Module 1.2: Moment of a 1D Cantilever Beam

Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Problem Description</td>
<td>2</td>
</tr>
<tr>
<td>Theory</td>
<td>2</td>
</tr>
<tr>
<td>Geometry</td>
<td>3</td>
</tr>
<tr>
<td>Preprocessor</td>
<td>6</td>
</tr>
<tr>
<td>Element Type</td>
<td>6</td>
</tr>
<tr>
<td>Real Constants and Material Properties</td>
<td>7</td>
</tr>
<tr>
<td>Meshing</td>
<td>9</td>
</tr>
<tr>
<td>Loads</td>
<td>10</td>
</tr>
<tr>
<td>Solution</td>
<td>13</td>
</tr>
<tr>
<td>General Postprocessor</td>
<td>13</td>
</tr>
<tr>
<td>Results</td>
<td>15</td>
</tr>
<tr>
<td>Validation</td>
<td>16</td>
</tr>
</tbody>
</table>
**Problem Description**

![Beam Diagram]

**Nomenclature:**
- \( L = 110 \text{ m} \)  
  Length of beam
- \( b = 10 \text{ m} \)  
  Cross Section Base
- \( h = 1 \text{ m} \)  
  Cross Section Height
- \( M = 70 \text{kN}\cdot\text{m} \)  
  Applied Moment
- \( E = 70 \text{ GPa} \)  
  Young’s Modulus of Aluminum at Room Temperature
- \( \nu = 0.33 \)  
  Poisson’s Ratio of Aluminum

In this module, we will be modeling an Aluminum cantilever beam with a bending moment loading about the z-axis with one dimensional elements in ANSYS Mechanical APDL. We will be using beam theory and mesh independence as our key validation requirements. The beam theory for this analysis is shown below:

**Theory**

**Von Mises Stress**

Assuming plane stress, the Von Mises Equivalent Stress can be expressed as:

\[
\sigma' = (\sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}}
\]  
(1.2.1)

Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

\( \tau_{xy} = 0, \sigma_y = 0 \).  
(1.2.2)

Using these simplifications, the Von Mises Equivalent Stress from equation 1 reduces to:

\[
\sigma' = \sigma_x
\]  
(1.2.3)

Bending Stress is given by:

\[
\sigma_x = -\frac{M(x)c}{I}
\]  
(1.2.4)
Where \( I = \frac{1}{12}bh^3 \) and \( c = \frac{h}{2} \). From statics, we can derive:

\[
M(x) = M \tag{1.2.5}
\]

\[
\sigma_x = \frac{6M}{bh^2} = 42\text{KPa} \tag{1.2.6}
\]

**Beam Deflection**

The beam equation to be solved is:

\[
\frac{d^2y}{dx^2} = \frac{M(x)}{EI} \tag{1.2.7}
\]

After two integrations:

\[
y = \frac{Mx^2}{2EI} \tag{1.2.8}
\]

With Maximum Deflection at \( x=L \):

\[
y = \frac{ML^2}{2EI} = 7.26\text{mm} \tag{1.2.9}
\]

**Geometry**

**Opening ANSYS Mechanical APDL**

1. On your Windows 7 Desktop click the **Start** button
2. Under **Search Programs and Files** type “ANSYS”
3. Click on **Mechanical APDL (ANSYS)** to start ANSYS. This step may take time.
Preferences

1. Go to Main Menu -> Preferences
2. Check the box that says Structural
3. Click OK

Title:

To add a title

1. Utility Menu -> ANSYS Toolbar -> type /prep7 -> enter
2. Utility Menu -> ANSYS Toolbar -> type /Title, “Title Name” -> enter
Key points

Since we will be using 1D Elements, our goal is to model the length of the beam.

1. Go to **Main Menu -> Preprocessor -> Modeling -> Create -> Keypoints -> On Working Plane**
2. Click **Global Cartesian**
3. In the box underneath, write: 0,0,0. This will create a key point at the origin.
4. Click **Apply**
5. Repeat Steps 3 and 4 for 110,0,0
6. Click **Ok**
7. The **Triad** in the top left corner is blocking keypoint 1. To get rid of the triad, type `/triad,off` in **Utility Menu -> Command Prompt**

8. Go to **Utility Menu -> Plot -> Replot**

Line

1. Go to **Main Menu -> Preprocessor -> Modeling -> Create -> Lines -> Lines -> Straight Line**
2. Select **Pick**
3. Select **List of Items**
4. Type 1,2 for points previously generated.
5. Click **Ok**
The resulting graphic should be as shown:

Saving Geometry

We will be using the geometry we have just created for 3 modules. Thus it would be convenient to save the geometry so that it does not have to be made again from scratch.

1. Go to File -> Save As …
2. Under Save Database to pick a name for the Geometry. For this tutorial, we will name the file ‘1D Cantilever’
3. Under Directories: pick the Folder you would like to save the .db file to.
4. Click OK

Preprocessor

Element Type

1. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete
2. Click Add
3. Click Beam -> 3D Elastic 4
4. Click OK
Real Constants and Material Properties

Now we will add the thickness to our beam.

