Module 1.7: Point Loading of a 3D 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>4</td>
</tr>
<tr>
<td>Preprocessor</td>
<td>6</td>
</tr>
<tr>
<td>Element Type</td>
<td>6</td>
</tr>
<tr>
<td>Material Properties</td>
<td>7</td>
</tr>
<tr>
<td>Meshing</td>
<td>8</td>
</tr>
<tr>
<td>Loads</td>
<td>9</td>
</tr>
<tr>
<td>Solution</td>
<td>15</td>
</tr>
<tr>
<td>General Postprocessor</td>
<td>15</td>
</tr>
<tr>
<td>Results</td>
<td>18</td>
</tr>
<tr>
<td>Validation</td>
<td>19</td>
</tr>
</tbody>
</table>
Problem Description:

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

\[ \sigma' = \left( \sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3 \tau_{xy}^2 \right)^{\frac{1}{2}} \]  \hspace{1cm} (1.7.1)

During our analysis, we will be analyzing the set of nodes through the top center of the cross section of the beam:

\[ \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots \]

Due to symmetric loading about the yz cross sections, analyzing the nodes through the top center will give us deflections that approximate beam theory. Additionally, since the nodes of choice are located at the top surface of the beam, the shear stress at this location is zero.

\[ (\tau_{xy} = 0, \ \sigma_y = 0). \]  \hspace{1cm} (1.7.2)

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

\[ \sigma' = \sigma_x \]  \hspace{1cm} (1.7.3)
Bending Stress is given by:

\[ \sigma_x = -\frac{Mc}{I} \]  \hspace{1cm} (1.7.4)

Where \( I = \frac{1}{12}bh^3 \) and \( c = \frac{h}{2} \).

From statics, we can derive:

\[ M = P(x - L) \]  \hspace{1cm} (1.7.5)

Plugging into equation 1.7.4, we get:

\[ \sigma_x = \frac{6P(x-L)}{bh^2} = 66kPa \]  \hspace{1cm} (1.7.6)

**Beam Deflection**

\[ \frac{d^2y}{dx^2} = \frac{M(x)}{EI} \]  \hspace{1cm} (1.7.7)

Plugging in equation 1.7.5, we get:

\[ EI \frac{d^2y}{dx^2} = P(x - L) \]  \hspace{1cm} (1.7.8)

Integrating once to get an angular displacement, we get:

\[ EI \frac{dy}{dx} = P\left(\frac{x^2}{2} - L\right) + C_1 \]  \hspace{1cm} (1.7.9)

At the fixed end (\( x=0 \)), \( \theta(0) = \frac{dy(0)}{dx} = 0 \), thus \( C_1 = 0 \)

\[ EI \frac{dy}{dx} = P\left(\frac{x^2}{2} - L\right) \]  \hspace{1cm} (1.7.10)

Integrating again to get deflection:

\[ Ely = P\left(\frac{x^3}{6} - L\frac{x^2}{2}\right) + C_2 \]  \hspace{1cm} (1.7.11)

At the fixed end \( y(0)=0 \) thus \( C_2 = 0 \), so deflection (\( \delta = y \)) is:

\[ \delta = \frac{P}{EI}\left(\frac{x^3}{6} - L\frac{x^2}{2}\right) = \frac{Px^2}{6EI} \left( x - 3L \right) \]  \hspace{1cm} (1.7.12)

The maximum displacement occurs at the point load (\( x=L \))

\[ \delta_{max} = \frac{PL^3}{3EI} = 7.61\text{mm} \]  \hspace{1cm} (1.7.13)

**WARNING:** In three dimensional cantilever beams, beam theory is just an approximated answer, it is NOT exact.
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 and Triad:

To add a title

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

The Triad in the top left will block images along the way.
To get rid of the triad, type /triad,off in Utility Menu -> Command Prompt

Beam:

1. Go to ANSYS Main Menu -> Preprocessor -> Modeling -> Create -> Volumes -> Block -> By Dimensions. This will open a new window, Create Block by Dimensions, where the Geometry will be created.
2. In Create Block by Dimensions -> X1,X2 X-coordinates -> input 0 -> tab 2 input 10
3. In Create Block by Dimensions -> Y1,Y2 Y-coordinates -> input 0 -> tab 2 input 1
4. In Create Block by Dimensions -> Z1,Z2 Z-coordinates -> input 0 -> tab 2 input 110
5. Then hit Ok to create the 3-Dimensional Cantilever Beam
This will generate a cantilever beam 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.

