Column Buckling Analysis
by
Anthony Beeman
An Engineering Project Submitted to the Graduate Faculty of Rensselaer Polytechnic Institute in Partial Fulfillment of the Requirements for the degree of Master of Engineering in Mechanical Engineering

Rensselaer Polytechnic Institute
Groton, CT
April 26, 2014
# CONTENTS

Column Buckling Analysis ................................................................................................. i

ABSTRACT .......................................................................................................................... iii

1. Introduction and Background ....................................................................................... 1
   1.1 Overview of Column Buckling Problem ................................................................. 1

2. Theory and Methodology .............................................................................................. 2
   2.1 Other End Condition ............................................................................................... 3
   2.2 Buckling Theoretical Solution for a Fixed-Free Column .................................... 4

3. Results & Discussion ................................................................................................... 5
   3.1 Comsol Multiphysics ............................................................................................. 5
      3.1.1 Geometry & Materials .................................................................................. 5
      3.1.2 Loads and Boundary Condition .................................................................... 5
      3.1.3 Mesh Refinement Study ............................................................................... 6
      3.1.4 Results .......................................................................................................... 7
   3.2 Abaqus ..................................................................................................................... 8
      3.2.1 Geometry & Materials .................................................................................. 8
      3.2.2 Loads and Boundary Condition .................................................................... 8
      3.2.3 Mesh Refinement Study ............................................................................... 9
      3.2.4 Results .......................................................................................................... 10
   3.3 ANSYS .................................................................................................................... 11
      3.3.1 Geometry & Materials .................................................................................. 11
      3.3.2 Loads and Boundary Condition .................................................................... 12
      3.3.3 Mesh Refinement Study ............................................................................... 12
      3.3.4 Results .......................................................................................................... 14

4. Conclusion ..................................................................................................................... 15

5. References .................................................................................................................... 16
ABSTRACT

This paper determined the critical load that causes elastic instability for a fixed-free circular column that is 10 meters in length and has a radius of 0.5 meters. The critical load was determined with Euler's buckling load theory and the theoretical results were compared with Comsol Multiphysics, Abaqus, and ANSYS finite element analysis software analytical results. The analytical critical loads determined by Comsol Multiphysics, Abaqus, and ANSYS compared very with Euler's analytical critical buckling load. After mesh refinement studies the percent error determined for each critical load value was 0.49, 1.52, and 0.16 for Comsol Multiphysics, Abaqus, and ANSYS respectively.
1. Introduction and Background

Buckling is characterized by a sudden failure of a structural member subject to high compressive stress. A buckling analysis is particularly important for axial loaded members because the subjected compressive stress at the point of failure is less than the materials ultimate compressive stress. As a result, special consideration must be given to the compressive load and the components geometry when designing axially compressed members in order to ensure failure will not occur from elastic instability. This paper shall determine the critical load of an axial loaded column with several commercial Finite Element Analysis FEA software program, Comsol Multiphysics, Abaqus, and ANSYS, and compare the results with Euler's theoretical critical buckling load for accuracy.

1.1 Overview of Column Buckling Problem

The purpose of the following calculation is to determine the critical load that causes elastic instability for a circular column that is 10 meters in length and has a radius of 0.5 meters. The axial loaded column is made of structural steel. The structural properties of the column are listed in Table 1. It should be noted that the column is fixed at its base and is free to deflect at its tip when a load P is applied.

![Column Geometry](image)

**Table 1: Structural Steel Mechanical Properties**

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\rho) [kg/m(^3)]</td>
<td>7800</td>
<td>Density</td>
</tr>
<tr>
<td>(\nu) [Dim]</td>
<td>0.3</td>
<td>Poisson's ratio</td>
</tr>
<tr>
<td>(E) [Pa]</td>
<td>2e11</td>
<td>Young’s Modulus</td>
</tr>
</tbody>
</table>

**Figure 1:** Column Geometry
2. Theory and Methodology

For a straight, homogenous, pin-ended column that is being subjected to a compressive load the pinned ends cause the moment to be zero at each end of the column. As such, this is considered the fundamental beam buckling problem\[1\]. The described problem and associated free body diagram are provided in Figure 2 and 3 respectively.

