Module 7: Thermal Buckling of a 2D Beam Fixed at Both Ends

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>Real Constants and Material Properties</td>
<td>7</td>
</tr>
<tr>
<td>Meshing</td>
<td>9</td>
</tr>
<tr>
<td>Solution</td>
<td>10</td>
</tr>
<tr>
<td>Static Solution</td>
<td>10</td>
</tr>
<tr>
<td>Eigenvalue</td>
<td>12</td>
</tr>
<tr>
<td>Mode Shape</td>
<td>13</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:

Nomenclature:

- \( L = 200 \text{mm} \) Length of beam
- \( b = 10 \text{mm} \) Cross Section Base
- \( h = 1 \text{ mm} \) Cross Section Height
- \( E = 200,000 \frac{N}{\text{mm}^2} \) Young’s Modulus of Steel at Room Temperature
- \( \nu = 0.33 \) Poisson’s Ratio of Steel
- \( T = 26^\circ \text{C} \) Uniform temperature of beam
- \( T_{ref} = 25^\circ \text{C} \) Reference Temperature
- \( \alpha = 11.7 \times 10^{-6} \text{ C}^{-1} \) Thermal Expansion (Secant) Coefficient of Steel

In this module we will study thermal buckling resulting from an elevated temperature on a rectangular beam fixed at both ends. We modeled the beam using two dimensional shell elements and ANSI 1030, a low carbon steel. Buckling is inherently non-linear, but we will linearize the problem through the Eigenvalue method. This solution is an overestimate of the theoretical value since it does not consider imperfections and nonlinearities in the structure such as warping and manufacturing defects. This module will be compared against analytical results in an elasticity textbook.

Theory

Thermal Load

\[
\sigma = E \epsilon \\
\epsilon = \frac{\Delta L}{L} \\
\sigma = \frac{p}{A} \\
\frac{p}{A} = \frac{E \Delta L}{L} \quad \text{(7.1)}
\]

A change in temperature causes a change in length (\( \Delta L \)) from the original length

\[
\Delta L = \alpha L \Delta T \quad \text{(7.2)}
\]
Substituting equation 7.2 into equation 7.3 we get

\[ \frac{P}{A} = E\alpha \Delta T \]  

(7.4)

The force due to a change in temperature is then:

\[ P = E A \alpha \Delta T \]  

(7.5)

**Buckling Load**

Using the theory derived in the simply supported buckling module 3 & 4:

\[ P = \frac{n^2 \pi^2 EI}{L_c^2} \]  

(7.6)

Where the lowest Buckling Load is at \( n^2 = 1 \)

\[ P = \frac{\pi^2 EI}{L_c^2} \]  

(7.7)

This is an over estimate so there are certain correction factors (C) to account for this. (C) is dependent on the beam constraints.

\[ P = \frac{C \pi^2 EI}{L_c^2} \]  

(7.8)

Where C=1 for a conservative answer. \( L_c = \frac{L}{2} \)

So the Critical Buckling Load is

\[ P = \frac{4\pi^2 EI}{L^2} \]  

(7.9)

Since

\[ P_{thermal} = P_{buckling} \]

The combination of 7.9 & 7.5 is:

\[ E A \alpha \Delta T = \frac{4\pi^2 EI}{L^2} \]  

(7.10)

Then the critical temperature change to cause buckling is

\[ \Delta T = \frac{4\pi^2 EI}{E A \alpha L^2} \]  

(7.11)

Because

\[ I = \frac{1}{12} bh^3 \]

And

\[ A = bh \]

Critical Temperature simplifies to

\[ \Delta T = \frac{\pi^2 h^2}{3\alpha L^2} = 7.028^\circ C \]  

(7.12)
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
Key points

Since we will be using 2D Elements, our goal is to model the length and width 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 keypoint at the Origin.
4. Click Apply
5. Repeat Steps 3 and 4 for the following points in order: 0,0,10 200,0,10 200,0,0
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

Areas

1. Go to Main Menu -> Preprocessor -> Modeling -> Create -> Areas -> Arbitrary -> Through KPs
2. Select Pick
3. Select Min, Max, Inc
4. In the space below, type 1,4,1. If you are familiar with programming, the ‘Min,Max, Inc’ acts as a FOR loop, connecting nodes 1 through 4 incrementing (Inc) by 1.
5. Click OK
6. Go to Plot -> Areas
7. Click the Top View tool
Your beam should look as below:

Saving Geometry

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 ‘Buckling simply supported’
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 Shell -> 4node181
4. Click OK

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, Poisson’s Ratio and the Secant Coefficient.

1. Go to **Main Menu -> Preprocessor -> Material Props -> Material Models**
2. Go to **Material Model Number 1 -> Structural -> Linear -> Elastic -> Isotropic**
3. Input 2E5 for the Young’s Modulus (Steel in mm) in EX.
4. Input 0.3 for Poisson’s Ratio in PRXY
5. Click **OK**

6. Go to **Structural -> Thermal Expansion -> Secant Coefficient -> Isotropic**
7. Under **Reference temperature** enter 25
8. Under **ALPX** enter **11.7E-6**
9. Click **OK**
10. Go to **Define Material Model Behavior -> Material -> Exit**
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 5 elements across the width of the beam.
4. Click OK
5. Click Mesh
6. Click Pick All

7. Go to Utility Menu -> Plot -> Nodes
8. Go to Utility Menu -> Plot Controls -> Numbering…
9. Check NODE Node Numbers to ON
10. Click OK
The resulting graphic should be as shown using the Top View:

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.

