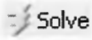
12. Solve 

The computed frequencies are shown below.

Table 9C.2 - Frequency Results (Refined Mesh)

Mode	Frequency (Hz)	Description
1	12.5	Rigid body
2	1067	Rotor twisting about vertical axis
3	1130	Side-to-side bending of brackets
4	1558	Fore-aft bracket bending
5	1659	Rotor twisting about horizontal axis
6	3673	Rotor flexure about a diameter

We see from these results that the both first mode (a rigid body mode) as well as the higher elastic modes are measurably influenced by the quality of the mesh. The contact description, as well as the distribution of mass and stiffness, strongly depends upon the mesh, and thus the frequencies calculated can be highly dependent upon the mesh quality.

Refine the mesh further as your computer hardware and time budget permit and you may see a point at which the results **converge** to values somewhat smaller than those shown. The calculated Mode 1 frequency is not zero but it's ten times smaller than our first estimate and almost 100 times smaller than Mode 2, the first elastic frequency.

13. Save your work.

9-6 BUCKLING LOADS

ANSYS Simulation provides tools for computing buckling estimates of elastic structures. We start by considering two problems that can readily be checked using Euler column theory from solid mechanics, a **fixed-free** column and a **pinned-pinned** column.

9-7 TUTORIAL 9D – FIXED-FREE COLUMN (FLAGPOLE)

First use DesignModeler or another solid modeler to create geometry for a solid bar **0.5 x 1 x 25 inch**. See below.

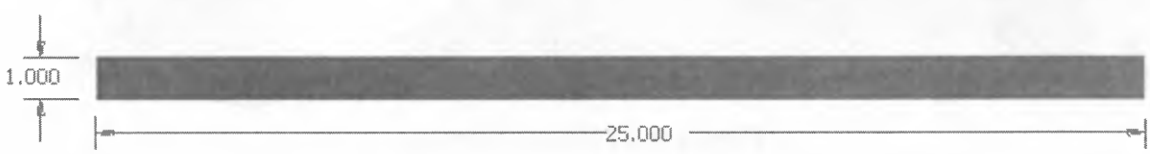


Figure 9-32 25 x 1 x 0.5 inch solid.

1. **Start ANSYS Workbench. Use DesignModeler to create** or **attach the column geometry**. Check the **units**. We will use the default **structural steel** material.

2. **Workbench Project > Double Click** or **Drag Static Structural** to add it to the **Project > Share the Geometry**.

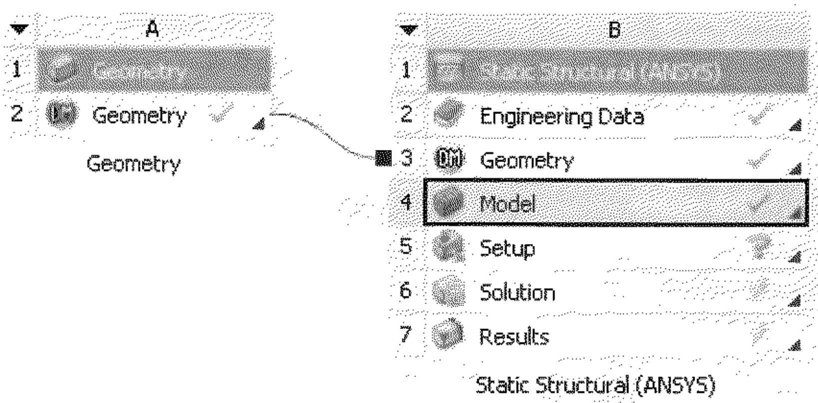


Figure 9-33 Insert Static Structural.

3. **Double Click Model** to start **ANSYS Mechanical**.

4. **Mesh > Advanced > Element Size > 0.5 in > Generate Mesh**

Fix the left end completely.

5. **Environment > Supports > Fixed Support**

On the right end apply a **unit load of 1.0 lbf** in the **negative X Direction**. The computed **buckling load** will be a **multiple** of this **unit load**.