![Figure 2: Pin-Pin Axially Loaded Column](image1)

![Figure 3: Pin-Pin Free Body Diagram](image2)

The bending moment at any section, \( M = -Pv \), when inserted into the equation for the elastic behavior of a beam, \( EIv'' = M \), yields

\[
EI \frac{d^2v}{dx^2} + Pv = 0 \tag{1}
\]

Eq.(1) is an second order ordinary differential equation that can be readily solved using the general solution:

\[
v(x) = c_1 \sin(cx) + c_2 \cos(cx) \tag{2}
\]

where \( c \) is equal to \( \frac{P}{\sqrt{EI}} \). Substituting \( c \) into the general solution yields the following general solution:

\[
v(x) = c_1 \sin \left( \frac{P}{\sqrt{EI}} x \right) + c_2 \cos \left( \frac{P}{\sqrt{EI}} x \right) \tag{3}
\]

Next, one can apply the first boundary condition \( v=0 \) at \( x=0 \) to obtain
\[ v(0) = c_1 \sin\left(\frac{p}{\sqrt{EI}} \cdot 0\right) + c_2 \cos\left(\frac{p}{\sqrt{EI}} \cdot 0\right) \quad \text{Eq. (4)} \]

\[ 0 = c_2 \cos\left(\frac{p}{\sqrt{EI}} \cdot 0\right) \quad \text{Eq. (5)} \]

\[ c_2 = 0 \quad \text{Eq. (6)} \]

Then, one can apply the second boundary condition \( v=0 \) at \( x=L \) to obtain

\[ v(L) = c_1 \sin\left(\frac{p}{\sqrt{EI}} \cdot L\right) \quad \text{Eq. (7)} \]

\[ 0 = c_1 \sin\left(\frac{p}{\sqrt{EI}} \cdot L\right) \quad \text{Eq. (8)} \]

It can concluded that \( c_1 \) cannot equal 0 as it would result in a trivial solution. Therefore, it must be assumed that:

\[ 0 = \sin\left(\frac{p}{\sqrt{EI}} \cdot L\right) \quad \text{Eq. (9)} \]

which is only satisfied when:

\[ \frac{p}{\sqrt{EI}} \cdot L = n\pi \quad \text{where} \ n = 1, 2, 3 \ldots \quad \text{Eq. (10)} \]

The value of \( P \) in Eq. (10) is the load in which the column can maintain its deflected shape. The critical load for each mode, \( n \), of deflection can be obtained by solving Eq. (10) for \( P \).

Rearranging Eq. (10) to solve for \( P \) yields:

\[ (P_{cr})_n = \frac{n^2 \pi^2 EI}{L^2} \quad \text{Eq. (11)} \]

It should be noted that the length \( L \) represents the original length of the column. The deflection is expected to occur about the axis through the centroid for which the second moment of area, \( I \), is the smallest\(^{[1]} \). The smallest critical load is known as Euler's buckling load for a pin-ended column and occurs when \( n=1 \). Therefore, the critical load for a pin-ended column can be represented as:

\[ P_{cr} = \frac{\pi^2 EI}{L^2} \quad \text{Eq. (12)} \]

### 2.1 Other End Condition

Based on the derivation of the critical load for a pin-ended column it is clear that the critical load depends on the selected end conditions of the column. By substituting the appropriate boundary
conditions into the general solution, Eq. (3), one can determine the resulting critical load for
various boundary condition combinations. Alternatively, one can slightly modify the derived
Euler buckling formula for a pin-ended column to make it applicable to a large variety of end
conditions. The subject modification can be represented as:

\[ P_{cr} = \frac{\pi^2 E I}{L_e^2} \]  
Eq. (13)

The modified Euler buckling formula, Eq. (13), replaces the original column length, \( L \), with the
effective column length, \( L_e \). The effective column length corresponds to the distance between
points of inflection on the elastic curve\(^\text{[1]}\). Therefore, the effective column length, \( L_e \), for a pin-
ended column is equal to the column length \( L \). Alternatively, the effective column length for a
column fixed at one end and free at the tip is equal to \( 2L \). This is illustrated in Figure 4.

\[ P_{cr} = \frac{\pi^2 E I}{(2L)^2} \]  
Eq. (14)

\[ P_{cr} = \frac{\pi^2 + 2e11 \times 10^5 \times \pi^4}{(2 \times 10 \times 10^{-2} \times 0.5)^4} \]  
Eq. (15)

\[ P_{cr} = 2.422e8 \]  
Eq. (16)

