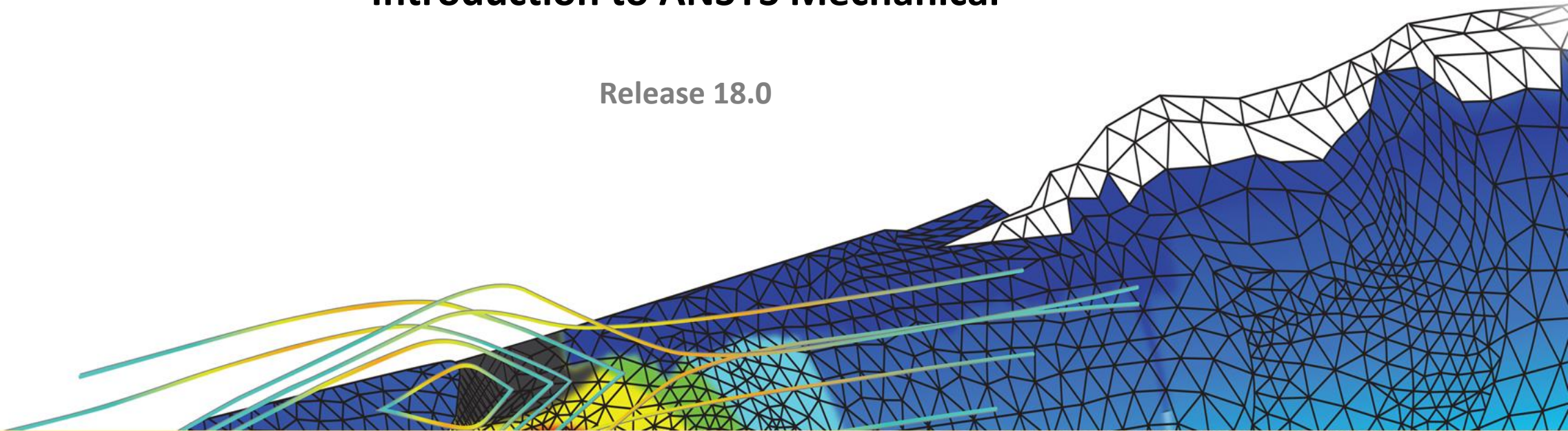




Workshop 08.1: Eigenvalue Buckling with Linear Pre-Stress

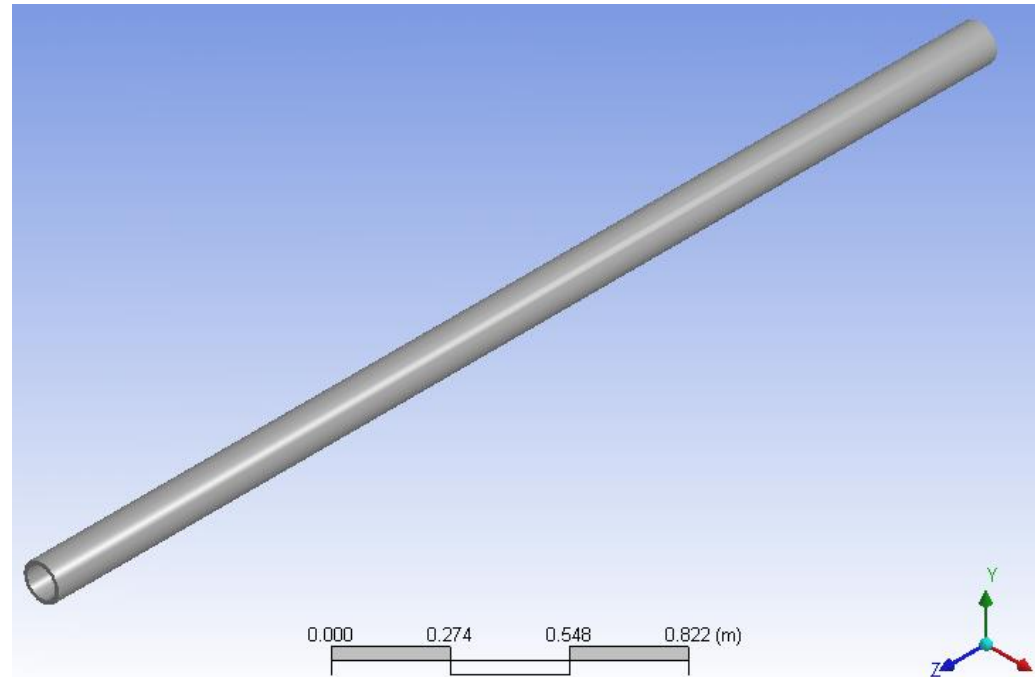
Introduction to ANSYS Mechanical

Release 18.0



Goals

Our primary goal in this workshop is to predict the eigenvalue buckling load for the pipe geometry shown below and to compare the analytical results to closed-form calculations from an engineering handbook. In addition, we will apply an expected compressive load of 10,000 lbf and determine the factor of safety with respect to buckling. Finally, we will verify that the structure's material will not fail before buckling occurs.



Assumptions

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

$$\text{OD} = 4.5 \text{ in, ID} = 3.5 \text{ in, } E = 30 \times 10^6 \text{ psi, } I = 12.771 \text{ in}^4, L = 120 \text{ in}$$

We will assume that the pipe buckling behavior conforms to the following handbook formula, where P' is the critical (primary buckling) load:

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

For the case of a fixed-free beam, parameter $K = 0.25$.

