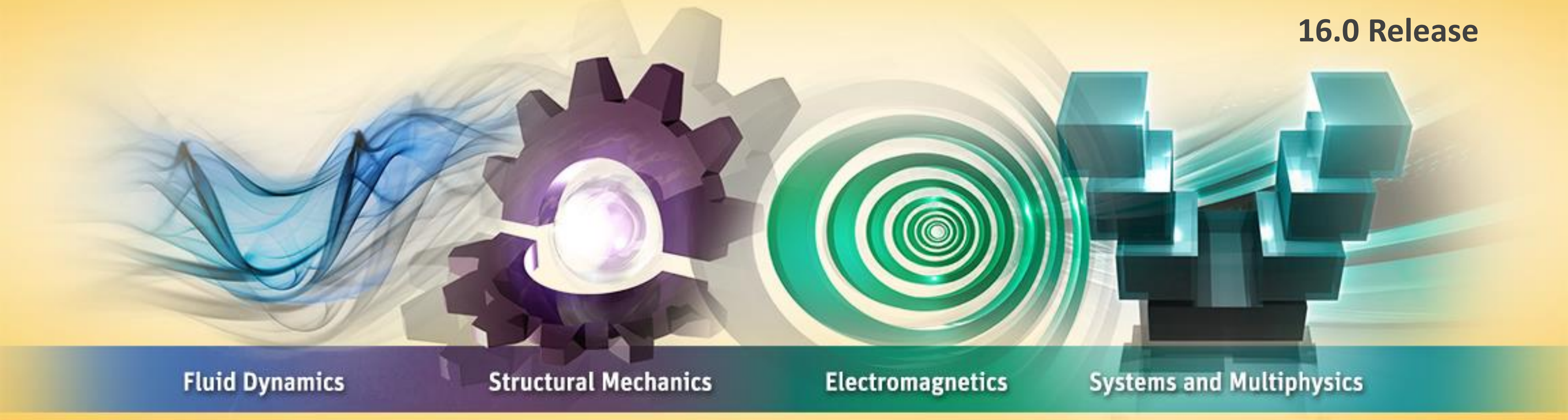
Appendix A
Linear Buckling Analysis

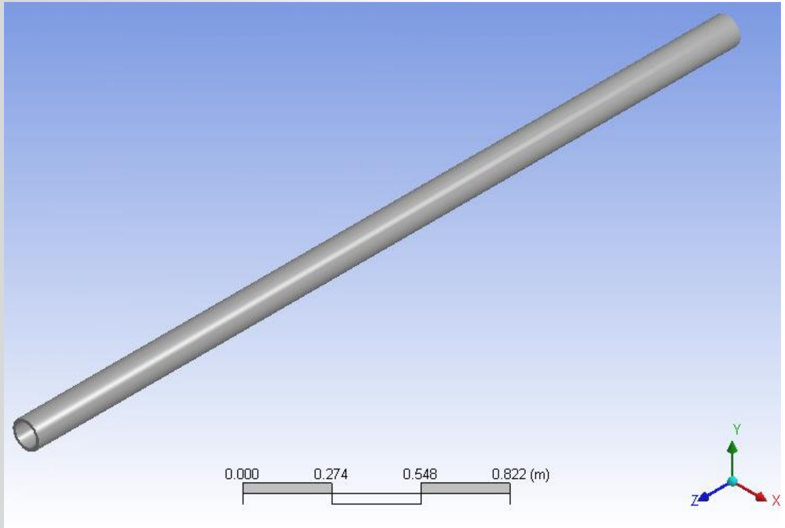
16.0 Release



Introduction to ANSYS Mechanical

Workshop App1 – Linear Buckling

- **Workshop WSApp1 – Linear Buckling**
- **Goal:**
  - **Verify linear buckling results in Mechanical for the pipe model shown below. Results will be compared to closed form calculations from a handbook.**



Goals

The goal in this workshop is to verify linear buckling results in ANSYS Mechanical. Results will be compared to closed form calculations from a handbook.

We will apply an expected load of 10,000 lbf to the model and determine its factor of safety.

Finally we will verify that the structure's material will not fail before buckling occurs.

