Lab 4: Tapered Beam

Keywords: beam, convergence

4.1 Problem Statement and Objectives

A tapered beam subjected to a tip bending load will be analyzed in order to predict the distributions of stress and displacement in the beam. The geometrical, material, and loading specifications for the beam are given in Figure 4.1. The thickness of the beam is $2h$ inches, where $h(x)$ is described by the equation: $h(x) = 4 - 0.6x + 0.03x^2$

<table>
<thead>
<tr>
<th>Geometry:</th>
<th>Material: Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length: L=10”</td>
<td>Yield Strength: 36 ksi</td>
</tr>
<tr>
<td>Width: b=1” (uniform)</td>
<td>Modulus of Elasticity: 29 Msi</td>
</tr>
<tr>
<td>Thickness: 2h (a function of x)</td>
<td>Poisson’s Ratio: 0.3</td>
</tr>
<tr>
<td>Density = 0.0088 slugs/in$^3$</td>
<td>Loading: Transverse Load: Q=10,000 lbs</td>
</tr>
</tbody>
</table>

![Figure 4.1 Geometry, material, and loading specifications for a tapered beam.](image)

4.2 Analysis Assumptions

- The beam 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 beam is tapered, the stress and strain states are not uniform along the length of the beam.
- The taper in the beam also gives rise to non-zero normal stresses in the thickness direction, but these stress components should be small compared to the bending stresses in the x-direction. Therefore, normal stresses in the thickness direction will be neglected.
4.3 Mathematical Idealization

Based on the assumptions above, a beam analysis will be performed. In this model, the main axis of the beam is discretized using two-noded beam finite elements having a uniform cross-sectional shape within each element. Thus, the geometry is idealized as having a piecewise constant cross-section, as shown in Figure 4.2. The section properties assigned to each element are derived from the actual thickness of the tapered beam at the x-coordinate corresponding to the centroid of that element. The equation $h(x)$ is used to determine these element thicknesses.

![Figure 4.2 Idealized geometry for a tapered beam.](image)

4.4 Finite Element Model

The beam analysis should be performed four times, each with a different mesh. Meshes consisting of 1, 2, 3 and 4 beam 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 beam elements have six degrees of freedom at each node ($U, V, W, \theta_x, \theta_y, \theta_z$), which are the displacements in the x, y and z directions, respectively and the rotations about the x, y and z axes, respectively. So the finite element system has six possible rigid body modes that must be restrained prior to obtaining a solution. In this problem, we will fix all six degrees of freedom at the base of the beam ($x=0$), which is clamped.

4.5 Model Validation

Simple hand calculations can be performed to estimate the stresses and deflections in this beam 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 deflection at the end of the beam can be approximated by assuming the beam 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 beam (at x=5\textquoteleft\textquoteleft). Then the displacement can be estimated using the well-known relation:

\[ \delta = \frac{PL^3}{3EI} \]

where \( \delta \) is the tip displacement of the beam, \( I \) is the (uniform) second moment of inertia of the cross-sectional area, and the other parameters are defined in Figure 4.1.

The bending stress at any cross-section in the beam can be estimated by using beam theory:

\[ \sigma = \frac{My}{I} \]

where \( I \) is the actual second moment of inertia of the cross-sectional area at the section under consideration.

### 4.6 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 Deflection</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 beam solutions as the number of elements is increased.

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

6. **Please submit a formal lab report for this project.**