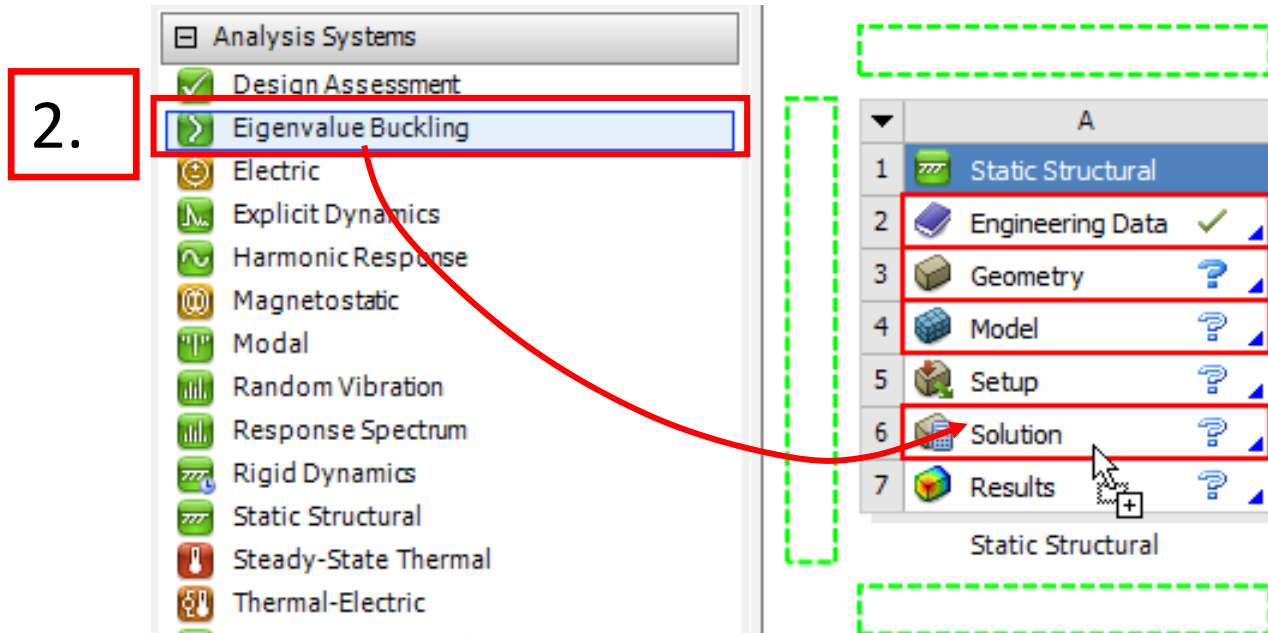
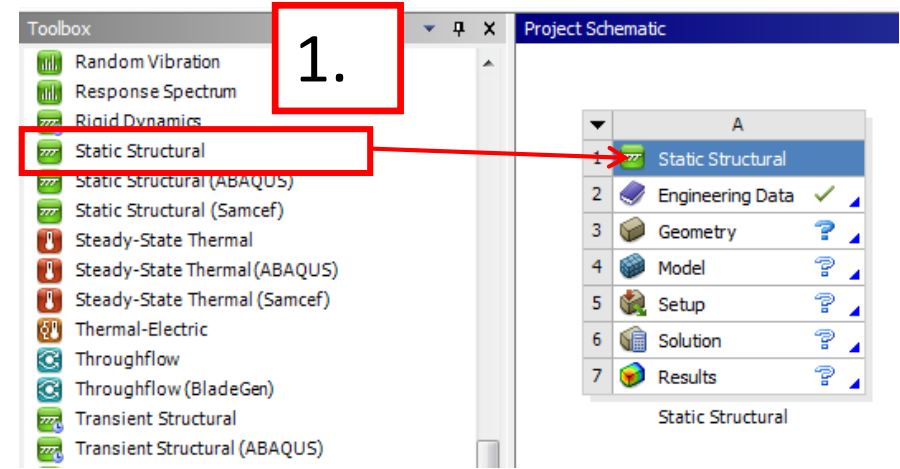
Assumptions

Using the formula and data from the previous page, we calculate the buckling load as:

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

Project Schematic

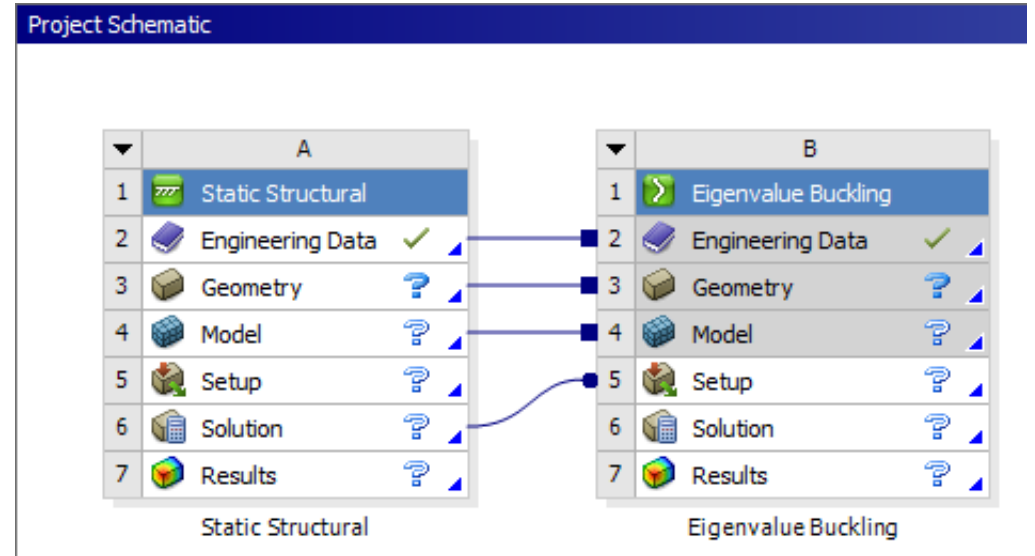
1. From the Toolbox, double-click Static Structural to create a new analysis system.
2. Drag and drop an “Eigenvalue Buckling” analysis system from the Toolbox onto the Solution cell of the Static Structural system.