**Figure 4:** Effective lengths for Pinned-Pinned and Fixed-Free Boundary Conditions

### 2.2 Buckling Theoretical Solution for a Fixed-Free Column

The following calculation shall determine the theoretical critical load that causes elastic
instability for a circular column that is 10 meters in length and has a radius of 0.5 meters made of
structural steel. The column is fixed at its base and is free to deflect at its tip when a load \( P \) is
applied.
3. Results & Discussion

3.1 Comsol Multiphysics

Comsol Multiphysics is a finite element analysis, solver, and simulation software package for various physics and engineering applications. The following sections shall provide details about the Finite Element Model FEM that was used to determine the critical load of a column with the geometry and material properties as defined in section 1.1. The subject FEM was created using Comsol’s 3D solid mechanics linear buckling module.

3.1.1 Geometry & Materials

In order to model the geometry described in section 1.1, a cylinder with a radius of 0.5 meters and a length of 10 meters has been created. The cylinder geometry generated in Comsol is illustrated in Figure 5. Next, steel material properties were assigned to the newly created cylinder region as illustrated in Figure 6. The assigned material properties are provided in Table 2.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ [kg/m³]</td>
<td>7800</td>
<td>Density</td>
</tr>
<tr>
<td>ν [Dim]</td>
<td>0.3</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>E [Pa]</td>
<td>2e11</td>
<td>Young’s Modulus</td>
</tr>
</tbody>
</table>

Table 2: Structural Steel Mechanical Properties

3.1.2 Loads and Boundary Condition

Loads and boundary conditions must be applied to the cylinder geometry in order to accurately model a column with fixed-free boundary conditions. This is done by placing a 1 Newton compressive point load at the tip of the column and applying a fixed boundary condition at the base of the column. It should be noted that a 1 Newton compressive unit load force has been
selected such that the resulting eigenvalue will represent the column’s critical load. Figure 7 illustrates the applied load and boundary conditions applied to the FEM.

Figure 7: Load and Boundary Condition Locations

3.1.3 Mesh Refinement Study

In order to validate the results calculated in Comsol Multiphysics a mesh refinement study shall be performed. The mesh refinement study consists of three mesh extensions with varying mesh densities. Each mesh consists solely of linear tetrahedral elements. The mesh for each of the cases analyzed is illustrated in Figures 8-10.
3.1.4 Results

Figures 11-13 provide the first mode deflected shape calculated by Comsol Multiphysics for each of the mesh refinement studies analyzed. Additionally, Table 3 provides the calculated critical load and compares the results with Euler’s theoretical solution for column buckling.

![Figure 11: Case 1 1st Mode Deflection](image1)
![Figure 12: Case 2 1st Mode Deflection](image2)
![Figure 13: Case 3 1st Mode Deflection](image3)

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Analytical Critical Load [N]</th>
<th>Theoretical Critical Load [N]</th>
<th>Percent Error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>2.422e8</td>
<td>2.422e8</td>
<td>0.00%</td>
</tr>
<tr>
<td>Case 2</td>
<td>2.414e8</td>
<td>2.422e8</td>
<td>0.33%</td>
</tr>
<tr>
<td>Case 3</td>
<td>2.410e8</td>
<td>2.422e8</td>
<td>0.49%</td>
</tr>
</tbody>
</table>

Table 3: Critical Load Results

The results from Table 3 compare very well to Euler’s theoretical solution for column buckling. The coarsest mesh, Case 1, resulted in the smallest percent error when compared to the more refined cases. However, it should be noted that this is only a coincident because as the mesh is refined the Comsol solution converges to 2.410e8 Newtons. The convergence to 2.410e8 Newtons is clearly evident when comparing the difference between the calculated values between cases 1 and 2 and then cases 2 and 3. Based on the results of the mesh refinement study, Comsol Multiphysics is able to calculate the critical buckling load of a column accurately within half a percent of the theoretical value.
3.2 Abaqus

Abaqus is a software suite for finite element analysis used for modeling and analysis of mechanical components and assemblies. Abaqus is capable of both pre and post processing. The following sections shall provide details about the Finite Element Model FEM that was used to determine the critical load of a column with the geometry and material properties as defined in section 1.1. The physics for the subject FEM were modeled by adding a static linear perturbation step in Abaqus.