6. **Environment > Loads > Force > X Component > -1.0**

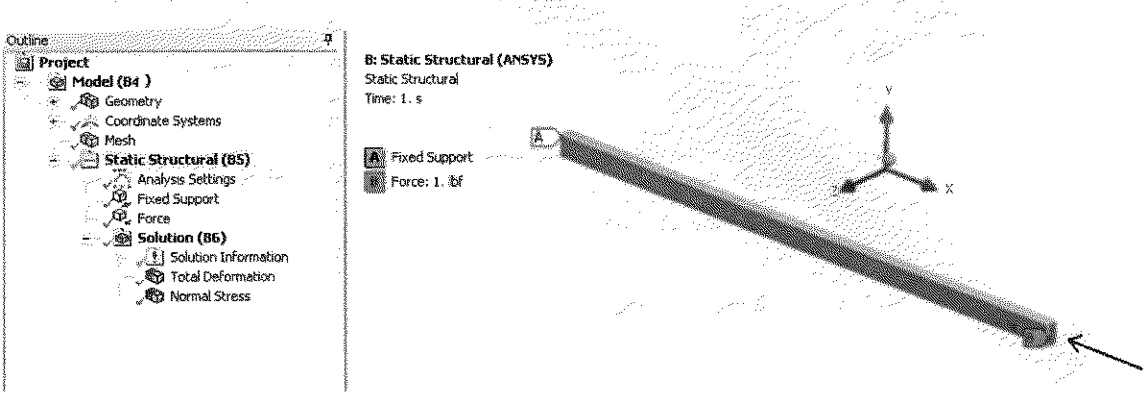


Figure 9-34 Boundary conditions and loading.

7. **Solution > Total Deformation & Normal Stress in X-Direction**

8. **Solve** Solve Find the deformation and stress. This step establishes the **internal stress distribution** associated with the **axial load.**

Next insert the **Linear Buckling** object.

9. **Right Click** on **cell B6 Solution > Transfer Data** to **New > Linear Buckling** (See next figure.)

This adds the Linear Buckling Analysis to the Project and transfers to model and data from the Static Structural Analysis including the solution results.

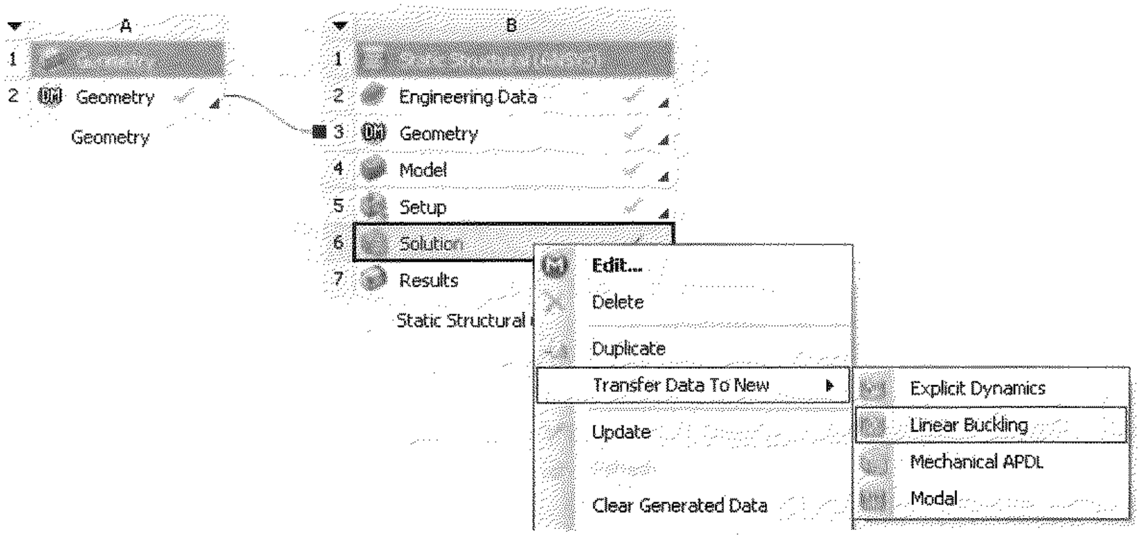


Figure 9-35 Transfer Data.