Project Schematic

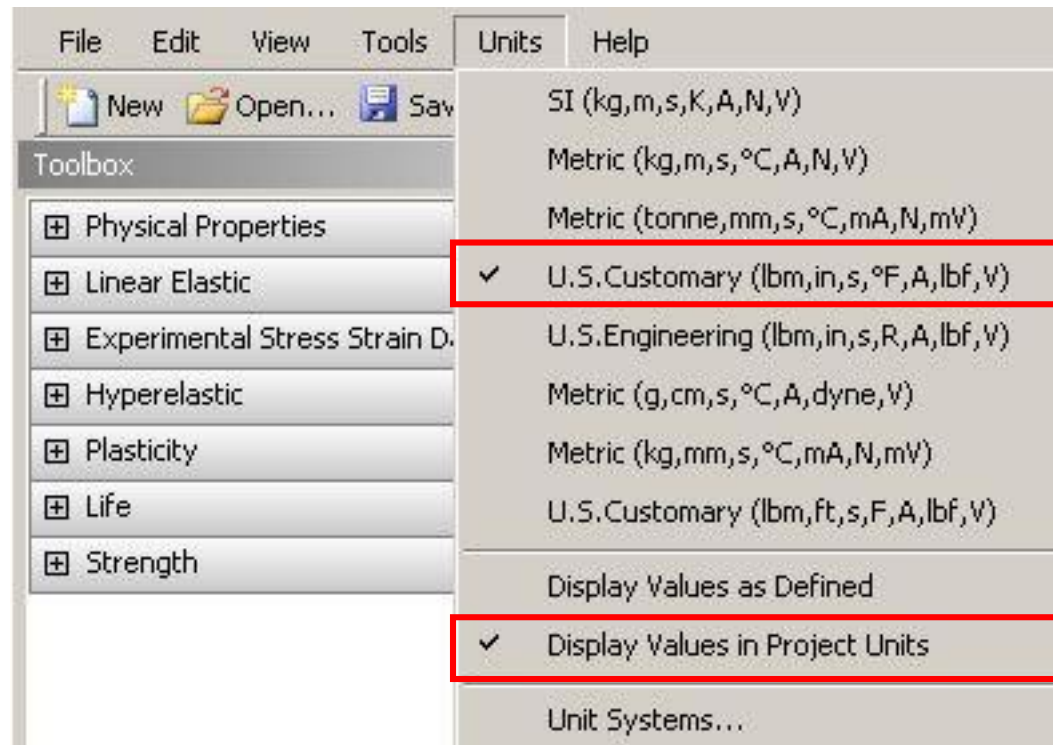
When the schematic is set up correctly, it should appear as shown here.

Notice that the “drop target” in the Project Schematic previewed the outcome of the drag-and-drop operation. Cells A2 through A4 in System A are shared by the corresponding cells in System B. Similarly, data from Solution Cell A6 is transferred to Setup Cell B5.



Project Schematic

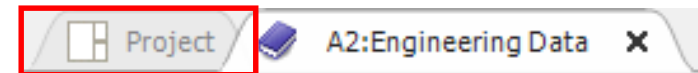
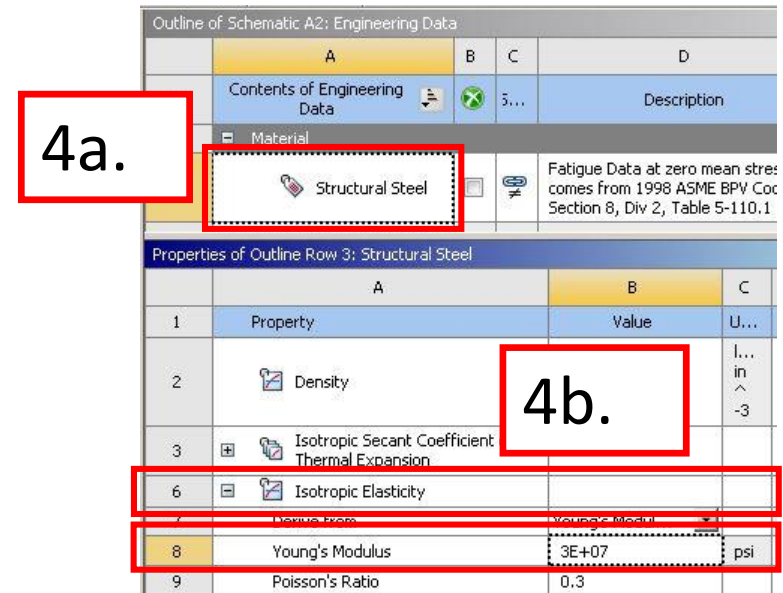
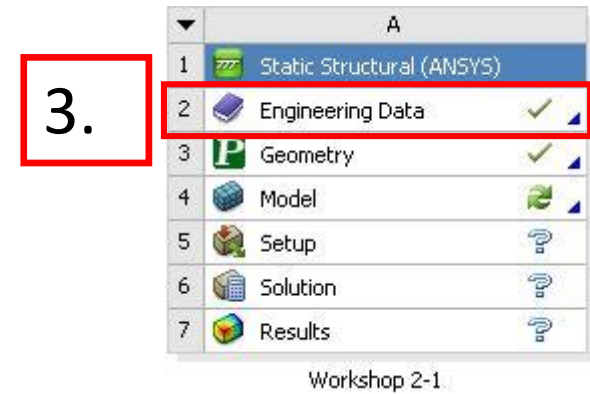
Verify that the Project units are set to “US Customary (lbm, in, s, °F, A, lbf, V)” and that “Display Values in Project Units” is checked.