**Solution**

There are two types of solution menus that ANSYS APDL provides; the Abridged solution menu and the Unabridged solution menu. Before specifying the loads on the beam, it is crucial to be in the correct menu.

Go to **Main Menu -> Solution -> Unabridged menu**

This is shown as the last tab in the Solution menu. If this reads “Abridged menu” you are already in the Unabridged solution menu.

**Static Solution**

**Analysis Type**

1. Go to **Main Menu -> Solution -> Analysis Type -> New Analysis**
2. Choose **Static**
3. Click **OK**

4. Go to **Main Menu -> Solution -> Analysis Type -> Analysis Options**
5. Under [SSTIF][PSTRES] Stress stiffness or prestress select Prestress ON
6. Click OK

Prestress is the only change necessary in this window and it is a crucial step in obtaining a final result for eigenvalue buckling.

Displacement
1. Go to Main Menu -> Solution -> Define Loads -> Apply -> Structural -> Displacement -> On Nodes
2. Select Box
3. Box nodes: 44, 45, and 4 (far right nodes)
4. Box nodes: 1, 3, and 2 (far left nodes)
5. Click OK
6. Under Lab2 DOFs to be constrained select All DOF
7. Under VALUE Displacement value enter 0

8. Click OK

This creates the fixed end on the left and on the right, using the front view, your beam should be:
Uniform Temperature
1. Go to Main Menu -> Preprocessor -> Loads -> Define Loads -> Apply -> Structural
   -> Temperature -> On Areas
2. Since only one area was made click Pick All
3. In the new window, under VAL1 Temperature enter 26
4. Click OK

Solve
1. Go to Main Menu -> Solution -> Solve -> Current LS
2. Go to Main Menu -> Finish

TIP: The difference between Reference Temperature and Applied Temperature is only 1 because eigenvalues are calculated by a factor of the load applied, so having a difference of 1 will make the eigenvalue answer equal to the Temperature difference to cause buckling.

Eigenvalue
1. Go to Main Menu -> Solution -> Analysis Type -> New Analysis
2. Choose Eigen Buckling
3. Click OK
4. Go to Main Menu -> Solution -> Analysis Type -> Analysis Options
5. Under NMODE No. of modes to extract input 5
6. Click OK

7. Go to Main Menu -> Solution -> Solve -> Current LS
8. Go to Main Menu -> Finish

Mode Shape
1. Go to Main Menu -> Solution -> Analysis Type -> ExpansionPass
2. Click [EXPASS] Expansion pass to ensure this is turned on
3. Click OK

4. Go to Main Menu -> Solution -> Load Step Opts -> ExpansionPass -> Single Expand -> Expand Modes
5. Under NMODE No. of modes to expand input 5
6. Click OK
7. Go to **Main Menu -> Solution -> Solve -> Current LS**
8. Go to **Main Menu -> Finish**

**General Postprocessor**

**Buckling Temperature**

Now that ANSYS has solved these three analysis lets extract the lowest eigenvalue. This represents the lowest force to cause buckling.

Go to **Main Menu -> General Postproc -> List Results -> Detailed Summary**

![Results Table]

Results for the change in temperature to cause buckling:

**ΔT = 7.0816° C**

**Mode Shape**

To view the deformed shape of the buckled beam vs. original beam:

1. Go to **Main Menu -> General Postproc -> Read Results -> First Set**
2. Go to **Main Menu -> General Postproc -> Plot Results -> Deformed Shape**
3. Under **KUND Items to be plotted** select **Def + undeformed**
4. Click **OK**

![Mode Shape Visualization]

UCONN ANSYS –Module 7: Thermal Buckling of a 2D Beam Fixed at Both Ends
The graphics area should look as below using the Oblique view:

Results

\( \Delta T = 7.0816^\circ \text{C} \)

\( \Delta T = 7.028^\circ \text{C} \)

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

\[
%E = \left( \frac{\delta_{\text{theoretical}} - \delta_{\text{model}}}{\delta_{\text{theoretical}}} \right) \times 100 = 0.76266\%
\]
**Validation**

<table>
<thead>
<tr>
<th></th>
<th>Theoretical</th>
<th>320 Elements</th>
<th>180 Elements</th>
<th>80 Elements</th>
<th>20 Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Critical Buckling Temperature</strong></td>
<td>7.028º C</td>
<td>7.0524</td>
<td>7.0578</td>
<td>7.0816</td>
<td>7.1577</td>
</tr>
<tr>
<td><strong>Percent Error</strong></td>
<td>0%</td>
<td>0.34718%</td>
<td>0.42402%</td>
<td>0.76266%</td>
<td>1.8455%</td>
</tr>
</tbody>
</table>

This table provides the critical buckling loads and corresponding error from the Theory (Euler), and four different ANSYS results; one with 320, 180, 80 and 20 elements. This is to prove mesh independence, showing with increasing mesh size, the answer approaches the theoretical value. The results here show that using a coarse mesh of 20 elements creates an acceptable error baseline but you can still refine the mesh for a fast and more accurate solution. The eigenvalue buckling method over-estimates the “real life” buckling load. This is due to the assumption of a perfect structure, disregarding flaws and nonlinearities in the material. There is no such thing as a perfect beam so the structure will never actually reach the eigenvalue load that is calculated.