Module 1.5: Moment Loading of a 2D Cantilever Beam

Table of Contents

<table>
<thead>
<tr>
<th>Topic</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>4</td>
</tr>
<tr>
<td>Preprocessor</td>
<td>7</td>
</tr>
<tr>
<td>Element Type</td>
<td>7</td>
</tr>
<tr>
<td>Real Constants and Material Properties</td>
<td>8</td>
</tr>
<tr>
<td>Meshing</td>
<td>9</td>
</tr>
<tr>
<td>Loads</td>
<td>10</td>
</tr>
<tr>
<td>Solution</td>
<td>14</td>
</tr>
<tr>
<td>General Postprocessor</td>
<td>14</td>
</tr>
<tr>
<td>Results</td>
<td>15</td>
</tr>
<tr>
<td>Validation</td>
<td>16</td>
</tr>
</tbody>
</table>
Problem Description

![Diagram of beam with applied moment](image)

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:

$$\sigma' = (\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2)^{\frac{1}{2}} \quad (1.5.1)$$

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

$$\tau_{xy} = 0, \quad \sigma_y = 0. \quad (1.5.2)$$

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

$$\sigma' = \sigma_x \quad (1.5.3)$$

Bending Stress is given by:

$$\sigma_x = -\frac{M(x)c}{I} \quad (1.5.4)$$
Where \( I = \frac{1}{12}bh^3 \) and \( c = \frac{h}{2} \). From statics, we can derive:

\[
M(x) = M \tag{1.5.5}
\]

\[
\sigma_x = \frac{6M}{bh^2} = 42 \text{KPa} \tag{1.5.6}
\]

**Beam Deflection**

The beam equation to be solved is:

\[
\frac{d^2y}{dx^2} = \frac{M(x)}{EI} \tag{1.5.7}
\]

After two integrations:

\[
y = \frac{Mx^2}{2EI} \tag{1.5.8}
\]

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

\[
y = \frac{ML^2}{2EI} = 7.26 \text{mm} \tag{1.5.9}
\]

**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

![Preferences in ANSYS GUI]

1. Click Main Menu
2. Click Preferences
3. Select Structural
4. Click OK
Keypoints

Since we will be using 2D Elements, our goal is to model the length and width of the beam. The thickness will be taken care of later as a real constant property of the shell element we will be using.

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 keypoint at the Origin.
4. Click **Apply**
5. Repeat Steps 3 and 4 for the following points in order: 0,0,10
6. Click **Ok**

Let’s check our work.

7. Click the **Top View** tool

8. 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**

9. Go to **Utility Menu -> Plot -> Replot**
Your graphics window should look as shown:

**SAVE_DB**

Since we have made considerable progress thus far, we will create a temporary save file for our model. This temporary save will allow us to return to this stage of the tutorial if an error is made.

1. Go to Utility Menu -> ANSYS Toolbar ->SAVE_DB This creates a save checkpoint
2. If you ever wish to return to this checkpoint in your model generation, go to Utility Menu -> RESUM_DB

**WARNING:** It is VERY HARD to delete or modify inputs and commands to your model once they have been entered. Thus it is recommended you use the SAVE_DB and RESUM_DB functions frequently to create checkpoints in your work. If salvaging your project is hopeless, going to Utility Menu -> File -> Clear & Start New -> Do not read file ->OK is recommended. This will start your model from scratch.

**Areas**

1. Go to Main Menu -> Preprocessor -> Modeling -> Create -> Areas -> Arbitrary -> Through KPs
2. Select Pick
3. Select List of Items
4. In the space below, type 1,2,3,4. This will select the four keypoints previously plotted
5. Click OK

**WARNING:** If you declared your keypoints in another order than specified in this tutorial, you may not be able to loop. In which case, plot the keypoints and pick them under List of Items in a “connect the dots” order that would create a rectangle.
6. Go to Plot -> Areas
7. Click the Top View tool if it is not already selected.
8. Go to Utility Menu -> Ansys Toolbar -> SAVE_DB

Your beam should look as below:

![Beam Image]

**Preprocessor**

**Element Type**

1. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete
2. Click Add
3. Click Shell -> 4node 181 the elements that we will be using are four node elements with six degrees of freedom.
4. Click OK
5. Click Close

SHELL181 is suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. (If the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
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. Click OK
4. Under Real Constants for SHELL181 -> Shell thickness at node I TK(I) enter 1 for the thickness
5. Click OK
6. Click Close

Now we must specify Young’s Modulus and Poisson’s Ratio

7. Go to Main Menu -> Material Props -> Material Models
8. Go to Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic
9. Enter $7E10$ for Young’s Modulus (EX) and .33 for Poisson’s Ratio (PRXY)
10. Click OK
11. out of Define Material Model Behavior window