Project Schematic

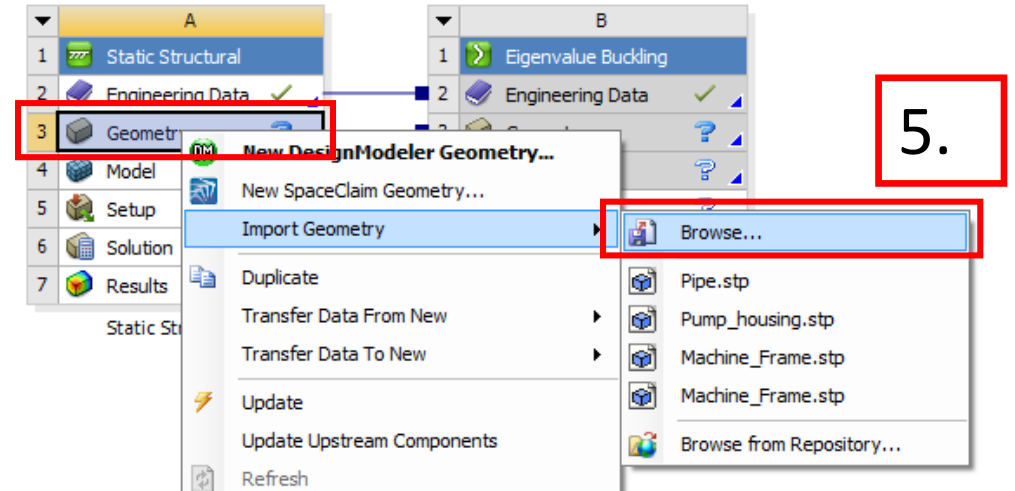
3. From the Static Structural analysis system, double-click the Engineering Data cell.
4. To match the hand calculations referenced above, change the value of Young's modulus for Structural Steel.
 - a. Select Structural Steel.
 - b. In the Properties view, expand "Isotropic Elasticity" and set Young's Modulus to "30.0E6 psi."
 - c. Return to the Project Schematic view.

Note: Changing a property here does not affect the stored value for Structural Steel in the General Materials library.

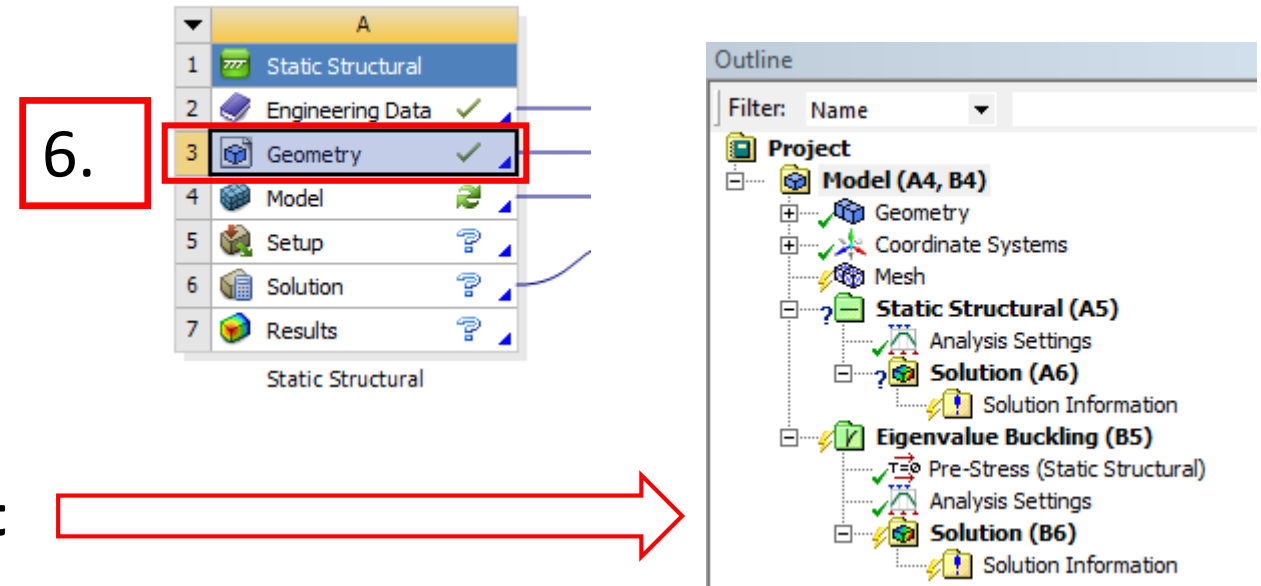


Project Schematic

5. From the Static Structural system, select the Geometry cell, then RMB > Import Geometry > Browse... > file "Pipe.stp" > OK.