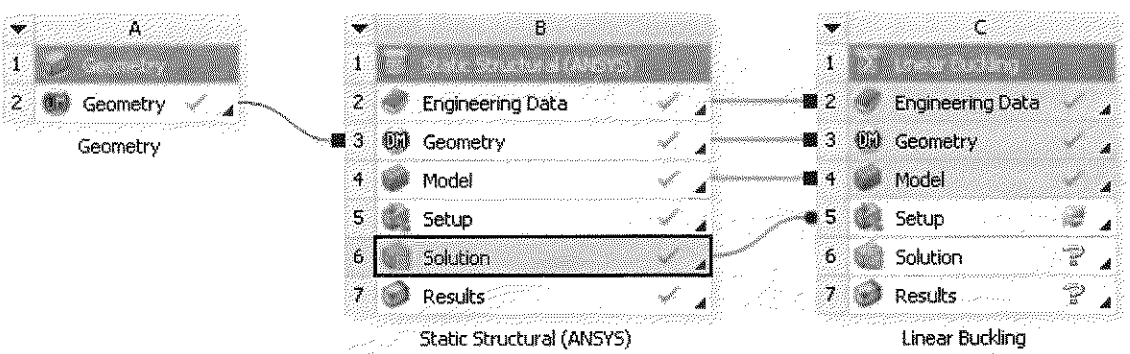


Figure 9-36 Linear Buckling.

9. **Select ANSYS Mechanical** from list of **processes running** B : Static Structural...

10. Analysis Settings Max Modes to Find = 1 and Insert > Total Deformation

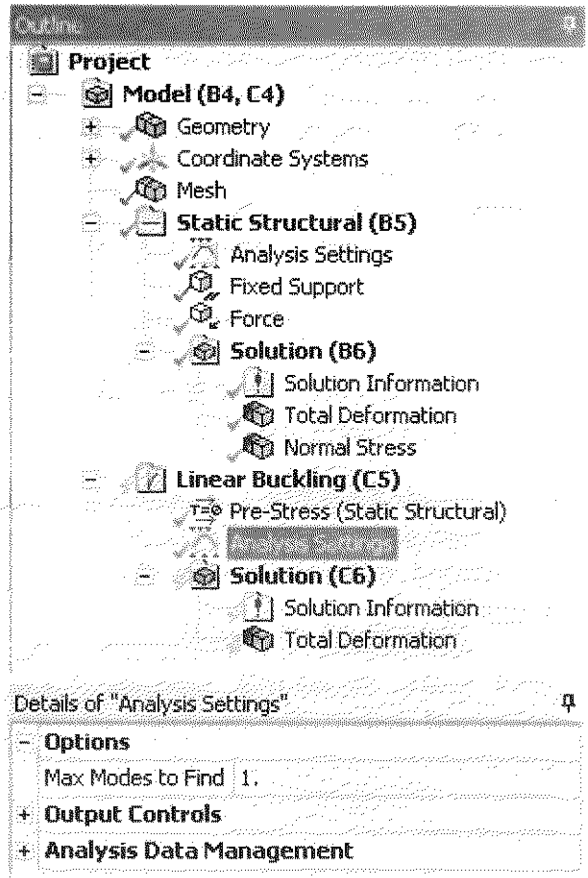


Figure 9-37 Model tree.

