Problem:

The hydraulic cylinder shown in the figure has a 3 in bore and is to operate at a pressure of 800 psi. With the clevis mount shown, the piston rod should be sized as a column with both ends rounded for any plane buckling. The rod is to be made of forged AISI 1030 steel without further heat treatment.

(a) Use a design factor $n=3$ and select a preferred size for the rod diameter if the column length is 60 in.
(b) Repeat part (a) but for a column of 18 in
(c) What factor of safety actually results for each of the cases above

Joseph Shigley and Charles Mischke. *Mechanical Engineering Design*
Anticipated time to complete this tutorial: 30 minutes

**Tutorial Overview**
This tutorial is divided into five parts:
1) Tutorial Basics
2) Preprocessing
3) Solution
4) Post Processing
5) Hand Calculations

**Audience**
This tutorial assumes familiarity of ANSYS 8.0; therefore, it does not go into step by step detail.

**Prerequisites**
1) ANSYS 8.0 in house “Structural Tutorial”
2) Completion of all Basic Machine Design Tutorials
3) Completion of three or more Guided Machine Design Tutorials
4) Completion of Buckling of Euler Column Guided Tutorial

**Objectives**
1) Use Pipe elements to create geometry
2) Use advanced solution techniques to solve for the critical buckling load
3) Use symmetry to simplify end conditions

**Outcomes**
1) Explore possibilities with the graphical user interface (GUI)
2) Learn how to create and mesh simple geometries
3) Increase efficiency in problem set up and solving speed
In this tutorial:

- Instructions appear on the left.
- Visual aids corresponding to the text appear on the right.
- All commands on the toolbars are labeled. However, only operations applicable to the tutorial are explained.

The instructions should be used as follows:

- **Bold** > Text in bold are buttons, options, or selections that the user needs to click on

Example: 
> **Preprocessor > Element Type > Add/Edit/DeleteFile**

would mean to follow the options as shown to the right to get you to the **Element Types** window

- **Italics** > Text in italics are hints and notes

- **MB1** > Click on the left mouse button
- **MB2** > Click on the middle mouse button
- **MB3** > Click on the right mouse button

Some basic ANSYS functions are:

To **rotate** the models use **Ctrl** and **MB3**.

To **zoom** use **Ctrl** and **MB2** and move the mouse **up** and **down**.

To **translate** the models use **Ctrl** and **MB1**.
1) Change the working directory, jobname, and title of your project.

2) Set Preferences to Structural.

3) Add a Pipe16 element.

4) Set the Real Constants for Pipe 16.
   - Outside diameter: 1.433
   - Wall thickness: 0.7165

5) Create a Material Model.
   - E: $30\times10^6$
   - PRXY: .3

6) Create two Keypoints.
   - KP1: (0,0,0)
   - KP2: (30,0,0)

7) Create a Line between keypoint one and two.

8) Change the Number of element divisions to 30.
   - > Preprocessor > Meshing
   - > Size Cntrls > Manual Size > Lines
   - > All Lines

9) Mesh the line (use > Mesh > Lines command)

Your model should look similar to the picture shown below.
10) Apply **Structural Displacement** constraint to the far left keypoint. Constrain all of its degrees of freedom.

11) Apply a **Structural Force** to the far right keypoint. Give it a value of 1 and put it in the negative X direction. (i.e. a value of -1)

Your model should now look like the picture shown to the right.
12) > Solution > Analysis Type
   > New Analysis

Select Static and click ok.

13) Open up the unabridged menu and then open the analysis options. Select to include Prestresses.
   > Solution > Unabridged Menu

then

   > Solution > Analysis Type
   > Analysis Options

then

next to stress stiffness or prestress select Prestress ON

14) Solve the model
   > Solution > Solve > Current LS

15) When it is done solving select Finish
   > Main menu > Finish
16) Setup the system to find the critical loads.

> Solution > Analysis Type
> New Analysis

Select **Eigen Buckling** and click ok.
**Close** any warning message that may have appeared when you clicked New Analysis.

17) Set up the buckling options

> Solution > Analysis Type
> Analysis Options

Select **Block Lanczos** and put a value of 1 for the **number of modes to extract**

18) Solve the model again.

> Solution > Solve > Current LS

19) When it is done solving select **Finish**.