Assumptions

The model is a steel pipe that is assumed to be fixed at one end and free at the other with a purely compressive load applied to the free end. Dimensions and properties of the pipe are:

OD = 4.5 in ID = 3.5 in. E = 30e6 psi, I = 12.7 in^4, L = 120 in.

In this case we assume the pipe conforms to the following handbook formula where P' is the critical load:

$$P' = K \bullet \left[\frac{\left(\pi^2 \bullet E \bullet I\right)}{L^2} \right]$$

For the case of a fixed / free beam the parameter K = 0.25.

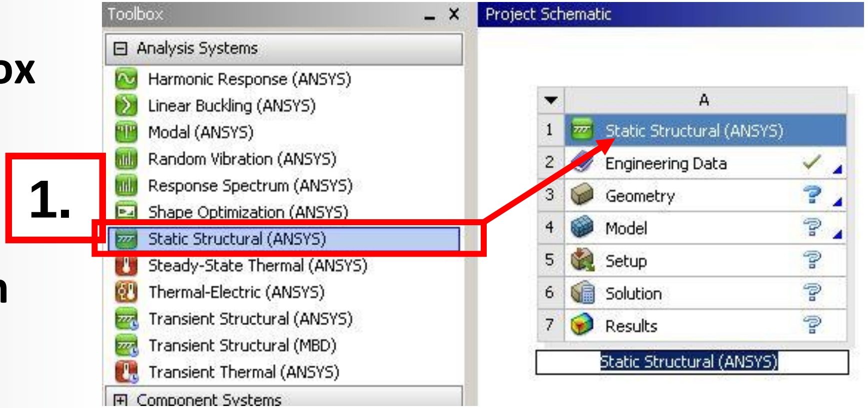
Assumptions

- **Using the formula and data from the previous page we can predict the buckling load will be:**

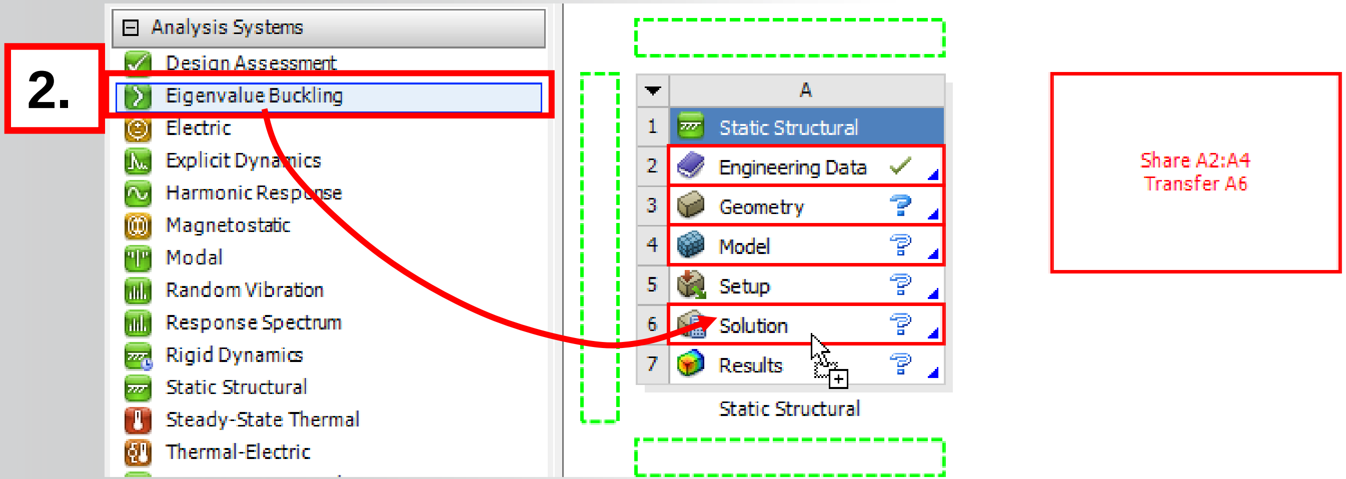
$$P' = 0.25 \bullet \left[\frac{\left(\pi^2 \bullet 30e6 \bullet 12.771\right)}{(120)^2} \right] = 65648.3 lbf$$