1. Go to **Main Menu -> Preprocessor -> Real Constants -> Add/Edit/Delete**
2. Click **Add**
3. Choose **Type 1 Beam4**
4. Click **OK**
5. **Cross-sectional area AREA** enter 10
6. Under **Area moment of inertia IZZ** Enter 10/12 this is shown in Theory under Eq. 1.2.4
7. Under **Thickness along Z axis TKZ** enter 10
8. Under **Thickness along Y axis TKY** enter 1
9. Click **OK**
10. Close out of the **Real Constants** window
Now we must specify Young’s Modulus and Poisson’s Ratio

1. Go to **Main Menu -> Preprocessor -> Material Props -> Material Models**
2. Go to **Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic**

3. Enter $7E10$ for Young’s Modulus (EX) and $.33$ for Poisson’s Ratio (PRXY)
4. Click **OK**
5. **Exit** out of **Define Material Model Behavior**
Mesh

We are going to re-mesh this beam to have ten elements through the length of the beam.

1. Go to Main Menu -> Preprocessor -> Meshing -> Mesh Tool
2. Size Controls -> Global -> Set
3. Set SIZE Element edge length to 11. This will map 1 element every 11 meters down the beam for a total of 10 elements.
4. Click OK
5. Select Mesh -> Lines
6. Click Mesh
7. Click Pick All

Unlike remeshing areas and solids, line meshes must be cleared manually before remeshing is allowed.

8. Go to Main Menu -> Preprocessor -> Meshing -> Clear -> Lines
9. Click Pick All
10. Repeat steps 1-6 to update the mesh.
11. Go to Utility Menu -> Command Prompt and enter /triad, off followed by /replot
Loads

1. Go to Utility Menu -> Plot -> Nodes
2. Go to Utility Menu -> Plot Controls -> Numbering...
3. Check NODE Node Numbers to ON
4. Click OK

The resulting graphic should be as shown:

ANSYS numbers nodes from the left extreme to the right extreme and then numbers from left to right.
Displacement

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads ->Apply ->Structural -> Displacement -> On Nodes

2. Select Pick -> Single -> and click node 1

3. Click OK

4. Under Lab 2 DOFs to be constrained select All DOF

5. Under Value Displacement value enter 0

6. Click OK

The resulting graphic should look as shown below:
Force/Moment

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes
2. Select Pick -> Single -> and click node 2

3. Click OK
4. Under Direction of force/mom select MZ
5. Under VALUE Force/moment value enter 70000
6. Click OK

The resulting graphic should look as shown below:
Solution

Now that our boundary conditions have been specified, it’s time to solve the problem.

1. Go to Main Menu -> Solution -> Solve -> Current LS
2. Click OK
3. If a warning box pops up click Yes

General Postprocessor

Now that ANSYS has solved the load step, let’s create plots of Y Deflection and Equivalent Von-Mises Stress.

Displacement

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solu
2. Go to Nodal Solution -> DOF Solution Y-Component of displacement
3. Click OK
4. To give the graph a title, go to Utility Menu -> Command Prompt and type /title, Deflection of a Cantilever Beam with a Moment
   followed by the return key and the command /replot

The resulting graph should look as shown below:
Equivalent (Von-Mises) Stress

Unfortunately, we cannot create a contour plot of Von-Mises stress for 1D elements unless more complicated loading conditions are applied. We can, however, look up the moment reactions at each element. If we plug this value into equation 1.3.4, we can readily calculate the bending stress in our model and by extension, the equivalent stress.

1. Go to Utility Menu -> List -> Results -> Element Solution …
2. Go to Element Solution -> All Available force items
3. Click OK

This chart shows all reaction forces and moments at each node in the domain. Since we are interested in reaction moments in the z direction, we will look to the last column in the chart:
Results

Max Deflection Error

\[ \delta_{\text{max}} = 7.26 \text{mm} \]

The percent error (\(\%E\)) in our model max deflection can be defined as:

\[
\%E = \left( \frac{\delta_{\text{theoretical}} - \delta_{\text{model}}}{\delta_{\text{theoretical}}} \right) \times 100 = 0\%
\]  

This shows that there is no error baseline using one dimensional elements.

Max Equivalent Stress Error

\[ \sigma_{\text{max}} = 42 \text{kPa} \]

Using equation (1.2.11) above, the percent error for Equivalent Stress in our model is 0%. This is due to the fact that ANSYS uses Gaussian Quadrature to interpolate between the integration points. This changes with respect to the element used. Since Beam4 has two integration points and two-point Gaussian Quadrature is fourth degree accurate, the answer will have no error baseline because this function is first degree. Fourth degree functions are accurate to third degree polynomials. Thus the one dimensional method has zero percent error in deflection and stress. The 1D method is both fast and accurate.
Validation:

Deflection vs. Length of Beam

Equivalent Stress vs. Length of Beam