Assignment #4

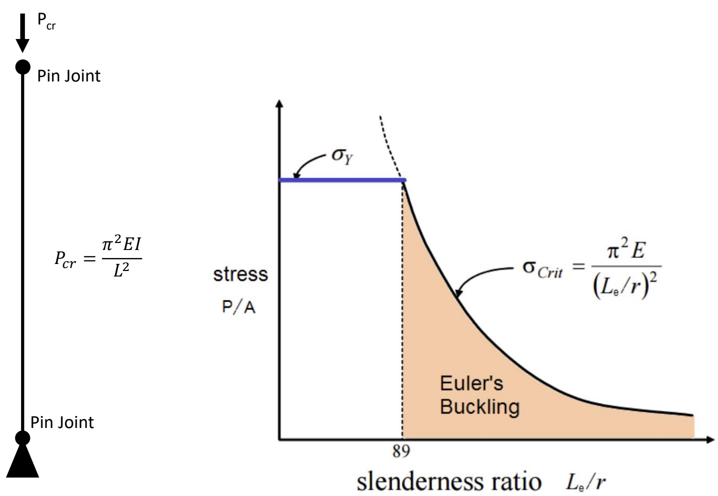
Buckling Analysis of a Symmetric Beam

Objective

The objective of this lab is to use ANSYS APDL to model and analyze the buckling behavior of a symmetric beam using Beam Element #188. The beam will have a solid circular cross-section, and both analytical and numerical solutions will be used to calculate the critical buckling force. Students will explore how boundary conditions impact the buckling force by comparing pinned and fixed supports.

Problem Description

You are tasked with modeling a symmetric beam with a solid circular cross-section and a diameter of 10 inches. The slenderness ratio (L/r) is 120, where r is the radius of gyration ($r = \sqrt{I/A}$). Assume the material is A500 steel, and both ends of the beam are pinned. Using this setup, you will calculate and compare analytical and numerical results for the yield force and critical buckling force. You will also modify the boundary condition at the bottom end to a fixed constraint and observe how this affects the critical buckling force.



Your task is to:

- Calculate the analytical yield force for the beam.
- Calculate the analytical critical buckling force for the pinned-pinned condition.
- Use ANSYS APDL to calculate the critical buckling force and compare it with the analytical solution.

- Change the boundary condition at the bottom to fixed and calculate the new critical buckling force using both an adjusted analytical expression (effective length) and ANSYS. Compare the results.

Questions

1. What is the analytical yield force for the beam?
2. What is the analytical critical buckling force for the pinned-pinned configuration?
3. What is the critical buckling force obtained from ANSYS, and how does it compare to the analytical result?
4. What is the critical buckling force when the bottom end is fixed (both analytically and numerically)? Compare the results and discuss any differences.

ANSYS APDL Setup and Simulation

Follow these step-by-step instructions to complete the analysis using ANSYS APDL:

Pre-Processing

1. Define the element type and material properties:

```
Preprocessor > Element Type > Add > Beam 188
Preprocessor > Material Props > Material Models > Structural >
    Linear > Elastic > Isotropic > Specify Ex and PRXY
```

2. Define the section properties for the solid circular cross-section:

```
Preprocessor > Sections > Beam > Common Sections > Circle-R
Specify the radius (half of the 10-inch diameter)
```

3. Create keypoints for the beam ends:

```
Preprocessor > Modeling > Create > Keypoints > In Active CS >
Define two keypoints to represent the beam's ends, based on the slenderness ratio
```

4. Create a line between keypoints to define the beam:

```
Preprocessor > Modeling > Create > Lines > Lines > Straight Lines
Create a line connecting the two keypoints
```

5. Mesh the line:

```
Preprocessor > Meshing > Size Cntrl > Manual Size > Lines > Element Edge Length
Set element edge length to 10 inches (or a reasonable value for the beam length)
Preprocessor > Meshing > Mesh > Lines > Pick All
```

Applying Boundary Conditions and Loads

6. Apply boundary conditions to simulate the pinned joints:

```
Preprocessor > Loads > Define Loads > Apply > Structural >
    Displacement > On Keypoints
At the first keypoint (pinned joint), apply UX=UY=UZ=ROTY=0
At the second keypoint, apply UX=UZ=ROTY=0
```

7. Apply the load to the beam:

```
Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment >
    On Keypoints > Select the second keypoint > Apply a force Fy = -1
```

Solution Phase

8. Solve the static analysis:

```
Solution > Analysis Type > New Analysis > Static > OK
Solve > Current LS > OK
```

9. Perform the buckling analysis:

```
Solution > Analysis Type > Eigen Buckling > OK
Solution > Analysis Type > Analysis Options > Set NMODE=6 (6 buckling modes)
Solve > Current LS > OK
```

Post-Processing

10. Interpret the results:

```
General Postproc > Read Results > By Pick
The time value represents the force factor, and each set corresponds to a
   different mode.
Typically, the first mode is the most critical for static buckling problems.
```

11. Plot the deformed shape:

```
General Postproc > Plot Results > Deformed Shape
```