Project Schematic

1. **Double click Static Structural in the Toolbox to create a new system.**

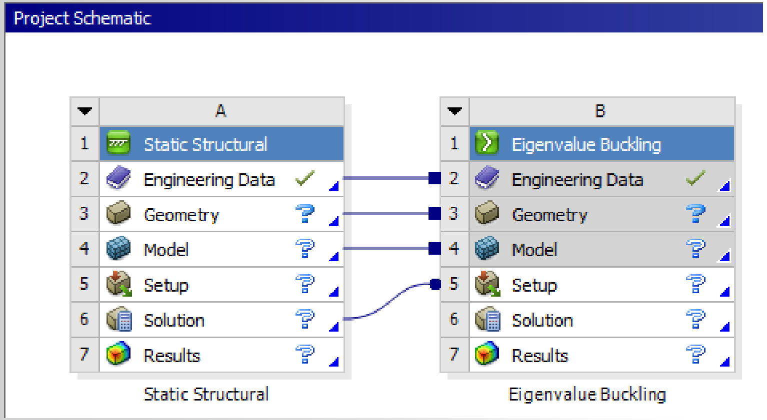


2. **Drag/drop a "Eigenvalue Buckling" system onto the "Solution" cell of the static structural system.**



Project Schematic

- **When the schematic is correctly set up it should appear as shown here.**



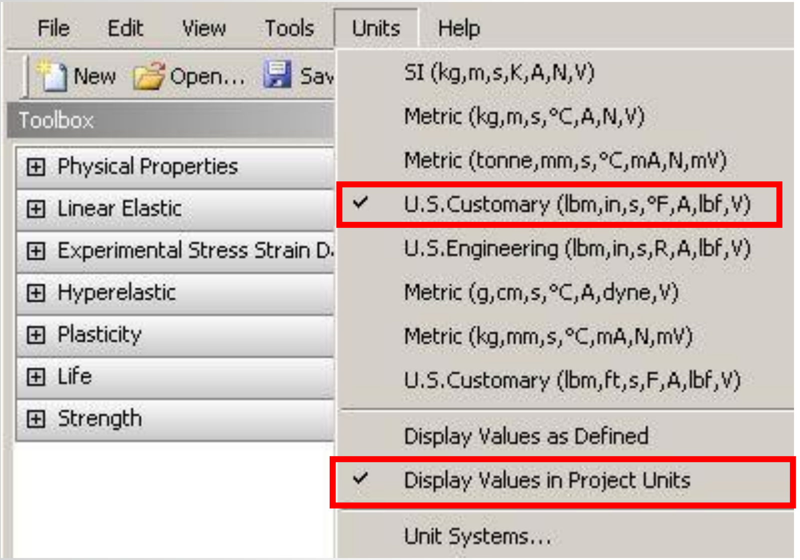
"Drop Target"

- **The "drop target" from the previous page indicates the outcome of the drag and drop operation. Cells A2 thru A4 from system (A) are shared by system (B). Similarly the solution cell A6 is transferred to the system B setup. In fact, the structural solution drives the buckling analysis.**

Project Schematic

Verify that the Project units are set to “US Customary (lbm, in, s, F, A, lbf, V).

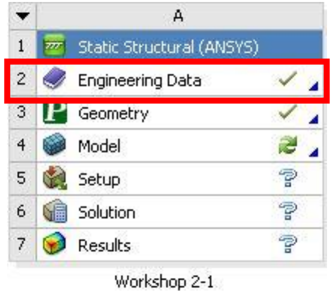
Verify units are set to “Display Values in Project Units”.