11. Solve Solve Find the Linear Elastic Buckling Load and the Deformed Shape.

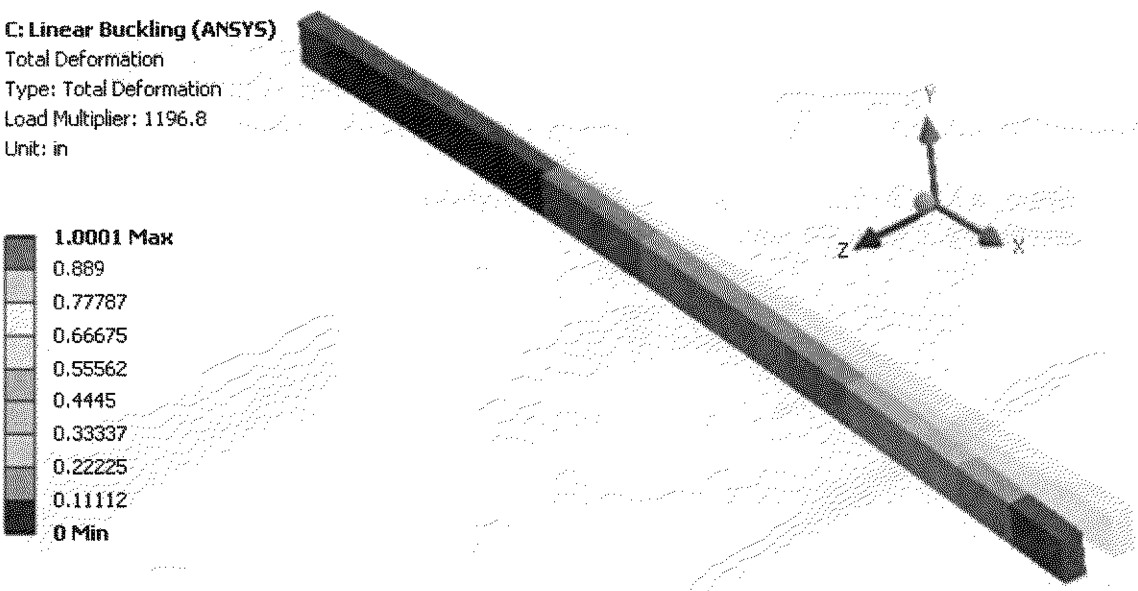


Figure 9-38 XZ Plane buckling.

Rotate the view to something like what is shown above. **Animation** shows clearly that the deformation is in the **XZ Plane** (the direction of weaker flexural stiffness). The critical load multiplier is 1197. Since we applied a 1.0 lbf load, our buckling load is **1197 x 1.0 lbf = 1197 lbf**. If we use solid mechanics theory to compute the solution to this problem (cross section base $w = 1.0$, height $h = 0.5$) we get a comparable critical load of **1193 lbf**. The scale on the deformation plot is arbitrarily chosen. The plot just shows the shape into which the column deforms if it becomes unstable.

To see if the mesh is influencing the results, reduce the element size and solve again.

To get more information about the behavior of this geometry, elect to solve for an additional buckling mode.

8. **Linear Buckling > Analysis Settings > Max Modes to Find > 2**

The second mode corresponds to buckling in the **XY Plane** with a load of about **4777 lbf**. This result also can be verified using column theory as before.

Why Static Structural Analysis + Linear Buckling Analysis? The static analysis performed before the buckling analysis is necessary in order to determine internal distribution of compressive stresses in the model. The structures in this example and the next are so simple it's obvious, but in more complex structures that distribution is not easy to see.

Save your work before moving to the next tutorial.

9-8 TUTORIAL 9E – BUCKLING OF A PINNED-PINNED COLUMN

We can use the geometry of Tutorial 9A to analyze buckling of a pinned-pinned column. The column is 0.5 x 1 and 25 inches between the 0.5 inch holes on each end.

1. **Start ANSYS Workbench. Use DesignModeler to create** or **attach the beam geometry**. Check the units.

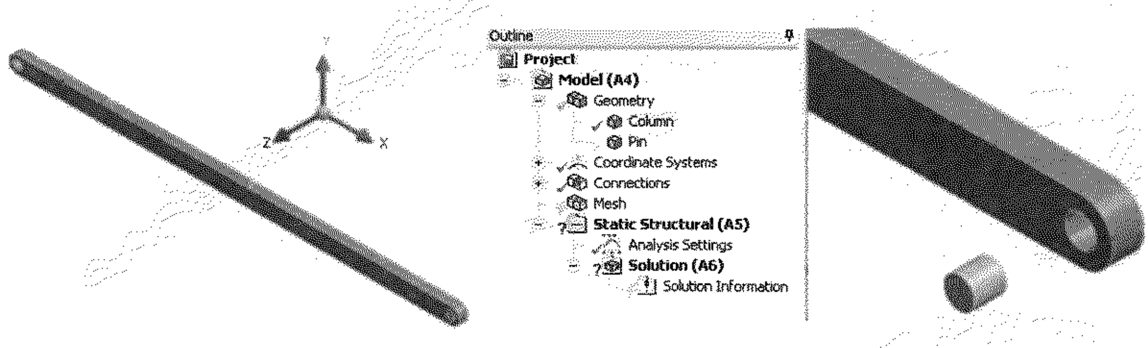


Figure 9-39 Pinned-pinned beam/column.

