Module 1.7W: 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>Workbench Analysis System</td>
<td>4</td>
</tr>
<tr>
<td>Engineering Data</td>
<td>5</td>
</tr>
<tr>
<td>Geometry</td>
<td>6</td>
</tr>
<tr>
<td>Model</td>
<td>11</td>
</tr>
<tr>
<td>Setup</td>
<td>13</td>
</tr>
<tr>
<td>Solution</td>
<td>14</td>
</tr>
<tr>
<td>Results</td>
<td>16</td>
</tr>
</tbody>
</table>
Problem Description

![Diagram of a cantilever beam with load and dimensions labeled]

Nomenclature:
- \( L = 110 \text{m} \) Length of beam
- \( b = 10 \text{m} \) Cross Section Base
- \( h = 1 \text{ m} \) Cross Section Height
- \( w = 20 \text{N/m} \) Distributed Load
- \( E = 70 \text{GPa} \) Young’s Modulus of Aluminum at Room Temperature
- \( \nu = 0.33 \) Poisson’s Ratio of Aluminum

This is a simple, single load step, structural analysis of a cantilever beam. The left side of the cantilever beam is fixed while there is a distributed load of 20N/m. The objective of this problem is to demonstrate a simple ANSYS Workbench problem with a textbook solution: finding Von Mises’ stresses and total deflection throughout the beam. 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:

\[
\sigma' = \left( \sigma_x^2 - \sigma_x \sigma_y + \sigma_y^2 + 3\tau_{xy}^2 \right)^{\frac{1}{2}} \tag{1.7W.1}
\]

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

\[
(\tau_{xy} = 0, \ \sigma_y = 0). \tag{1.7W.2}
\]

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

\[
\sigma' = \sigma_x \tag{1.7W.3}
\]

Bending Stress is given by:

\[
\sigma_x = \frac{P(x-L)c}{I} \tag{1.7W.4}
\]

Where \( I = \frac{1}{12}bh^3 \) and \( c = \frac{h}{2} \). From statics, we can derive:
\[ \sigma_x = \frac{6P(x-L)}{bh^2} = 66\text{kPa} \]  \hspace{1cm} (1.7W.6)

**Beam Deflection**

As in module 1.1, the beam equation to be solved is:

\[ \frac{d^2y}{dx^2} = \frac{M(x)}{EI} \]  \hspace{1cm} (1.7W.7)

Using Shigley’s Mechanical Engineering Design, the beam deflection is:

\[ \delta(x) = \frac{Px^2(x-3L)}{6EI} \]  \hspace{1cm} (1.7W.8)

With Maximum Deflection at:

\[ \delta = \frac{PL^3}{3EI} = 7.61\text{mm} \]  \hspace{1cm} (1.7W.9)

**Workbench Analysis System**

**Opening Workbench**

1. On your Windows 7 Desktop click the Start button.
2. Under Search Programs and Files type “ANSYS”
3. Click or open ANSYS Workbench to start workbench. This step may take time.
Static Structural Analysis

1. As you open ANSYS you can see the entire array of problems on the left hand side this software can help you solve. The problem at hand is a Static Structural problem. Double click Static Structural (ANSYS) to open the task manager for your problem set in the Project Schematic area.

2. ANSYS allows you to build on each problem, so it is smart to name each project. At the bottom of the task manager you will see Static Structural (ANSYS), double click this to change the name. For this problem choose “3D Cantilever beam.”
Engineering Data

To begin setup for your cantilever beam, double click or right click on **Engineering Data** and click **edit**. This will bring up another screen.

This new window will allow you to alter the material properties of your cantilever beam. Under **Outline of Schematic A2: Engineering Data**, it shows **click here to add a new material**, this menu allows you to input the material of your cantilever beam, double click and type **Aluminum**.

**WARNING**: Do not delete or change the Structural Steel, just another material.

Now expand **Linear Elastic** by double clicking on or on the plus symbol shown.

Double click on Isotropic Elasticity to give the material the same properties across the beam. This action brought up a new table on the right; this allows us to add necessary properties. As show on the top right of the screen in **Table of Properties Row 2: Isotropic Elasticity**:
1. Click in Temperature and type 25
2. Click in Young’s Modulus and type 70E9 or 7E10
3. Click in Poisson’s Ratio and type 0.33

**WARNING:** Make sure to DELETE the Temperature entry after property input before continuing! Failure to do so will lead to errors later.

After filling in the properties, this concludes the Engineering Data, to return to the project schematic area, click on ![Return to Project](image)

**Geometry**

**Base Geometry**

1. Go to **Workbench -> Project Schematic -> Geometry** and double click. This will open a new window for **ANSYS Design Modeler** where the Geometry will be created.

Note: Select meters and hit ok

2. In the new window, click the ![Display Plane](image) icon to toggle the coordinate system.
3. Go to **Design Modeler -> Tree Outline ->** right click on YZPlane. Click **Look At** to view the YZ plane.
4. Go to **Design Modeler -> Tree Outline -> Sketching**
5. Click on **Rectangle** and Click off **Auto-Fillet**:
6. Bring your cursor into the workspace at point 0,0, over the origin until ‘P’ appears directly above the origin.
7. Click on the origin to place the lower left corner of our rectangle on the origin.
8. Click on a point in the first quadrant to define the top right corner of our rectangle. The point is arbitrary as we will be fixing dimensions momentarily.