. . . Project Schematic

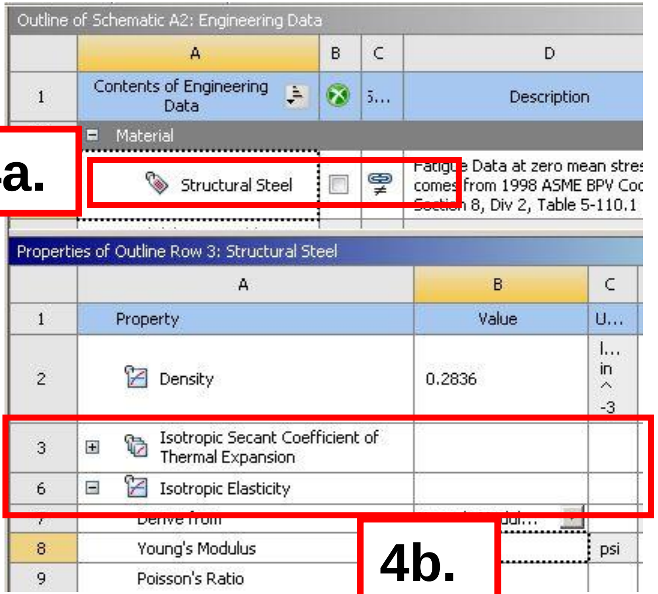
3. From the static structural system (A), double click the Engineering Data cell.

3.



4. To match the hand calculations referenced earlier, change the Young's modulus of the structural steel.
   a. Highlight Structural Steel.
   b. Expand "Isotropic Elasticity" and modify Young's Modulus to 3.0E7 psi.
   c. "Return to Project"

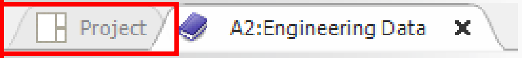
4a.



4b.

Note : changing this property here does not affect the stored value for Structural Steel in the General Material library. To save a material for future use we would "Export" the properties as a new material to the material library.

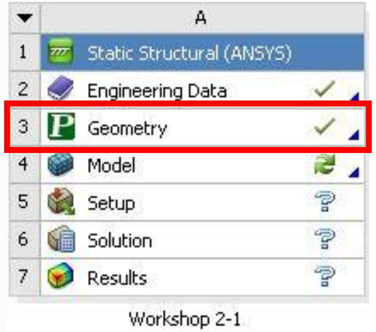
4c.

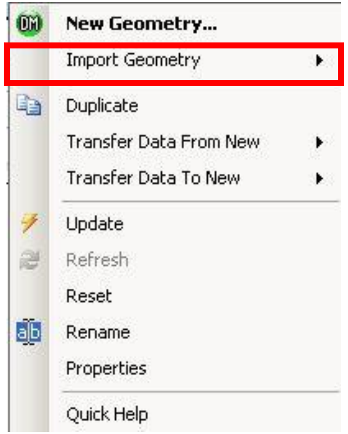


. . . Project Schematic

5. From the static structural system (A), RMB the Geometry cell and "Import Geometry". Browse to the file "Pipe.stp".

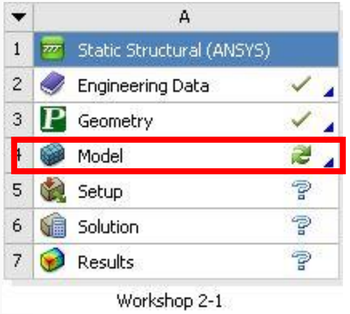
5.

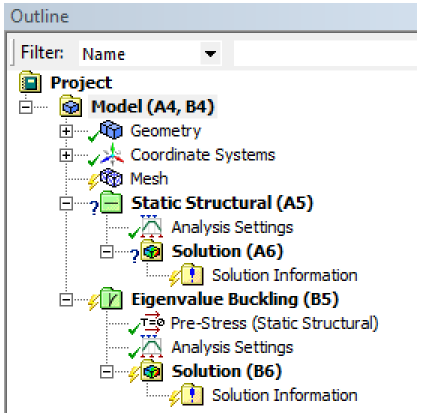




6. Double click the Model cell to start Mechanical.

6.





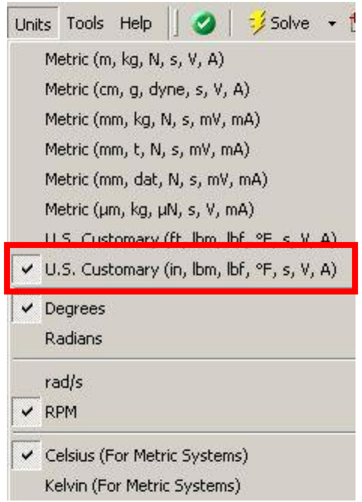
- When the Mechanical application opens the tree will reflect the setup from the project schematic.