3.2.1 Geometry & Materials

In order to model the geometry described in section 1.1, a cylinder with a radius of 0.5 meters and a length of 10 meters has been created. The cylinder geometry generated in Abaqus is illustrated in Figure 14. Next, steel material properties were assigned to the newly created cylinder region as illustrated in Figure 15. The assigned material properties are provided in Table 4.

![Column Geometry](image1.png) ![Section Assigned Steel Material Properties](image2.png)

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$ [kg/m$^3$]</td>
<td>7800</td>
<td>Density</td>
</tr>
<tr>
<td>$\nu$ [Dim]</td>
<td>0.3</td>
<td>Poisson's ratio</td>
</tr>
<tr>
<td>$E$ [Pa]</td>
<td>2e11</td>
<td>Young's Modulus</td>
</tr>
</tbody>
</table>

Table 4: Structural Steel Mechanical Properties

3.2.2 Loads and Boundary Condition

Loads and boundary conditions must be applied to the cylinder geometry in order to accurately model a column with fixed-free boundary conditions. Similar to what was previously done in the Comsol Multiphysics model, a 1 Newton compressive point load will be applied at the tip of the column and a fixed (Encastre) boundary condition at the base of the column. In order to ensure that the point load is applied at the center of the column a reference point is created at the center tip of the FEM and a kinematic coupling constraint is assigned to the reference point. The
kinematic coupling constrains the reference point’s six degrees of freedom (U1, U2, U3, UR1, UR2, and UR3) to the column's tip surface. Next, an Encastre boundary condition is applied to the base of the column. The Encastre boundary condition restrains movement in the U1, U2, U3, UR1, UR2, and UR3 direction.

![Figure 16: Kinematic Coupling Constraint](image)

![Figure 17: Encastre Boundary Condition & Applied Unit Load](image)

### 3.2.3 Mesh Refinement Study

In order to validate the results calculated in Abaqus a mesh refinement study shall be performed. The mesh refinement study consists of three mesh extensions with varying mesh densities. Each mesh consists solely of linear four node tetrahedral elements (C3D4). The mesh for each of the cases analyzed is illustrated in Figures 18-20.

![Case 1: 285 DOF](image)

![Case 2: 490 DOF](image)

![Case 3: 48,145 DOF](image)
3.2.4 Results

Figures 11-13 provide the first mode deflected shape calculated by Abaqus for each of the mesh refinement studies analyzed. Additionally, Table 3 provides the calculated critical load and compares the results with Euler’s theoretical solution for column buckling.

![Figure 21: Case 1 1st Mode Deflection](image1)

![Figure 22: Case 2 1st Mode Deflection](image2)

![Figure 23: Case 3 1st Mode Deflection](image3)

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Analytical Critical Load [N]</th>
<th>Theoretical Critical Load [N]</th>
<th>Percent Error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>3.102e8</td>
<td>2.422e8</td>
<td>2.86</td>
</tr>
<tr>
<td>Case 2</td>
<td>2.851e8</td>
<td>2.422e8</td>
<td>1.82</td>
</tr>
<tr>
<td>Case 3</td>
<td>2.447e8</td>
<td>2.422e8</td>
<td>1.52</td>
</tr>
</tbody>
</table>

Table 5: Critical Load Results

The results from Table 5 compare very well to Euler’s theoretical solution for column buckling. As one would assume the coarsest mesh, Case 1, resulted in the largest percent error when compared to the more refined cases. The convergence to 2.447e8 Newtons is clearly evident when comparing the difference between the calculated critical load value between cases 1 and 2 and then cases 2 and 3. Based on the results of the mesh refinement study, Abaqus is able to calculate the critical buckling load of a column accurately within half a 1.5 percent of the critical value.
3.3 ANSYS

ANSYS is an engineering simulation software suite used for a variety of engineering applications. ANSYS is capable of both pre and post processing. The following sections shall provide details about the Finite Element Model FEM that was used to determine the critical load of a column with the geometry and material properties as defined in section 1.1. The physics for the subject FEM were modeled by dragging and dropping the static structural and linear buckling modules into the project schematic window within ANSYS workbench.

