Module 6: Rotation Loading 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>3</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>7</td>
</tr>
<tr>
<td>Meshing</td>
<td>9</td>
</tr>
<tr>
<td>Loads</td>
<td>10</td>
</tr>
<tr>
<td>Solution</td>
<td>12</td>
</tr>
<tr>
<td>General Postprocessor</td>
<td>12</td>
</tr>
<tr>
<td>Results</td>
<td>15</td>
</tr>
<tr>
<td>Validation</td>
<td>16</td>
</tr>
</tbody>
</table>
Problem Description

In this module, we will solve for the deflections and stresses resulting from rotation loading of a cantilever beam. The results gathered from this tutorial will help us in module 11 when we go to model a fan blade with similar features. We will model this beam using 1D BEAM Elements. By the end of this tutorial, we will gain confidence modeling stationary objects in the rotational frame of reference.

Theory

Axial Stress

\[ \sigma A = \left( \sigma + \frac{\partial \sigma}{\partial x} dx \right) A \]

From the analysis of a stress element in the beam, equilibrium dictates:

\[ \sum F_x = ma \]  \hspace{1cm} (6.1a)

\[ \left( \sigma + \frac{\partial \sigma}{\partial x} dx \right) A - \sigma A = (\rho A dx) \ast (x \omega^2) \]  \hspace{1cm} (6.1b)
Integrating with respect to x and using the boundary condition $\sigma(L + a) = 0$ we get:

$$\sigma(x) = \frac{\rho \omega^2}{2} ((L + a)^2 - x^2) \frac{1}{386.4} \quad (6.2)$$

With max stress $\sigma(a) = 15.87 \text{ ksi}$. Multiplying by Area, we get $F_x(a) = 3340.1 \text{ lbf}$

Since the max stress is 12.4% the yield stress of Titanium, we are in the linear elastic range of the stress-strain relationship.

Axial Deflection

Given the relationship:

$$\varepsilon = \frac{\partial u(x)}{\partial x} \quad (6.3)$$

and

$$\sigma(x) = E\varepsilon \quad (6.4a)$$

We get

$$\sigma(x) = E \frac{\partial u}{\partial x} \quad (6.4b)$$

Integrating eqn 6.2 with respect to x with the boundary $U(a) = 0$ we get

$$U(x) = \frac{\rho \omega^2}{2E} \left((L + a)^2 \left(x - a\right) + \left(\frac{a^3 - x^3}{3}\right)\right) \frac{1}{386.4} \quad (6.5)$$

With maximum deflection $U(L + a) = 0.003443 \text{ in}$
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**
Keypoints

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

Go to **Main Menu -> Preprocessor -> Modeling -> Create -> Keypoints -> On Working Plane**

1. Click **Global Cartesian**
2. In the box underneath, write 4.686,0 creating a keypoint at the start of the beam.
3. Click **Apply**
4. Repeat Steps 3 and 4 for the point 11,0
5. Click **Ok**

Let’s check our work.

6. Click the [Dynamic Model Mode](#) icon. On the graphics window, right click and drag the cursor down. You should now be able to see the two key points you have just created.

7. To get rid of the triad, type `/triad,off` in **Utility Menu -> Command Prompt**

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

Your graphics window should look as shown:
1. Go to **Main Menu** -> **Preprocessor** -> **Modeling** -> **Create** -> **Lines** -> **Lines** -> **Straight Line**
2. Select **Pick**
3. Enter 1,2 for keypoints
4. Click **OK**

Go to **Utility Menu** -> **Ansys Toolbar** -> **SAVE_DB**

**Saving Geometry**

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 Rotating Beam’
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 -> 2D Elastic 3
4. Click OK
5. Go to Utility Menu -> ANSYS Toolbar -> SAVE_DB

BEAM 3 is a 1D element with two translational and one rotational degree of freedom. For more information, click Help. For a link to ANSYS Help.

**Real Constants and Material Properties**

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

3. Enter 1.65e7 for Young’s Modulus (EX) and .342 for Poisson’s Ratio (PRXY)
4. Click OK
5. out of **Define Material Model**

   **Behavior**

6. Go to **Utility Menu -> SAVE_DB**

Now we will add the other properties to our beam.