Preprocessing

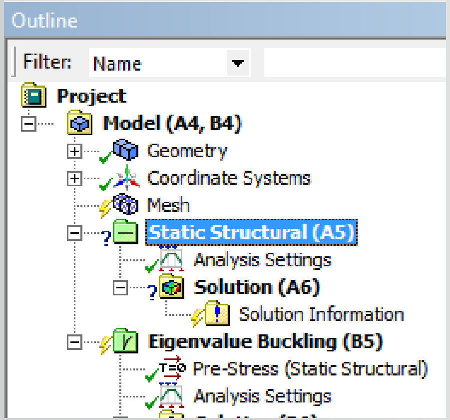
7. Set the working unit system to the U.S. customary system:
   a. U.S. Customary (in, lbm, psi, °F, s, V, A).

8. Apply constraints to the pipe:
   a. Highlight the Static Structural branch (A5).
   b. Select the surface on one end of the pipe.
   c. “RMB > Insert > Fixed Support”.

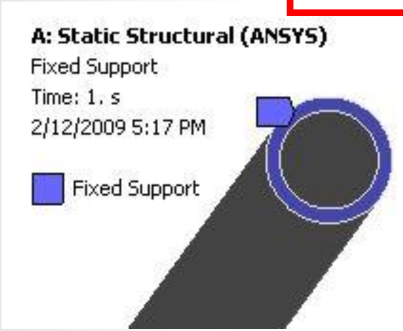
7a.



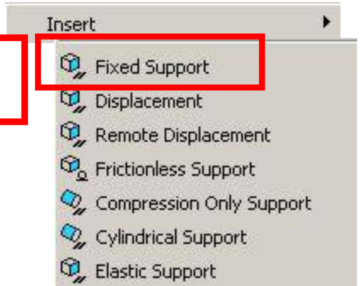
8a.



8b.

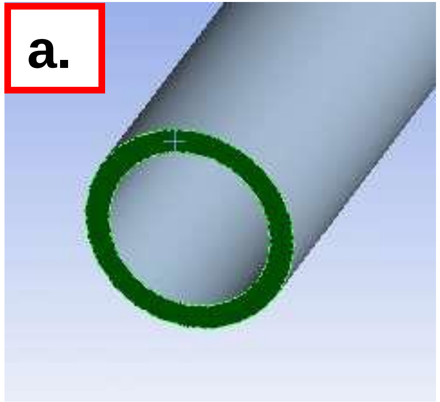


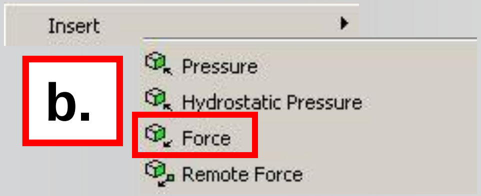
8c.

