss finite elements having a uniform thickness within each element. Thus, the geometry is idealized as having a piecewise constant cross-section, as shown in Figure 2.2. The thickness assigned to each element is taken to be equal to the actual thickness of the tapered bar at the x-coordinate corresponding to the centroid of that element. The equation \( h(x) \) is used to determine these element thicknesses.

![Figure 2.2 Idealized geometry for a tapered bar.](image)

2.4 Finite Element Model

The 1D analysis should be performed four times, each with a different mesh. Meshes consisting of 1, 2, 3 and 4 bar elements should be developed in order to study convergence of the solution, as the taper is represented better with a greater number of elements.

Note that 1D bar elements are actually 3D elements with three degrees of freedom at each node (U, V, W), which are the displacements in the x, y and z directions, respectively. So the finite element system has six possible rigid body modes that must be restrained prior to obtaining a solution. All of the V (y-direction) and W (z-direction) degrees of freedom should be restrained in this problem.

2.5 Model Validation

Simple hand calculations can be performed to estimate the stresses and deflections in this bar structure. The results of these calculations should be used to assess the validity of the finite element results (i.e., to make sure that the finite element results are reasonable and do not contain any large error due to a simple mistake in the model).

The displacement at the end of the bar can be approximated by assuming the bar is of uniform cross-sectional shape. The cross-sectional shape used in this calculation may be, for example, the
cross-sectional shape at the mid-point of the bar (at \( x=5'' \)). Then the displacement can be estimated using the well-known relation:

\[
\delta = \frac{PL}{EA}
\]

where \( \delta \) is the tip displacement of the bar, \( A \) is the (uniform) cross-sectional area, and the other parameters are defined in Figure 2.1.

The axial stress at any cross-section in the bar can be estimated by neglecting all other stress components and assuming that the axial stress is uniformly distributed over the cross-section. This assumption is not strictly valid for a tapered bar, but such an assumption should allow a reasonably accurate solution to be obtained for the purpose of validation. From equilibrium, it is found that the resultant force at any cross-section is \( P \), so the axial stress can be estimated using the relation:

\[
\sigma = \frac{P}{A}
\]

where \( A \) is the actual cross-sectional area at the section under consideration.

### 2.6 Post Processing

A total of four finite element models should be developed. Based on the results of these analyses, perform and submit the following post-processing steps.

1. Complete the following table:

<table>
<thead>
<tr>
<th>Model ID</th>
<th>Tip Displacement</th>
<th>Stress at ( x = 5.0 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1D – one element</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D – two elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D – three elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1D – four elements</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Validation hand calculation</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

2. Create an Excel plot of the distribution of displacement along the \( x \)-axis as predicted by the four models. Put all of the results on a single plot so that comparisons among the solutions can be made.

3. Create an Excel plot of the distribution of axial stress along the \( x \)-axis as predicted by the four models. Put all of the results on a single plot so that comparisons among the solutions can be made.

4. Comment on the convergence of displacement and stress in the 1D elasticity solutions as the number of elements is increased.

5. Comment on the validity of the solutions. Show the hand calculations.