6. Double-click the Model cell to start Mechanical.

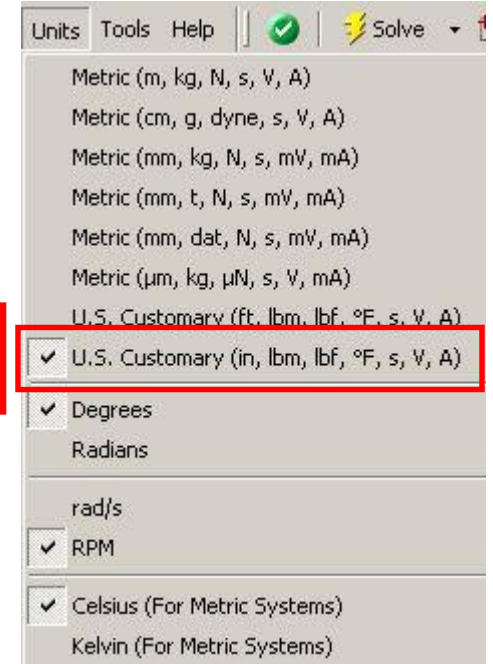


Note: When Mechanical opens, the tree will reflect the analysis setup in the Project Schematic.

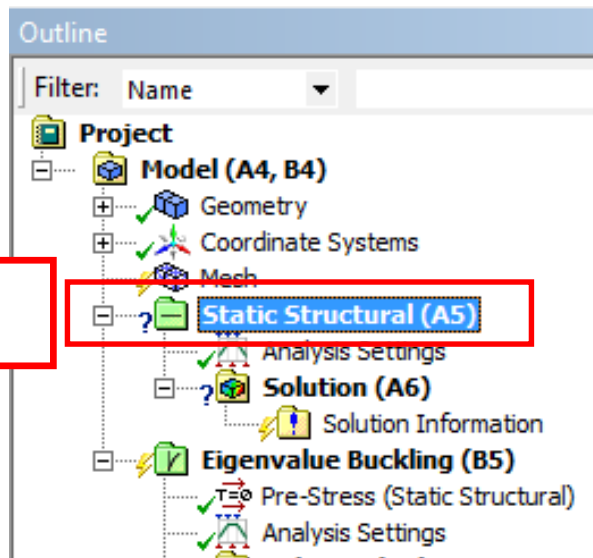
Preprocessing

7. Set the working unit system to “U.S. Customary (in, lbm, lbf, °F, s, V, A).”
8. Apply constraints to the pipe:
 - a. Highlight the Static Structural environment branch.
 - b. Select the surface of one end of the pipe.
 - c. RMB > Insert > Fixed Support.

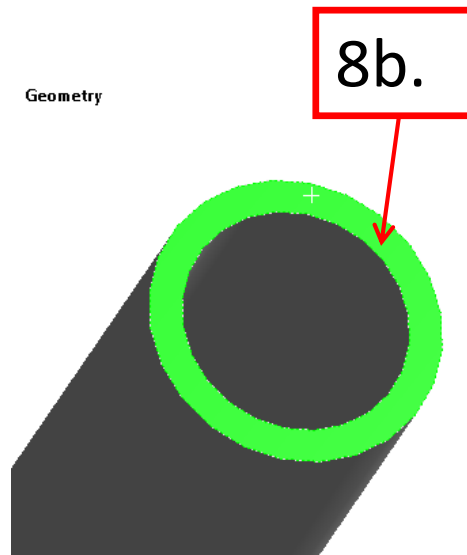
7.



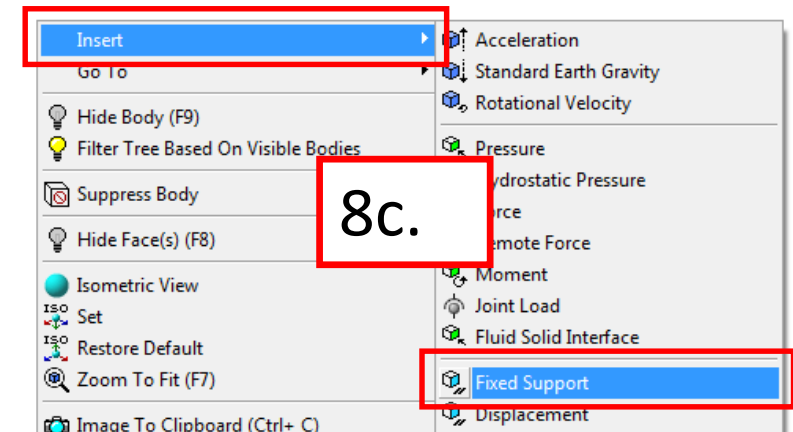
8a.



8b.