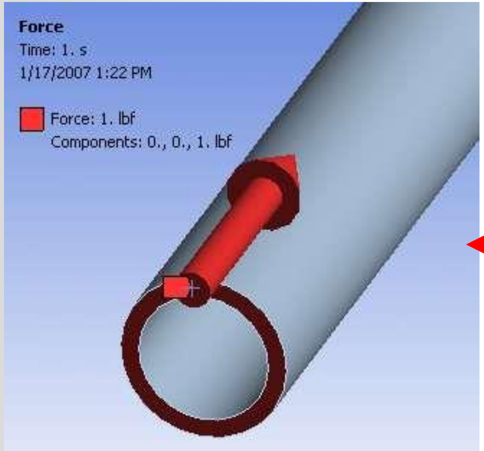


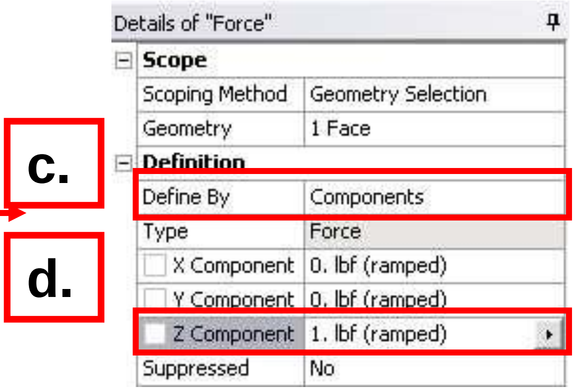
Environment

9. **Add buckling loads:**
   a. Select the surface on the opposite end of the pipe from the fixed support.
   b. "RMB > Insert > Force".
   c. In the force detail change the "Define by" field to "Components".
   d. In the force detail enter "1" in the "Magnitude" field for the "Z Component" (or use -1 depending on which ends of the pipe are selected).



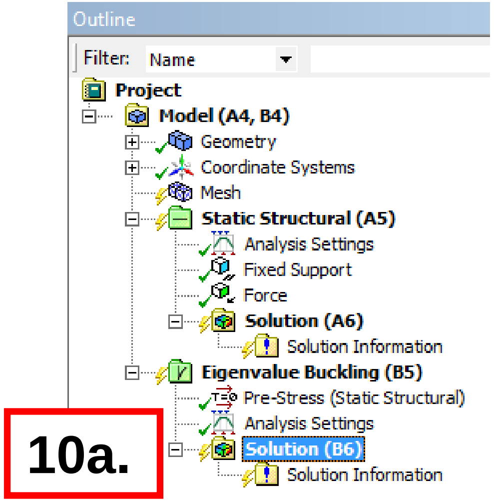




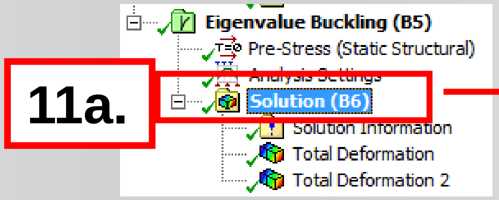


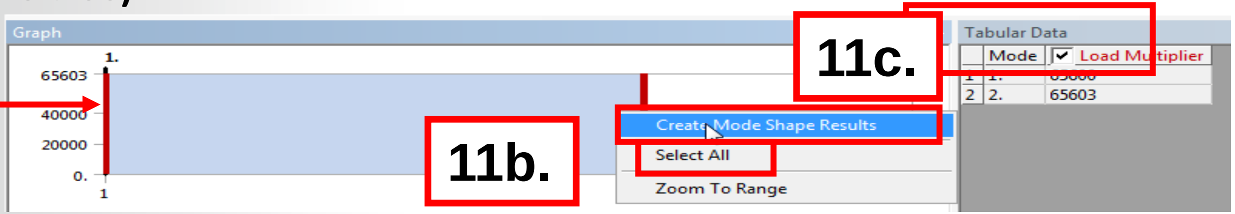
. . . Environment

10. **Solve the model:**
    a. Highlight the Solution branch for the Eigenvalue Buckling analysis (B6) and Solve.
       – Note, this will automatically trigger a solve for the static structural analysis above it.