The column structural model requires the **freedom to move** along the **X Axis** in the direction of an applied 1 lbf load as discussed at the end of the previous tutorial. If we use a cylindrical support at both ends as before, the radial constraint will prevent motion in the X direction. A '**work around**' is to use a **cylindrical pin** on one end and load the column through the pin.

2. **Use** DesignModeler or your other favorite solid modeler to create a **0.5 inch diameter pin** for the hole in the right end of the beam and **assemble the two components.**

We have not taken this approach in earlier tutorials, but Geometry can be directly accessed through the structural static analysis module.

3. Start a new **Project**, insert the **Structural Static** analysis module and **double click Geometry**. This starts **DesignModeler**. **Open** the **pin-beam assembly** in DesignModeler.

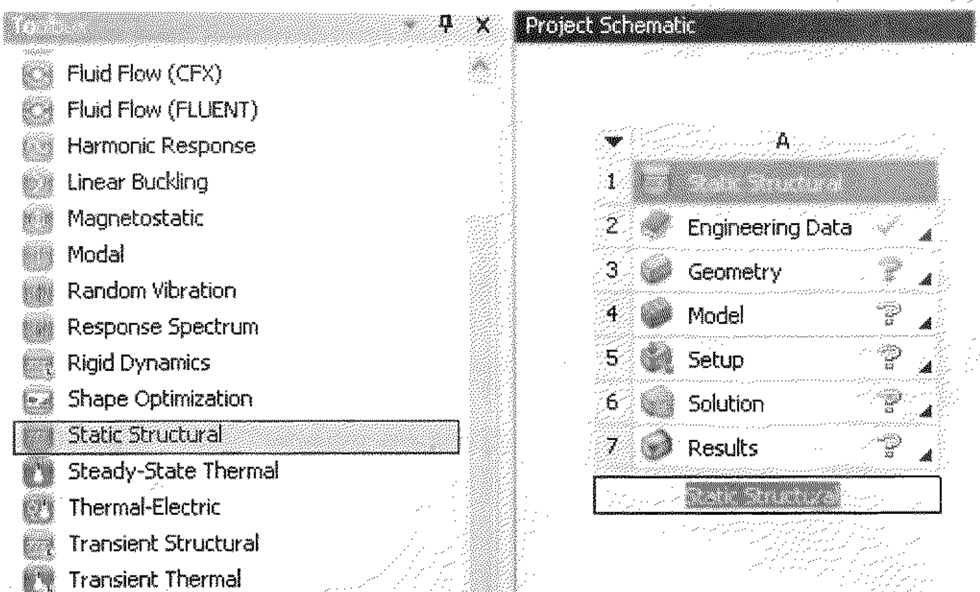


Figure 9-40 Pin contact with column.

Apply a **Cylindrical Support** to the left end of the column.

4. **Environment > Supports > Cylindrical Support**

5. **Ctrl Select** the **Inside surfaces** of the **cylindrical hole** at the **left end** of the **beam**.

6. **Radial > Fixed**

7. **Axial > Fixed**

8. **Tangential > Free** (See the next figure.)