3.3.1 Geometry & Materials

In order to model the geometry described in section 1.1, one should open the geometry step within the project schematic window and create a vertical line with a length of 10 meters. Next, a circular cross section with a diameter of 1 meter can be created. Once complete, one can select the line body from the tree outline and select the circular cross section within the detail view window. This will apply the cross section along the entire length of the line. The resulting column geometry is illustrated in Figure 24. Material properties can be applied to the newly created geometry by opening the model step within the project schematic window and applying steel material properties to the line body. The assigned material properties are provided in Table 6.

<table>
<thead>
<tr>
<th>Variable</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ρ [kg/m³]</td>
<td>7800</td>
<td>Density</td>
</tr>
<tr>
<td>ν [Dim]</td>
<td>0.3</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>E [Pa]</td>
<td>2e11</td>
<td>Young’s Modulus</td>
</tr>
</tbody>
</table>

Table 6: Structural Steel Mechanical Properties

Figure 24: Column Geometry
3.3.2 Loads and Boundary Condition

Loads and boundary conditions must be applied to the cylinder geometry in order to accurately model a column with fixed-free boundary conditions. This is done by opening the model step within the project schematic and applying the appropriate load and boundary conditions to the column geometry. The 1 Newton compressive point load can be applied by adding a force to the static structural step within the project outline and selecting the vertex at the free end of the column. Similarly, the fixed boundary condition can be applied by adding a remote displacement to the static structural step within the project outline and selecting the vertex at the fixed end of the column. The constrained degrees of freedom (U1, U2, U3, UR1, UR2, and UR3) will need to be set to 0 within the details window of the remote displacement boundary condition. Figure 25 illustrates the applied load and boundary conditions applied to the FEM.

3.3.3 Mesh Refinement Study

In order to validate the results calculated in ANSYS a mesh refinement study shall be performed. The mesh refinement study consists of three mesh extensions with varying mesh densities. The mesh for each of the cases analyzed is illustrated in Figures 26-28.
Case 1

Case 2

Figure 26: 21 DOF

Figure 27: 41 DOF

Case 3

Figure 28: 201 DOF
3.3.4 Results

Figures 29-31 provide the first mode deflected shape calculated by ANSYS for each of the mesh refinement studies analyzed. Additionally, Table 7 provides the calculated critical load and compares the results with Euler’s theoretical solution for column buckling.

<table>
<thead>
<tr>
<th>Case Number</th>
<th>Analytical Critical Load [N]</th>
<th>Theoretical Critical Load [N]</th>
<th>Percent Error [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>2.406e8</td>
<td>2.422e8</td>
<td>0.16</td>
</tr>
<tr>
<td>Case 2</td>
<td>2.406e8</td>
<td>2.422e8</td>
<td>0.16</td>
</tr>
<tr>
<td>Case 3</td>
<td>2.406e8</td>
<td>2.422e8</td>
<td>0.16</td>
</tr>
</tbody>
</table>

Table 7: Critical Load Results

The results from Table 7 compare very well to Euler’s theoretical solution for column buckling. Each of the three mesh refinement studies resulted in the same analytical critical load value of 2.406e8. It should be noted that ANSYS default static structural elements were used in lieu of tetrahedral elements. The subject element provided stable results with a minimal amount of nodes required. Based on the results of the mesh refinement study, ANSYS is able to calculate the critical buckling load of a column accurately within 0.16 percent of the critical value.
4. Conclusion

Buckling analysis are particularly important for axial loaded members because the subjected compressive stress at the point of failure is less than the materials ultimate compressive stress. As a result, special consideration must be given to the critical load when designing axially compressed members. The analytical critical loads determined by Comsol Multiphysics, Abaqus, and ANSYS compared very with Euler’s theoretical critical buckling load. After mesh refinement studies the percent error determined for each critical load value was 0.49, 1.52, and 0.16 for Comsol Multiphysics, Abaqus, and ANSYS respectively. It should be noted that both the Comsol Multiphysics and Abaqus FEM utilized tetrahedral elements. Typically, tetrahedral elements are not suited for structural calculations unless the mesh is very dense. This becomes evident when reviewing the FEM mesh refinement studies. The required number of nodes to obtain accurate results is very large when compared to the ANSYS FEM which did not utilize tetrahedral elements.
5. References