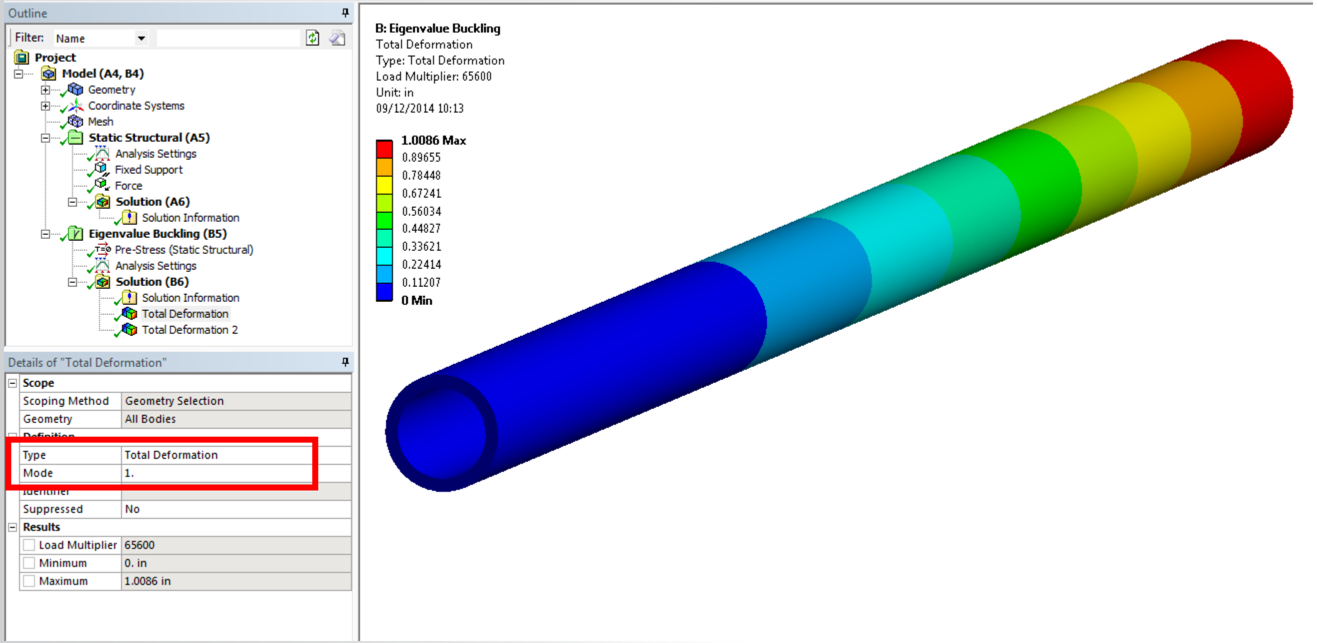
11. **When the solution completes:**
    a. Highlight the buckling "Solution" branch (B6).
       – The Timeline graph and the Tabular Data will display the first two buckling modes (more modes can be requested).
    b. RMB in the Timeline and choose "Select All".
    c. RMB > "Create Mode Shape Results" (this will add two "Total Deformation" branchs to the tree).





Results

– Click “Solve” to view the modes

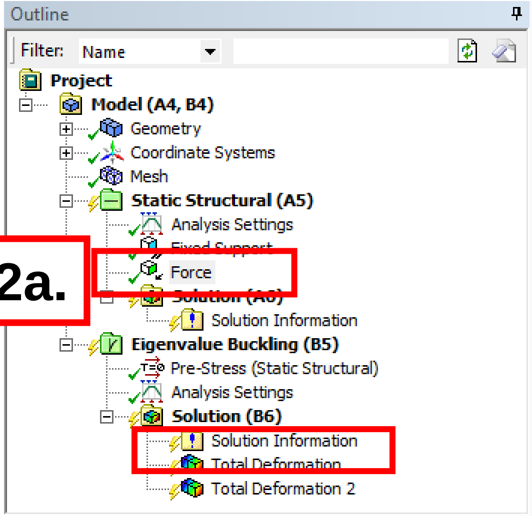


Recall that we applied a unit (1) force thus the result compares well with our closed form calculation of 65648 lbf.

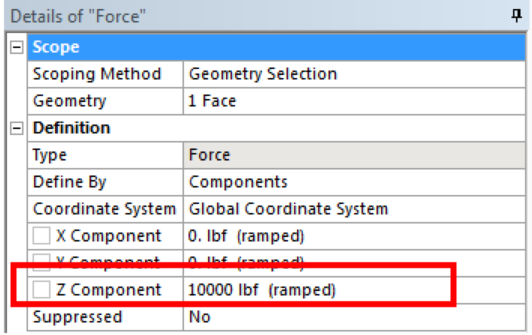
. . . Results

12. Change the force value to the expected load (10000 lbf):
    a. Highlight the “Force” under the “Static Structural (A5)” branch
    b. In the details, change the “Z Component” of the force to 10000 (or use -10000 depending on your selections).