Meshing

1. Go to Main Menu -> Preprocessor -> Meshing -> Mesh Tool
2. Go to Size Controls: -> Global -> Set
3. Under SIZE Element edge length put 5. This will create a mesh of square elements with width 5 meters. This gives us two elements through the width of the beam.
4. Click OK
5. Click Mesh
6. Click Pick All
Your mesh should look like this:

![Mesh Diagram]

**Loads**

**Displacements**

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 graphics area should look as below:

![Node Numbering]

This is one of the main advantages of *ANSYS Mechanical APDL* vs *ANSYS Workbench* in that we can visually extract the node numbering scheme. As shown, *ANSYS* numbers nodes at the left corner, the right corner, followed by filling in the remaining nodes from left to right.
5. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Displacement -> On Nodes**

6. Click **Pick -> Box** with your cursor, drag a box around nodes 1, 2, and 3 on the left face:

Picked Nodes will have a **Yellow Box** around them:

7. Click **OK**

8. Click **All DOF** to secure all degrees of freedom

9. Under **Value Displacement value** put 0. The left face is now a **fixed end**

10. Click **OK**

The fixed end will look as shown below:
Moment Load

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes
2. Select Pick -> Box -> and select nodes 4,27,26
3. Click OK
4. Under Direction of force/moment select MZ
5. Under VALUE Force/moment value enter 70000/3
6. Click OK

The resulting graphic should look as shown below:
Solution

1. Go to Main Menu -> Solution -> Solve -> Current LS (solve). LS stands for Load Step. This step may take some time depending on mesh size and the speed of your computer (generally a minute or less).

General Postprocessor

We will now extract the Displacement and Von-Mises Stress within our model.

Displacement

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution
2. Go to DOF Solution -> Y-Component of displacement
3. Click OK

4. Click the Front View and use the Dynamic Model Mode by right clicking and dragging down slightly.

*Numbers 5-11 make alterations in the contour plot for viewing pleasure*
5. Go to Utility Menu -> PlotCtrls -> Style -> Contours -> Uniform Contours…
6. Under NCOUNT enter 9
7. Under Contour Intervals click User Specified
8. Under VMIN enter -0.00721
   The beam deflects in the –Y direction so
   The max deflection is treated as a minimum
9. Under VMAX enter 0
10. Since we will be using 9 contour intervals, we will enter 0.00721/9 for VINC
11. Click OK
12. Let’s give the plot a title. Go to Utility Menu -> Command Prompt and enter:
    /title, Deflection of a Beam with a Distributed Load
    /replot
    The resulting plot should look like this:
Equivalent (Von-Mises) Stress

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution
2. Go to Nodal Solution -> Stress -> von Mises stress
3. Click OK
4. To get rid of the previous Plot Settings, go to PlotCtrls -> Reset PlotCtrls…
5. Using the same methodology as before, we can change the contour divisions from 0 to 42300 in increments of 42300/9
6. Change the Title to “Von-Mises Stress of a Beam with a Distributed Load”
7. Go to Utility Menu -> Plot -> Replot

The resulting plot should look as below:
Results

Max Deflection Error

\[ \delta_{\text{model}} = 7.21 \text{mm} \]

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

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

\[ \%E = \frac{\left| \delta_{\text{theoretical}} - \delta_{\text{model}} \right|}{\delta_{\text{theoretical}}} \times 100 = 0.964\% \] (1.5.13)

This is a very good error baseline for the mesh considering equation 1.5.8 is quadratic with respect to displacement. Since the 2D Elements we are using linearly interpolate between nodes, we can expect a degree of truncation error in our model. As we will show in our validation section, our model will converge to the expected solution as the mesh is refined.

Max Equivalent Stress Error

\[ \sigma_{\text{model}} = 42.4 \text{kPa} \]

\[ \sigma_{\text{theoretical}} = 42.0 \text{kPa} \]

Using the same definition of error as before, we derive that our model has 4.51% error in the max equivalent stress. The error results from extrapolation of stress at the integration points of the elements to the nodal values. We will see later that if higher order elements are used in this problem, the error in equivalent stress will disappear!
Validation

---

**Equivalent Stress vs. Length of Beam**

- **Y-axis:** Von-Mises Stress (Pa)
- **X-axis:** Percent Length (%)
- **Legend:**
  - 44 Elements
  - Theoretical
  - 275 Elements
  - 176 Elements

---

**Deflection vs. Length of Beam**

- **Y-axis:** Deflection (m)
- **X-axis:** Percent Length
- **Legend:**
  - 44 Elements
  - Theoretical
  - 275 Elements
  - 176 Elements