**Preprocessor**

**Element Type**

1. Go to Main Menu -> Preprocessor -> Element Type -> Add/Edit/Delete
2. Click Add
3. Click Solid -> 8node 185
4. Click OK
5. Click Close

*For more information Solid185 click Help

1. Go to ANSYS 12.1 Help -> Search Keyword Search -> type ‘Solid185’ and press Enter
2. Go to Search Options -> SHELL185
3. The element description should appear in the right portion of the screen.

Material Properties

1. Go to Main Menu -> Preprocessor -> Material Props -> Material Models
2. Click Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic
3. Input 7E10 for the Young’s Modulus (Aluminum) in EX
4. Input 0.33 for Poisson’s Ratio in PRXY
5. Click OK
6. 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 0.5. The SIZE Element edge length puts 1 element every distance you enter. This will do 2 elements every 1 meter.
4. Click OK
5. Click Mesh
6. Click Pick All

After meshing, pick the Front View. Your beam should look like the image below:
Loads

Because in real life a point load does not exist, displaying one correctly using 3D elements is tricky. Due to Saint-Venant’s Principle, we would like to model the point load as a load distributed across the right end face. The completely correct way to do so would be to model a parabolic shear distribution across the end face:

For a 1000N point load we will be putting a fraction of the force on each node. Since there are 2 elements every meter; on the end face there are 20 elements in the z direction and 2 elements in the y direction, for a total of 40 elements or 63 nodes. On each element there should be 25N of force (1000/40). Since there are four nodes per element, each node will get a quarter of the 25N, 6.25N. Since nodes overlap, certain nodes will get half of 25N and some will get the complete 25N. For example, here are four elements:

<table>
<thead>
<tr>
<th>Elements</th>
<th>Nodes</th>
<th>Force Distribution</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>(1/4) (1/2) (1/4)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(1/2) (1) (1/2)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(1/4) (1/2) (1/4)</td>
</tr>
</tbody>
</table>

As you can see with these four elements, this breaks down into three categories
1. Quarter of the force (1/4). These nodes have only one element around them.
2. Half of the force (1/2). These nodes only have two other elements either next to or above/below. (1/4) + (1/4) = (1/2)
3. Full force (1). These nodes have four elements surrounding them.
   (1/4) + (1/4) + (1/4) + (1/4) = (1)
Six Elements

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
5. Click the Left View to orient the cantilever beam horizontally down the z-axis
6. Shift the beam the left to view the far nodes more closely by pressing the Pan Model Left button then zoom in on the far right nodes using the Zoom in button or scrolling with the mouse
7. Use the Dynamic Model Mode and right clicking and dragging diagonally down slightly
The resulting graphic should be as shown:

This is one of the main advantages of ANSYS Mechanical APDL vs ANSYS Workbench in that we can visually extract the node numbering scheme.

Quarter Load

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes
2. Click Pick -> Single
3. Click List of Items and input 85,87,65,64
   This will select the four corner nodes of the cross sectional area
4. Click Ok
5. Under Lab Direction of Force/mom select FY
6. Under Value Force/moment value type -25/4
7. Press OK
   Force arrows will now appear on the selected nodes
Half Load
1. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes**
2. Click **Pick -> Single**
3. Click **Min, Max, Inc** and input **88,106,1**
   This will select the nodes from 88 to 106 using an increment of 1 on the top row of the cross section.
4. Click **Ok**
5. Under **Lab Direction of Force/mom** select **FY**
6. Under **Value Force/moment value** type **-25/2**
7. Press **Apply**
8. Repeat Steps 2-7 and input: **66,84,1**
   This will select the nodes from 66 to 84 using an increment of 1 on bottom row of the cross section.
9. Now click **List of Items** and input **107,86**
   This selects nodes 86 and 107, the beginning and end nodes of the middle row of the cross section.
10. Click **Ok**
11. Under **Lab Direction of Force/mom** select **FY**
12. Under **Value Force/moment value** type **-25/2**
13. Press **Ok**

Additional force arrows will now appear on the selected nodes.

**USEFUL TIP:** If you wish to assign new force values, **pick** the nodes of interest and replace that component of force **with 0** before assigning new values. This will delete the previous force assignment.
Full Load

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes
2. Click Pick -> Single
3. Click Min, Max, Inc and input 108,126,1
   This will select the nodes from 108 to 126 using an increment of 1 selecting the middle nodes of the middle row, excluding the initial and end nodes.
4. Click Ok
5. Under Lab Direction of Force/mom select FY
6. Under Value Force/moment value type -25
7. Press Ok

To view the cross section go to Utility Menu -> Plot Controls -> Numbering… -> Check NODE Node Numbers to Off -> Click OK then click Front View and zoom out if needed.