1. Go to **Main Menu -> Preprocessor -> Real Constants -> Add/Edit/Delete**
2. Click **Add**
3. Click **OK**

4. Under **AREA** put $0.0623 \times 3.379$
5. Under **IZZ** put $3.329 \times 0.0623 \times 0.0623 \times 0.0623 / 12$
6. Under **HEIGHT** put $0.0623$
7. Under **ADDMAS** put $0.16 \times 0.0623 \times 3.379 / 386.4$ this value is Essentially the mass per unit length of the beam. We added the gravitational constant to this value so the axial forces returned later will be in lbf.
8. Click **Close**

**WARNING:** If the **ADDMAS** field is not filled, the solution will not appear.
Meshing

1. Go to Main Menu -> Preprocessor -> Meshing -> Mesh Tool
2. Go to Size Controls: -> Global -> Set
4. Click OK
5. Click Mesh
6. Click Pick All
7. Click Close
8. Go to Utility Menu -> SAVE_DB
### Loads

#### Displacement

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

1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Displacement -> On Nodes
2. Click Pick -> Single with your cursor, click on first node
3. Click OK
4. Click All DOF to secure all degrees of freedom
5. Under Value Displacement value put 0.
6. Click OK
7. Go to Utility Menu -> SAVE_DB
The fixed end will look as shown below:

**Inertia**

1. Go to **Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural -> Inertia -> Angular Velocity -> Global**
2. Under **OMEGZ** enter **879.65**
3. Since this problem involves small deflections, we do not need to include the non-linear spin softening option. Select **No Modification**
4. Click **OK**
5. Go to **Utility Menu -> Plot -> Elements**

The resulting picture is shown below:

The blue marks the center of the axis of rotation. The beam is offset by the distance, a, which represents the hub of a rotating fan in future tutorials. We have modeled the beam as stationary in a *rotating frame of reference*, represented by the inertia load. In more complicated loading, one can select specific elements to follow specific axes of rotation. For the purpose of our tutorial, the axis of rotation is a global constraint.
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

Deflection

1. Go to Main Menu -> General Postprocessor -> Plot Results -> Deformed Shape
2. Select Def + undeformed
3. Click OK

As expected, the beam experiences slight axial deformation as shown below:

4. Go to Main Menu -> General Postprocessor -> Plot Results -> Contour Plot -> Nodal Solu -> DOF Solution -> X-Component of displacement -> OK

The following plot should generate:

As shown on the contour band, the largest deflection in the beam was calculated as 0.003443in, perfectly aligning with the theoretical value! For info on plot aesthetics, see module 1.4 page 16.
Axial Stress

Unfortunately, since we are modeling with 1D BEAM elements, we cannot generate plots for stress. We can however, look up all available force items in a list file organized by node. If we divide axial force by the cross sectional area of the beam, it should produce the expected axial stress.

1. Go to Utility Menu -> List -> Results -> Element Solution… ->
   All Available Force Items - > OK

The following list file should populate:

![List File Screenshot]

The FX column has all of the axial force items of interest. Looking at node 1, if we divide the value in the table by the cross sectional area of the beam, we get an axial stress of 15.87 ksi, the exact theoretical value!
Further Analysis

In addition to this baseline data, we can export both the deflection and axial stress data to Excel.

1. Go to PRNSOL Command -> File -> Save As …
2. Save the file as ROT_Axial Force.lis to the 
   path of your choice

3. Go to PRNSOL Command -> File -> Close
4. Open ROT_Axial Force.lis in Excel
5. Click Fixed Width
6. Click Next >

7. Click a location on the ruler between the NODE and 
   FX columns. This will cause Excel to separate these 
   columns into separate columns in the spreadsheet
8. Click Next >
9. Click Finish
Results

Max Deflection Error

\[ \delta_{\text{max}} = 0.003443 \]

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

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

As one can see, this analysis produces zero error with a very coarse mesh (2 elements!). This is because beam element displacement functions are fourth order accurate in ANSYS and the deflection is proportional to \( x^3 \), a fourth order function.

Max Stress Error

\[ \sigma_{\text{max}} = 15.87 \text{ ksi} \]

Using the same definition of error as before, we derive that our model has 0% error in the max equivalent stress. 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 Beam3 has two integration points and two-point Gaussian Quadrature is fourth degree accurate, the answer will have no error because the theoretical function (eqn 6.2) is a third order equation. Thus the one dimensional method has zero percent error in deflection and stress.
Validation

Axial Stress vs Percent Length of Beam

Axial Deflection vs Percent Length of Beam