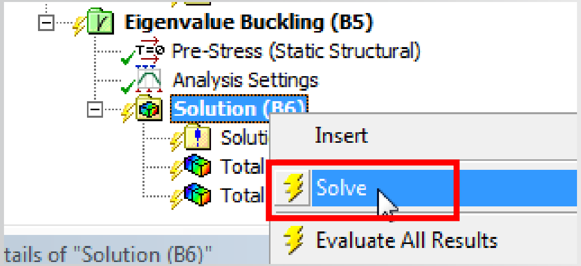
13. Solve:
    a. Highlight the Eigenvalue Buckling Solution branch (B6), RMB and “Solve”.



12a.



12b.



13a.

. . . Results

When the solution completes note the "Load Multiplier" field now shows a value of 6.56 for the first mode. Since we now have a "real world" load applied, the load multiplier is interpreted as the buckling factor of safety for the applied load.

Details of "Total Deformation"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
Mode	1.
Identifier	
Suppressed	No
Results	
Load Multiplier	6.56
Minimum	0. in
Maximum	1.0085 in

Given that we have already calculated a buckling load of 65600 lbf, the result is obviously trivial (65600 / 10000). It is shown here only for completeness.

Verification

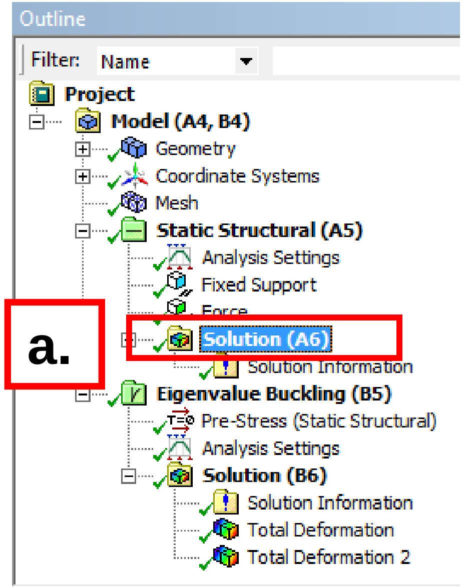
A final step in the buckling analysis is added here as a “best practices” exercise.

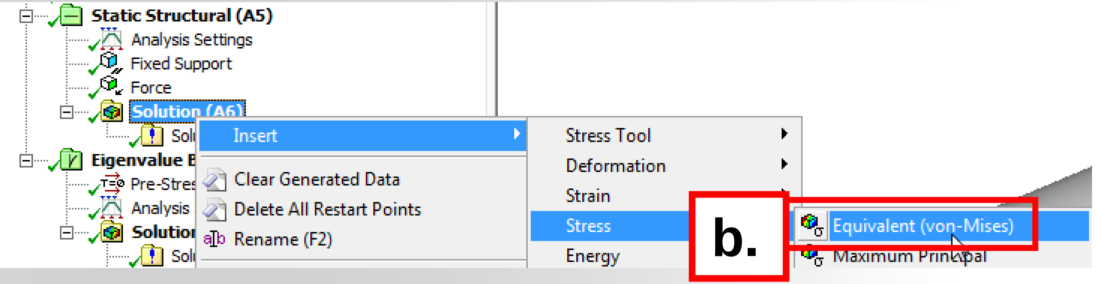
We have already predicted the expected buckling load and calculated the factor of safety for our expected load. The results so far ONLY indicate results as they relate to buckling failure. To this point we can say nothing about how our expected load will affect the stresses and deflections in the structure.

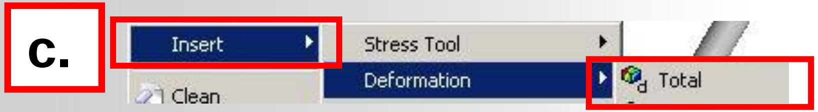
As a final check we will verify that the expected load (10000 lbf) will not cause excessive stresses or deflections before it is reached.

. . . Verification

14. **Review Stresses for 10,000lbf load:**
    a. Highlight the “Solution” branch under the “Static Structural” environment (A6).
    b. RMB > Insert > Stress > Equivalent Von Mises Stress.
    c. RMB > Insert > Deformation > Total.
    d. Solve.







. . . Verification

A quick check of the stress results shows the model as loaded is well within the mechanical limits of the material being used (Engineering Data shows compressive yield = 36,259 psi).

As stated, this is not a required step in a buckling analysis but should be regarded as good engineering practice.