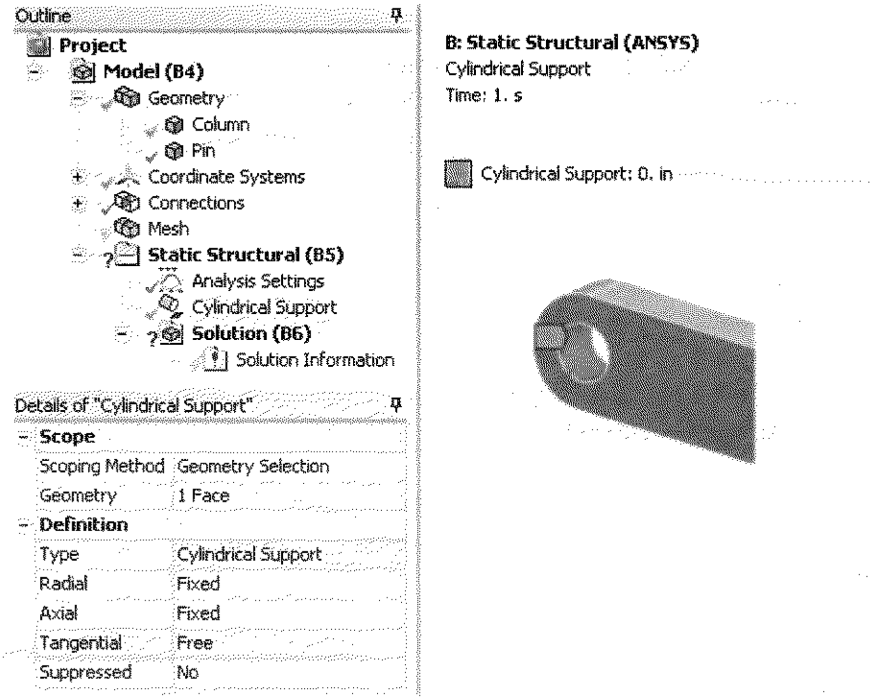


Figure 9-41 Cylindrical support.

On the **right end** set the **contact** between the **pin** and **hole** to be **No Separation**.

9. Contact > Contact Region > Type > No Separation

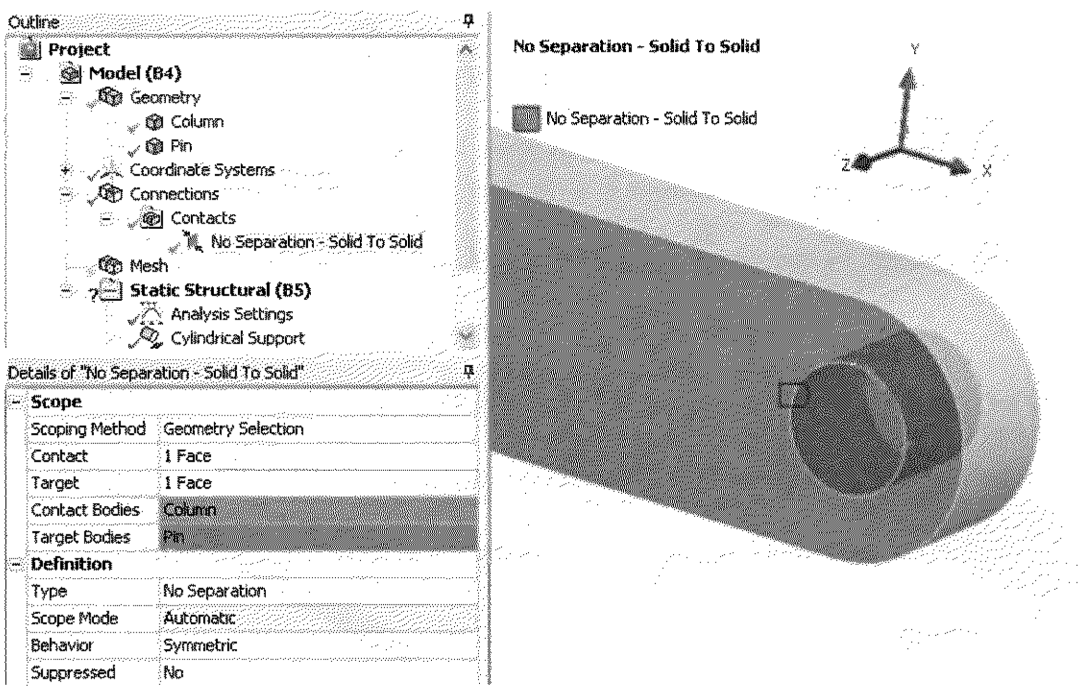


Figure 9-42 Pin contact with column.

Apply loads and boundary conditions to the **near** and **far side faces of the pin: Unit load** in the **negative X Direction, No displacement** in the **Y or Z Direction**.

Outline

Details of "Displacement"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	0. in (ramped)
Z Component	0. in (ramped)
Suppressed	No

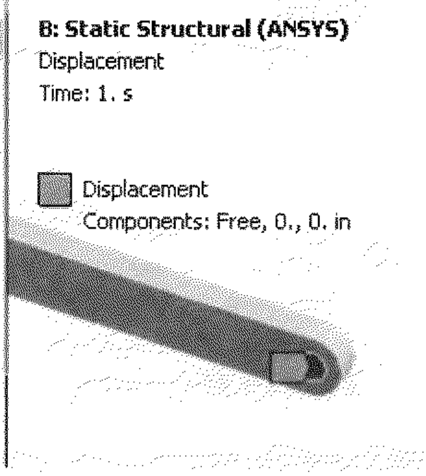


Figure 9-43 Displacement constraints applied to front and back of pin.

Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	-1. lbf (ramped)
Y Component	0. lbf (ramped)
Z Component	0. lbf (ramped)
Suppressed	No

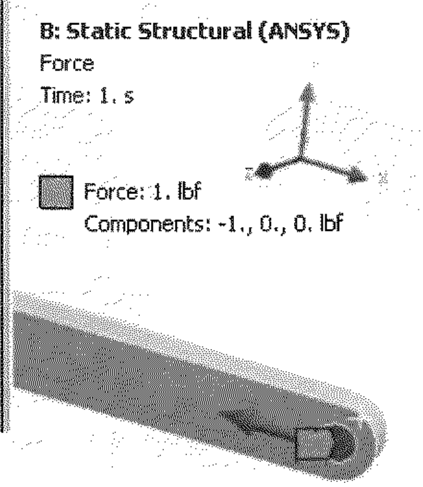


Figure 9-44 Loading applied to front and back of pin.

We need yet another constraint applied to the column to keep it from moving in the Z Direction. **Hide the pin** and apply a cylindrical constraint to the hole on the **right end**.

10. **Environment > Supports > Cylindrical Support**

11. **Ctrl Select** the **Inside surfaces** of the **cylindrical holes** on the **right end** of the **beam**.

12. **Radial > Free**

13. **Axial > Fixed**

14. Tangential > Free (See the next figure.)

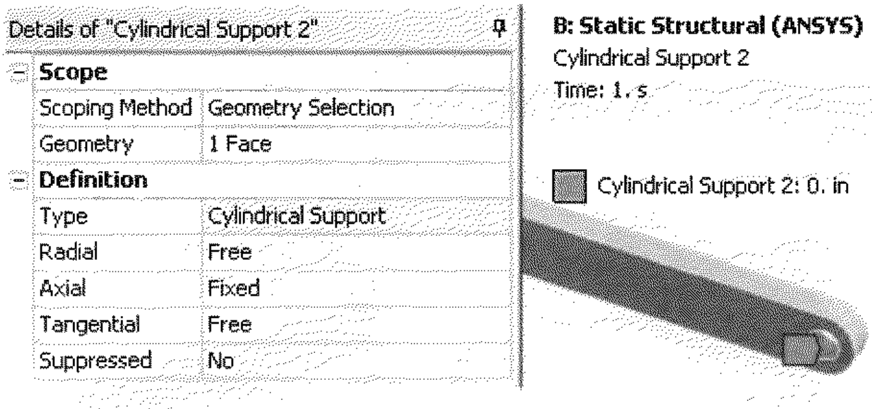


Figure 9-45 Cylindrical support to prevent Z-Direction movement.

15. Add **Total Deformation** and **Normal Stress** in the **X-Direction** to the **Solution**.

16. Set the mesh element size to 0.5 inch, **Solve** Solve.

17. Right Click on **cell B6 Solution > Transfer Data** to **New > Linear Buckling**

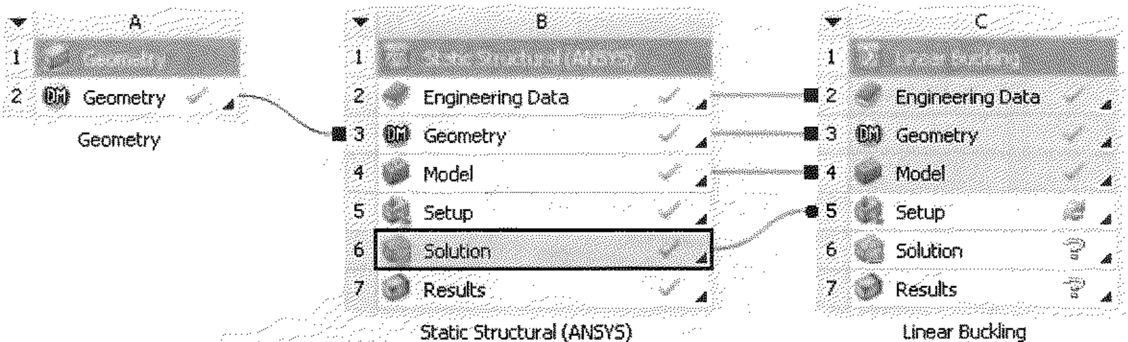


Figure 9-46 Transfer Data to Linear Buckling.