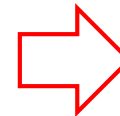
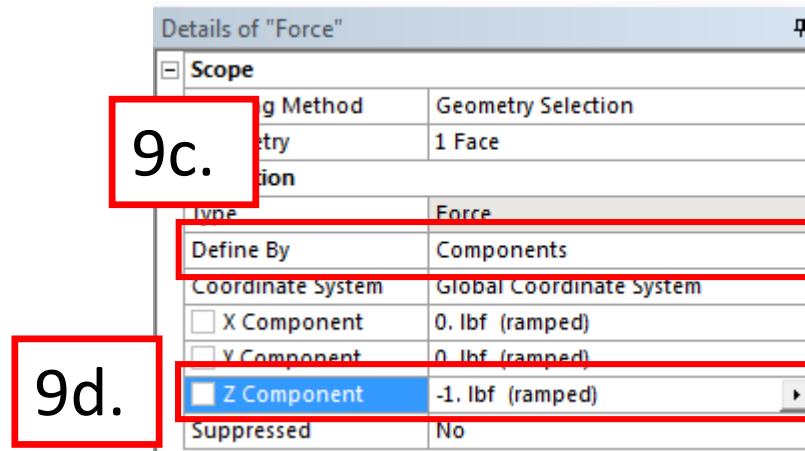
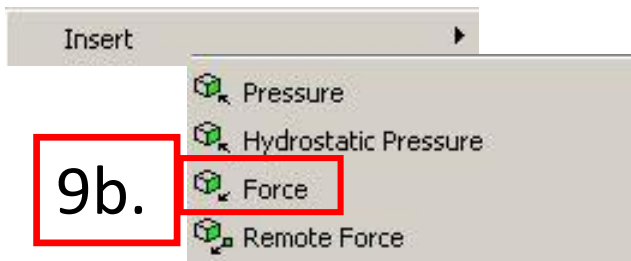
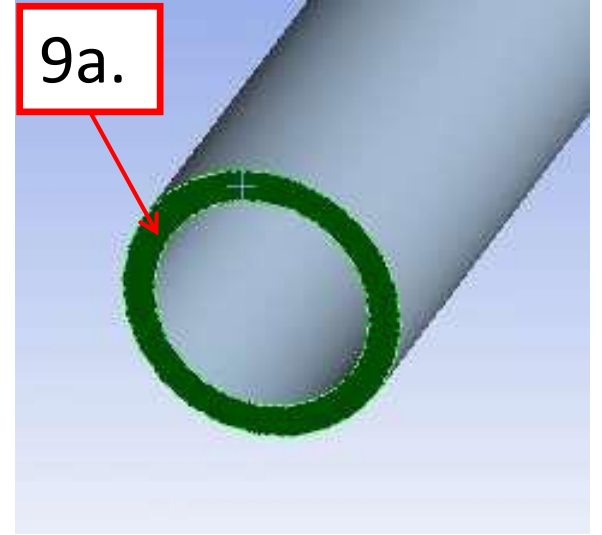
8c.




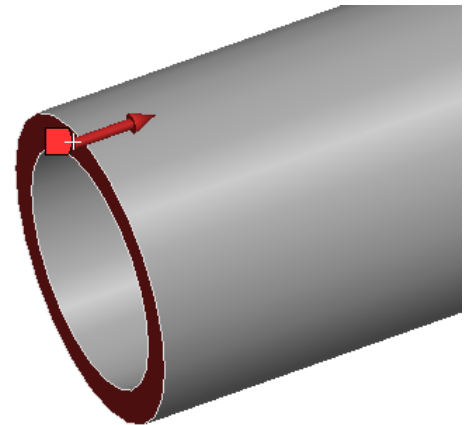
Environment

9. Add buckling loads:

- Select the surface on the opposite end of the pipe from the fixed support.
- RMB > Insert > Force.
- In the force Details view, set Define By to “Components.”
- In the force Details view, set Z Component to “1 lbf” or “-1 lbf.”
Note that the applied force should be compressive. The sign of the applied force will depend upon which end of the pipe was selected when the load was defined.



A: Static Structural
Force
Time: 1. s
 Force: 1. lbf
Components: 0., 0., -1. lbf

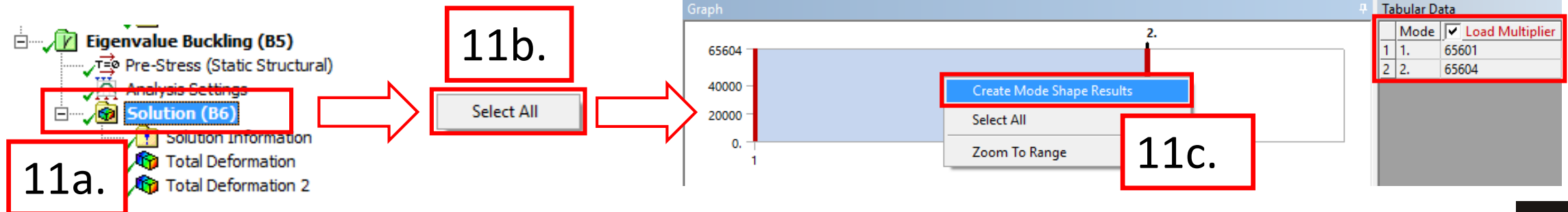
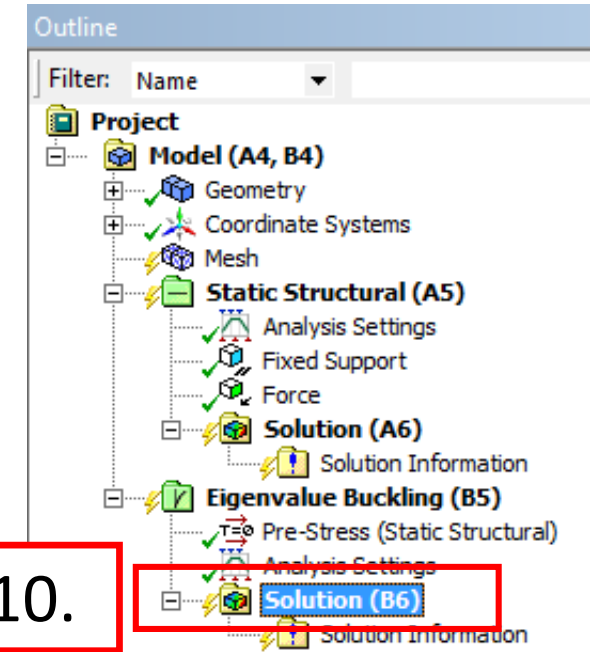


Postprocessing

10. Select the Solution branch for the Eigenvalue Buckling environment and click Solve. This will automatically perform solution of Static Structural environment above it.