> Main menu > Finish

20) Set up an expansion pass to extract the critical load data

> Solution > Load Step Opts
> Expansion Pass > Single Pass
> Expand Modes

put a 1 in the **number of modes to expand**

21) Solve the model for the last time

> Solution > Solve > Current LS
22) Use the post processing tools to get the load value
   > General Postproc > List Results
   > Detailed Summary

The value of the critical load for part (a) is 16993 lb.

```
***** INDEX OF DATA SETS ON RESULTS FILE *****

   SET   TIME/FREQ   LOAD   STEP   SUBSTEP   CUMULATIVE
   1     16993.1     1       1       1
```

Part (b) can be easily solved by reusing the model from part (a). Delete the mesh and the line, create a new keypoint at (9,0,0).

Set the **Real Constants** for Pipe 16.
   Outside diameter: .785
   Wall thickness: .3925

Create a new line and mesh it using **Number of element divisions** of 9.

**Repeat** steps 11 - 21 to get the value of the critical load for part (b).

The value of the critical load for part (b) is 16331 lb.

```
***** INDEX OF DATA SETS ON RESULTS FILE *****

   SET   TIME/FREQ   LOAD   STEP   SUBSTEP   CUMULATIVE
   1     16931.1     1       1       1
```
Part (c) is solved using slightly bigger diameters. Assume 1.5 in for part (a) and .875 for part (b).

**Repeat** steps 11 - 21 to get the value of the critical load for part (a) and (b).

The value of the critical load for part (a) is 20398 lb.

```
***** INDEX OF DATA SETS ON RESULTS FILE *****
SET TIME/FREQ LOAD STEP SUBSTEP CUMULATIVE
 1 20398.   1  1  1
```

The value of the critical load for part (b) is 26098 lb.

```
***** INDEX OF DATA SETS ON RESULTS FILE *****
SET TIME/FREQ LOAD STEP SUBSTEP CUMULATIVE
 1 26098.   1  1  1
```

Therefore $n=20398/5650=3.61$ for part (a).

and $n=26098/5650=4.61$ for part (b).
Only the upper half of the column is modeled because of symmetry. The boundary conditions become free-fixed for the half symmetry model.

The critical load (Pcr) for Euler buckling is given by:

\[ P_{cr} = nF = nP_{pressure}A \]

\[ A = \frac{\pi D^2}{4} = \frac{\pi 3^2}{4} \]

\[ P_{pressure} = 800 \text{ psi} \]

\[ n = SF = 3 \]

\[ P_{cr} = 17000 \text{ lb} \]

\[ P = \frac{P_{cr}}{n} = 5650 \]

Using equation (3 -63)

\[ d = \left( \frac{64L^2 P_{cr}}{\pi^4 CE} \right)^{\frac{1}{4}} \]
Buckling of Hydraulic Cylinder

Hand Calculations

Where:

C = constant for conditions at the end of the column
E = young’s modulus
L = is the length of the column

For this problem:

C = 1 (Both ends rounded)
E = 30e6 psi
L = 60 in or 18 in

\[
d = \left( \frac{(64)(Lin)^2(17000lb)}{\pi^3(1)(30000000 psi)} \right)^{1/4}
\]

\[
d = 1.433in \quad \text{When } L = 60 \text{ in}
\]

\[
d = .785in \quad \text{When } L = 18 \text{ in}
\]

Computer output;

D=1.375''
A=1.484892 square in
KX= KY=.34375 in
L/KX = L/KY=174.54
EULER PX = PY = 14431.06 lb
JOHNSON PX = PY = .75651.27 lb

D=1.5''
A=1.7671 square in
KX= KY=.375 in
L/KX = L/KY=160
EULER PX = PY = 20438.66 lb
JOHNSON PX = PY = -56458.13 lb

D=.75''
A=.4417861 square in
KX= KY=.1875 in
L/KX = L/KX = 96
EULER PX = PY = 14193.51 lb
JOHNSON PX = PY = 14145.3 lb

D=.875''
A=.60132 square in
KX= KY=.21875 in
L/KX = L/KX = 82.28571
EULER PX = PY = 26295.24 lb
JOHNSON PX = PY = 24993.6 lb

(a) d=1.5 in \quad (b) d=.875
(c) \quad n=20439/5650=3.61
\quad N=26295./5650=4.65