Problem:

Link 2 shown in the figure, is 1 in wide, has .5 in-diameter bearings at the ends, and is cut from low-carbon steel bar stock having a minimum yield strength of 24 kpsi. The end-condition constants are C=1 (Rounded-Rounded) and C=1.2 (Fixed-Fixed) for buckling in and out of the plane of the drawing, respectively.

(a) Use a design factor n=5, find a suitable thickness for the link.
(b) Determine if the bearing stresses at O and B are of any significance.
Buckling of a Two Part Assembly

Overview

Anticipated time to complete this tutorial: 40 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 the reaction solution to obtain the transmitted load to the link
2) Use Beam 4 to create the link
3) Use advanced solution techniques to solve for the critical buckling load
4) Learn to apply end conditions to the system

Outcomes
1) Explore possibilities with the graphical user interface (GUI)
2) Learn how to create and mesh complex 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 **Plane42** element.

4) Create a **Material Model**.
   
   \[
   
   \begin{align*}
   E : & \quad 30e6 \\
   PRXY : & \quad .3 
   \end{align*}
   
   \]

5) Create three **Keypoints**.
   
   \[
   
   \begin{align*}
   KP1 : & \quad (0,0,0) \\
   KP2 : & \quad (0,-1.75,0) \\
   KP3 : & \quad (2.5,-1.75,0) 
   \end{align*}
   
   \]

6) Create **Lines** between each keypoint.

7) Create an arbitrary area by lines.
   
   > Preprocessor > Modeling > Create
   
   > Areas > Arbitrary > By lines

   Select all 3 lines.

8) Mesh the Area using the **Mesh Tool**

   Check the Smart Size box and adjust to fine 4. Be sure to select “Areas” in front of Mesh.

9) Apply **Structural Displacement** constraint to keypoint 2. Constrain all of its degrees of freedom.

10) Apply **Structural Displacement** constraint to keypoint 1. Constrain only UX and UY

11) Apply a **Structural Force** to keypoint 3. Give it a value of 180 and put it in the negative Y direction. (i.e. a value of -180)
Your model should now look like the picture shown below:

12) > Solution > Analysis Type
    > New Analysis

Select static and click ok.

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

14) Use the post processing tools to get the load value
    > General Postproc > List Results
    > Reaction Solu

Select Struct force FX. The values are shown to the right. You are done with the first part of the analysis. Save the files and exit ANSYS.
15) Open ANSYS, change the working directory, jobname, and title of your project.

16) Set **Preferences** to **Structural**.

17) Add a **Beam4** element.

18) Set the **Real Constants** for Beam4.
   - Cross-sectional area: .444
   - Area moment of inertia IZZ: .007294
   - Area moment of inertia IYY: .037
   - Thickness along Z axis TKZ: 1
   - Thickness along Y axis TKZ: .444

19) Create a **Material Model**.
   - E: 30e6
   - PRXY: .3

20) Create two **Keypoints**.
   - KP1: (0,0,0)
   - KP2: (0,0,41.67)

21) Create a **Line** between keypoint one and two.

22) Change the **Number of element divisions** to 20.
   - > Preprocessor > Meshing
   - > Size Cntrls > Manual Size > Lines
   - > All Lines

23) Mesh the line (use > **Mesh > Lines** command)

   Your model should look similar to the picture shown to the right.
24) Apply **Structural Displacement** constraint to keypoint 1. Constrain all of its degrees of freedom.

25) Apply **Structural Displacement** constraint to keypoint 2. Constrain all of its degrees of freedom but UZ.

26) Apply a **Structural Force** to keypoint 2. Give it a value of 1 and put it in the negative Z direction. (i.e. a value of -1)

Your model should now look like the picture shown below.
Buckling of a Two Part Assembly

Solution

27) > Solution > Analysis Type
> New Analysis

Select static and click ok.

28) 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

29) Solve the model

> Solution > Solve > Current LS

30) When it is done solving select Finish

> Main menu > Finish
31) 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.

32) 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

33) Solve the model again.
   > Solution > Solve > Current LS

34) When it is done solving select Finish
   > Main menu > Finish

35) 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

36) Solve the model for the last time
   > Solution > Solve > Current LS
37) 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 4975.1 lb.

<table>
<thead>
<tr>
<th>INDEX OF DATA SETS ON RESULTS FILE</th>
</tr>
</thead>
<tbody>
<tr>
<td>SET</td>
</tr>
<tr>
<td>-----</td>
</tr>
<tr>
<td>1</td>
</tr>
</tbody>
</table>

However, this is assuming a theoretical value of C=4. For this problem C=1.2 so the critical load becomes 1490.4 lb (see Hand calculation for explanations). The theoretical value is 1490 lb.
Buckling of a Two Part Assembly

Hand Calculations

\[ \Sigma \text{Moments at A} \]
\[ x(1.75) - 180(2.5) = 0 \]
\[ x = 257.14 \text{lb} \]

*Also*
\[ \tan^{-1}\left(\frac{1.75}{3}\right) = 30.25^\circ \]

*And*
\[ L = \sqrt{1.75^2 + 3^2} = (3.47)(12) = 41.7 \text{ in} \]

Therefore,
\[ P = \frac{257.14}{\cos(30.25)} = 297.69 \text{ lb} \]
\[ P_{cr} = nP \]
\[ P_{cr} = 5(298) = 1490 \text{ lb} \]

For Fixed-Fixed conditions
\[ P_{cr} = \left(\frac{C\pi^2EI}{L^2}\right) \]

With
\[ C = \text{constant for conditions} \]
\[ \text{at the end of the column} \]
\[ E = \text{young’s modulus} \]
\[ I = \text{Moment of inertia} \]
\[ L = \text{is the length of the column} \]

For this problem:
\[ C = 4 \text{ (Theoretical)} \]
\[ E = 30e6 \text{ psi} \]
\[ I = 0.00729 \]
\[ L = 41.7 \text{ in} \]

\[ P_{cr} = \left(\frac{(4)(\pi^2)(30e6)(0.00729)}{41.7^2}\right) \]
\[ P_{cr} = 4968 \text{ lb} \]

\[ P_{cr} = 4975 \text{ lb} \text{ From Ansys} \]

However, for this problem C is specified
\[ C = 1.2 \]

\[ P_{cr} = \left(\frac{(1.2)(\pi^2)(30e6)(0.00729)}{41.7^2}\right) \]

\[ P_{cr} = 1490 \text{ lb} \]

Which is comparable to the \( P_{cr} = nP \) shown earlier.

Computer output;
\[ A = 0.375 \text{ square in} \]
\[ KX = 0.289 \text{ in} \]
\[ KY = 0.108 \text{ in} \]
\[ L/KX = 144.29 \]
\[ L/KY = 384.77 \]
\[ EULER PX = 5321.06 \text{ lb} \]
\[ EULER PY = 897.92 \text{ lb} \]
\[ JOHNSON PX = 5202.91 \text{ lb} \]
\[ JOHNSON PY = -13501.23 \text{ lb} \]

\[ A = 0.5 \text{ square in} \]
\[ KX = 0.289 \text{ in} \]
\[ KY = 0.1445 \text{ in} \]
\[ L/KX = 144.29 \]
\[ L/KY = 288.58 \]
\[ EULER PX = 7094.74 \text{ lb} \]
\[ EULER PY = 2128.42 \text{ lb} \]
\[ JOHNSON PX = 6937.22 \text{ lb} \]
\[ JOHNSON PY = -4875.91 \text{ lb} \]

(a) Use \( t = 0.5 \text{ in} \)
(b) \( \delta_b = -298/0.5 = -600 \text{ psi} \)
Not significant.