11. When the solution completes:

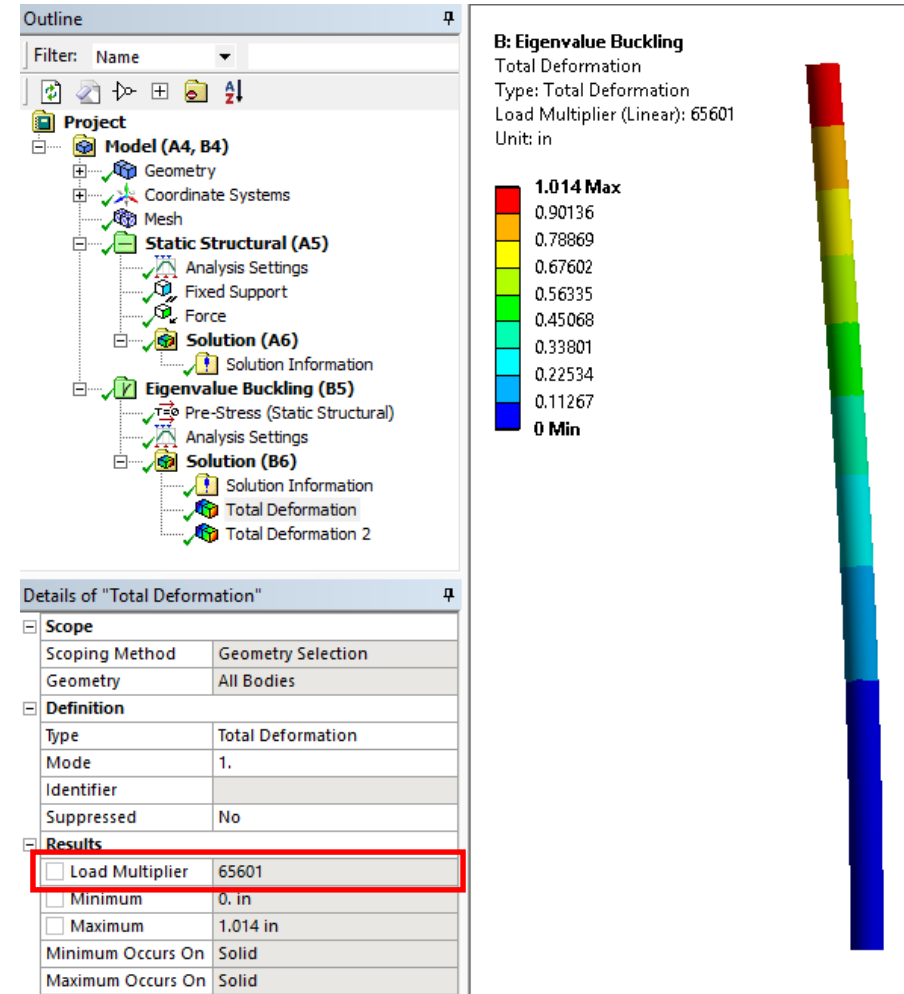
- Select the Solution branch in the Eigenvalue Buckling environment. The Timeline Graph and Tabular Data views will display the first two buckling modes (more can be requested).
- Point at the Graph view and RMB > Select All.
- Point at the Graph view and RMB > Create Mode Shape Results (this will add two Total Deformation branches to the tree).



Postprocessing

Click “Solve” to generate the mode shape results.

Recall that we applied a unit force. Thus, the predicted buckling load of 65,601 lbf compares well with our closed-form calculation of 65,648 lbf.