In linear buckling request that **two modes** be computed; the first mode is a **fixed-fixed XZ Plane mode**, and the second is the **pinned-pinned XY Plane mode** as shown below. (Pick two horizontal viewports.) Both elastic buckling loads are just over 19,000 lbf. This coincidence occurs because of different flexural inertias in the two planes combined with the different boundary conditions in the two planes of buckling.

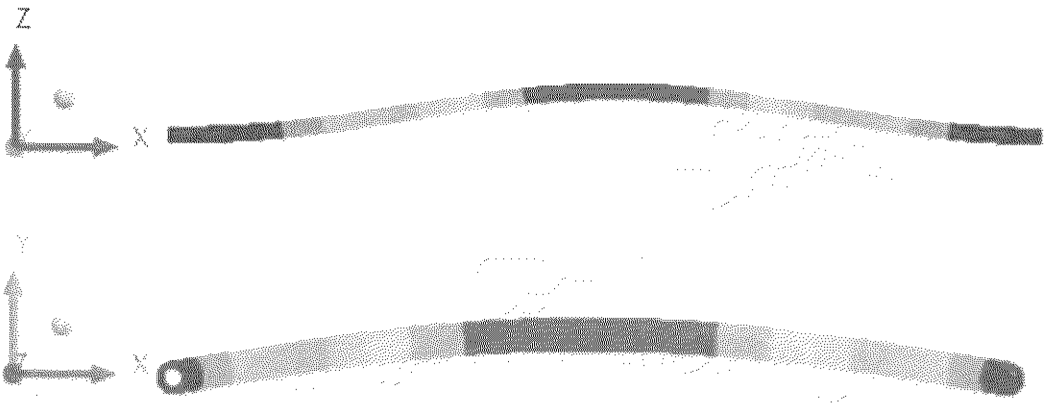


Figure 9-47 Fixed-fixed and Pinned-pinned column modes.

Once again the computed results can be verified using column theory from solid mechanics. **Save your work.**

9-9 TUTORIAL 9F – BUCKLING OF A BUILT-UP STRUCTURE

The next example in this chapter considers the determination of the buckling load estimate for the built-up structure shown in the figures below. The utility of finite element methods is evident for problems such as this one. It is a component of a truck dumping mechanism that is placed in compression when in service and is constructed by welding **structural steel** shapes together to produce the end result depicted. The construct is shown in two stages.

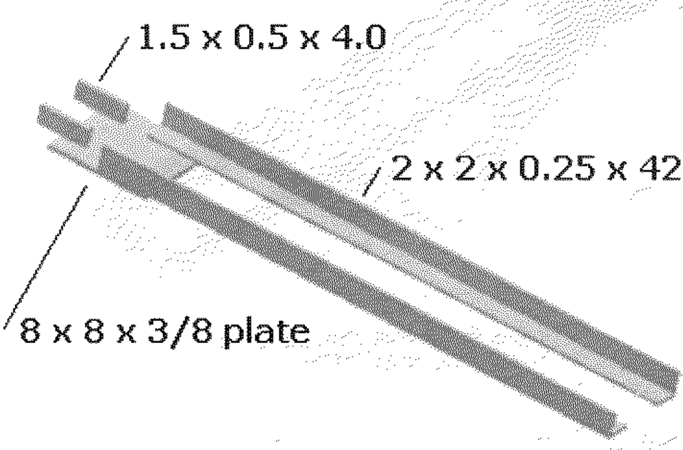


Figure 9-48 First stage of build up; dimensions in inches.

The left ends of the angle sections are placed at the center of the plate. The rectangular bars protrude 2 inches from the plate and are separated by 2 inches. The 0.25 x 2 x 8 braces shown below are equally spaced; the holes are centered 0.75 from the edges.