![Rectangle Sketch](image)

9. Go to **Sketching Toolboxes -> Dimensions**
10. Click **Horizontal** to specify a horizontal dimension.
11. Click the left and right faces of the rectangle in the sketch to specify that we will be dimensioning this horizontal length. A green line with a symbol should appear.
12. Drag the green line above the sketch and click to set its location.
13. Go to **Detail View -> Dimension 1**. In the first subcategory, replace the current dimension with 10. The units should populate automatically.
14. Go to Sketching Toolboxes -> Dimensions -> Vertical to specify the vertical dimension.
15. Click the bottom and top faces of the sketch to specify the vertical dimension. A green line should appear.
16. Drag the green line to the right of the sketch and click.
17. Go to Detail View -> Dimension 2. Replace the value with 10. The units should populate automatically (meters).

Now that we have modeled the base geometry, we will extrude it to create a 3D volume.
Extrude Sketch

1. Go to **Main Toolbar** -> and select **Extrude**
2. Go to **Modeling** -> **FD1, Depth (>0)** -> enter in **110**
3. Go to **Design Modeler** -> **Generate**.
4. To verify our geometry, look at the isometric view. Click the **blue dot** in the **triad** in the lower right corner of the screen to look at the isometric view.

Your 3D Surface should look like this:

Now that we have the geometry, we will mesh the beam using 3D Elements.
Model

Open ANSYS Mechanical

1. 
2. Go to Workbench -> Project Schematic -> Model This will open ANSYS Mechanical
Material Assignment

1. Go to Mechanical -> Outline -> Project -> Model -> Geometry -> Surface Body
2. Under Mechanical -> Details of “Surface Body” -> Material -> Assignment, change Structural Steel to Aluminum.
1. Go to **Mechanical -> Outline -> Project -> Model -> Mesh**

2. Go to **Mechanical -> Details of ‘Mesh’ -> Sizing -> Element Size** and change the value from **Default** to .5 m. This will give us 2 elements through the thickness of the beam.

3. Click **Mechanical -> Update**. This may take some time. Your mesh should look as shown below:
Setup

You can perform the rest of your analysis for this problem in the ANSYS Mechanical window. The other options in the Workbench window will link you back to the same screen (i.e. Setup, Solution, Results)

Fixed Support

1. Go to Mechanical -> Outline -> right click Static Structural (A5)
2. Go to Insert -> Fixed Support

We are going to fix the elements at the left end of the beam. In order to do this, we will use the **Edge** tool to select the left edge. However, from the current orientation of the beam, it is difficult to select this surface.

3. Using the **Rotate** tool click on the graphic area and move the mouse to the right. This will cause the left end of the beam to be oriented in a manner that can be clicked
4. Using the
[Pan tool, click the graphic area and drag the left face to the center of the graphic window. Use the mouse scroll to zoom in on the left face.]

5. Click the
[Edge tool.]

6. Go to Mechanical -> Outline -> Static Structural (A5) -> Fixed Support

7. Run the cursor across the left end face. When it becomes red, click it to select it.

8. Go to Mechanical -> Details of “Fixed Support” -> Geometry and select Apply

Setup

While in the Project Schematic double click Setup
This will open a new window similar to Model Space

Loads

1. Click the x-axis icon to get a side view of the cantilever beam
2. Click Fixed end On the tool bar, make sure vertex option is selected.
3. Go to Mechanical -> Outline -> Static Structural (A5) -> Fixed Support
4. Run the cursor across the left end face. When it becomes red, click it to select it.
5. Click the left side of the geometry; this will add a green box to select the point.
6. Right click
7. Click insert, and
8. This will add a fixed end to your cantilever beam in the work space.
9. Point Load On the tool bar, change selection option to Edge: edge instead of vertex.
10. Click on the geometry, this will highlight
11. Right click Static Structural (A5)
12. click insert , and A table will appear “Details of Line Pressure”
13. Under “Definition” you will see “Defined by” → Change this to “Components”
14. As shown, Y Component force is zero. → Change this to value to -20
15. This will show your cantilever beam with a load applied as shown. Leave the Setup screen open this time.
Solution

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Right Click Solution (A6) -> Insert -> Beam Tool

Deformation

Go to Mechanical -> Outline -> Project -> Model(A4) -> Static Structural (A5) -> Solution (A6) -> Beam Tool -> Insert -> Beam Tool -> Deformation -> Total
Now that our solvers have been defined, go to **Mechanical** -> **Solve**. The calculations in Workbench may take up to a minute to solve.

Go to **Mechanical** -> **Outline** -> **Project** -> **Model (A4)** -> **Solution (A6)** -> **Maximum Bending Stress**

Go to **Details of “Maximum Bending Stress”**-> **Integration Point Results** -> **Display Option** -> **Change to Unaveraged**

Your Stress plot should look as shown below:
Go to Mechanical -> Outline -> Project -> Model(A4) -> Solution(A6) -> Total Deformation

Your Von-Mises plot should look as shown below:
Results

Max Deformation Error

According to equation 1.4W.9, the theoretical max deflection is 7.16 mm. The percent error (%E) in our model can be defined as:

\[
%E = \frac{\delta_{\text{theoretical}} - \delta_{\text{model}}}{\delta_{\text{theoretical}}} \times 100 = 1.28\% 
\]  

(1.7W.10)

Max Equivalent Stress Error

According to equation 1.7W.6, the theoretical max equivalent stress is 66000 Pa. Using the same definition of error as before, we derive that our model has 6.3% 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.

Validation
Deflection vs Length of Beam

Von-Mises Stress vs Length of Beam