The resulting graphic should be as shown:

As you can see ANSYS takes into account larger and smaller forces, expressed by arrow size.
If end face does not resemble this graph:
1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Force/Moment -> On Nodes
2. Click Pick All
3. Under Lab Direction of Force/mom select FY, Under Value Force/moment value type 0
4. Press Ok
5. Redo the steps above. Check correct number orientation is used.
Displacement (Fixed End)

1. Click the Left View to see along the z-axis
2. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Displacement -> On Nodes
3. Click Pick -> Box
4. With your cursor, drag a box around the first set of nodes on the far left side of the beam:

5. Click OK
6. Click All DOF to secure all degrees of freedom
7. Under Value Displacement value put 0. The left face is now a fixed end.
8. Click OK

WARNING: Selecting the wrong/wrong amount of nodes will result in a wrong answer; make sure the only nodes selected are only the end set as shown.
The final result of your beam with a fixed end and 1000N point load applied should resemble the image 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).
2. A Note will pop up saying the ‘Solution is done!’ -> Press Close -> if necessary X out of the /STATUS Command window.

General Postprocessor

General Postprocessor processes the data from the solution and displays it with an array of styles. This is where we get the solutions to the deflection of the beam and Von-Mises Stress.

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
Let's change some plotting options and enhance the aesthetics.

4. Go to **Utility Menu -> PlotCtrls -> Style -> Contours -> Uniform Contours**
5. Under **NCOUNT** enter 9
6. Under **Contour Intervals** click **User Specified**
7. Under **VMIN** enter -0.0075
   The beam deflects in the –Y direction so the max deflection is treated as a minimum
8. Under **VMAX** enter 0
9. Since we will be using 9 contour intervals, we will enter 0.0075/9 for **VINC**
10. Click **OK**
11. **Utility Menu -> ANSYS Toolbar ->**
    type /Title, 3D Cantilever Beam Deflection -> enter

Resulting Answer:

- Maximum Deflection = 0.007443 m
Equivalent (Von-Mises) Stress

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solution-> Stress -> von Mises stress
2. Click OK
3. To get rid of the previous Plot Settings, go to PlotCtrls -> Reset PlotCtrls…
4. Go to Utility Menu -> Plot -> Replot

Aesthetics
5. Click the Isometric View to see a better view of your cantilever beam.
6. Utility Menu -> ANSYS Toolbar -> type /Title, 3D Cantilever Beam Von Mises Stress -> enter
7. Go to Utility Menu -> PlotCtrls -> Style -> Contours -> Uniform Contours…
8. Under NCOUNT enter 9
9. Under Contour Intervals click User Specified
10. Under VMIN enter 0
11. Under VMAX enter 61000
12. Under VINC enter 61000/9
13. Click Ok

Resulting Answer:

Maximum Stress= 60446 Pa
**Results**

**Max Deflection Error**

\[ \delta_{\text{max}} = 7.443 \text{mm} \]  
\[ \delta_{\text{max}} = 7.61 \text{mm} \]

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

\[
\%E = \left| \frac{\delta_{\text{theoretical}} - \delta_{\text{model}}}{\delta_{\text{theoretical}}} \right| \times 100 = 2.19\% \tag{1.7.14}
\]

This is a good error baseline considering mesh size used. There is an assumed deviation in the ANSYS results from the theoretical answers due to Beam Theory. Beam Theory is a derived equation solely for one dimensional cases. When an extra degree is added, this assumes a linear displacement in that extra degree of movement. As mesh is increased, nonlinear lines approach linearity. In the validation section, it is shown that with increased mesh size, these values converge to a closer representation of the theoretical value.

**Max Equivalent Stress Error**

\[ \sigma_{\text{max}} = 60.45 \text{kPa} \]  
\[ \sigma_{\text{max}} = 66 \text{kPa} \]

Using the same definition of error as before, we derive that our model has 8.41\% error in the max equivalent stress. The reason for the elevated stress level is singularity resulting from Poisson’s effect at the fixed support. In the validation section, it is shown that with increased mesh size, the analytical answers for Max Equivalent stress are closely represented in nodes close to but not at the region where singularity occurs. The effect of singularity is also reduced with the implementation of higher order elements.