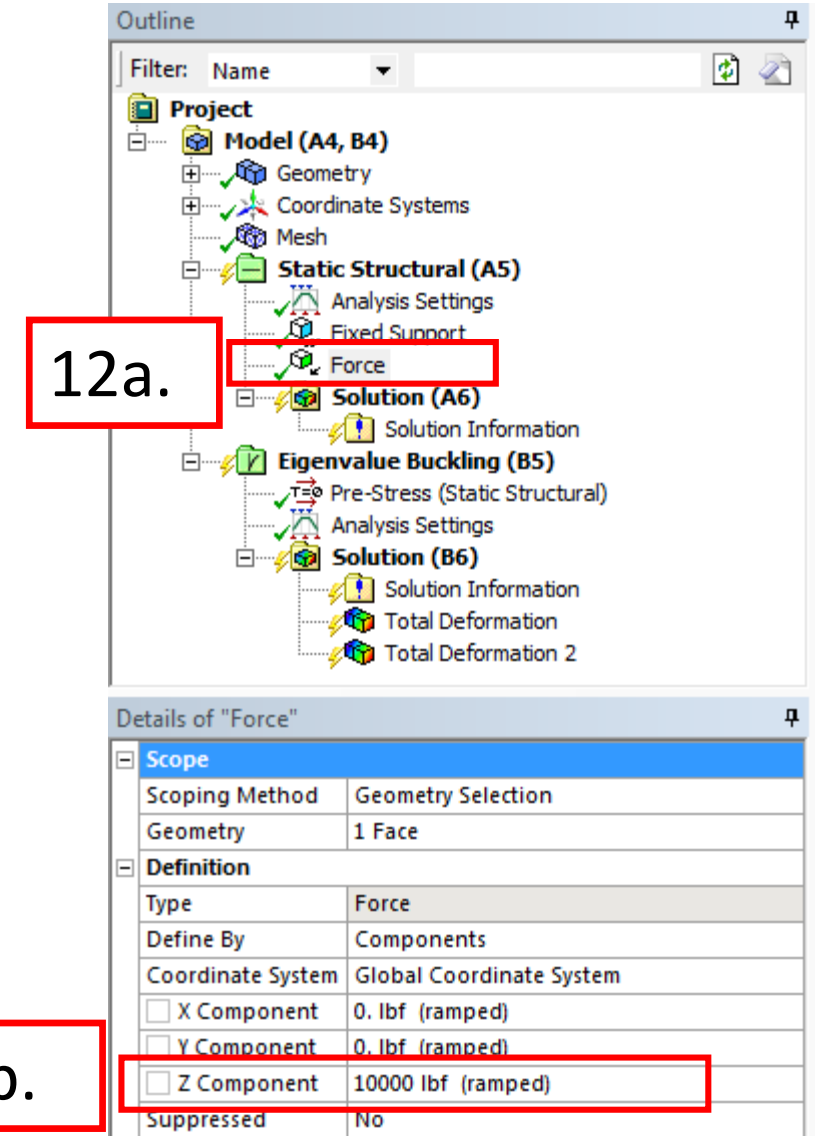
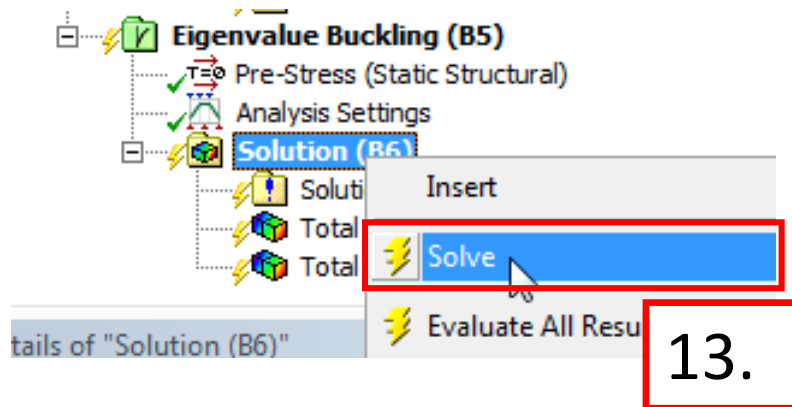


Postprocessing

12. Change the value of the applied force to the expected column load (10,000 lbf):

- Select Force under the Static Structural branch.
- In the Details view, set Z Component to "10,000 lbf" or "-10,000 lbf," again depending upon which end of the pipe was selected when the load was defined.

13. Select the Solution branch for the Eigenvalue Buckling environment and click Solve.



Postprocessing

When the solution completes, note that the Load Multiplier field now shows a value of 6.5601 for the first mode. Since we now have a “real world” load applied, the load multiplier may be interpreted as the factor of safety with respect to buckling 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	
<input type="checkbox"/> Load Multiplier	6.5601
<input type="checkbox"/> Minimum	0. in
<input type="checkbox"/> Maximum	1.0097 in
Minimum Occurs On	Solid
Maximum Occurs On	Solid

Given that we have already calculated the buckling load to be 65,601 lbf, the result is trivial and could have been easily predicted ($6.5601 = 65,601/10,000$).

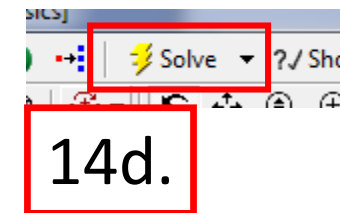
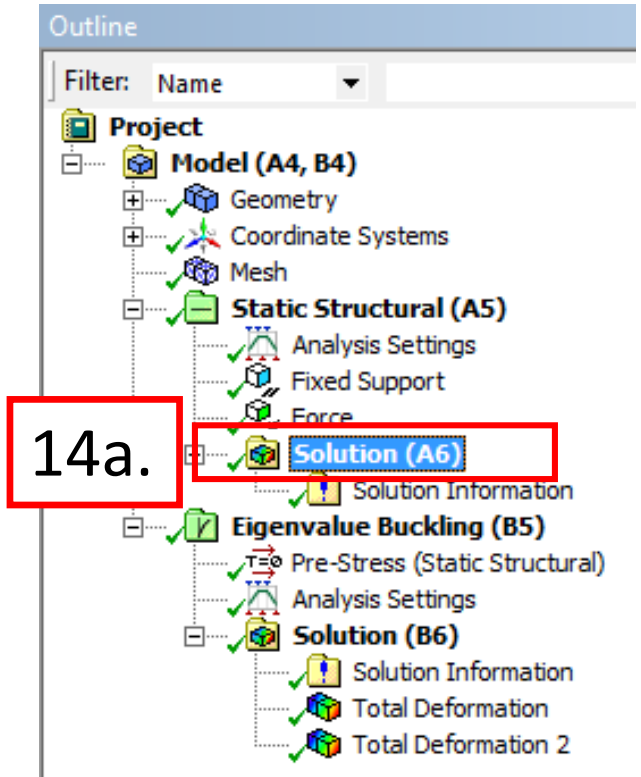
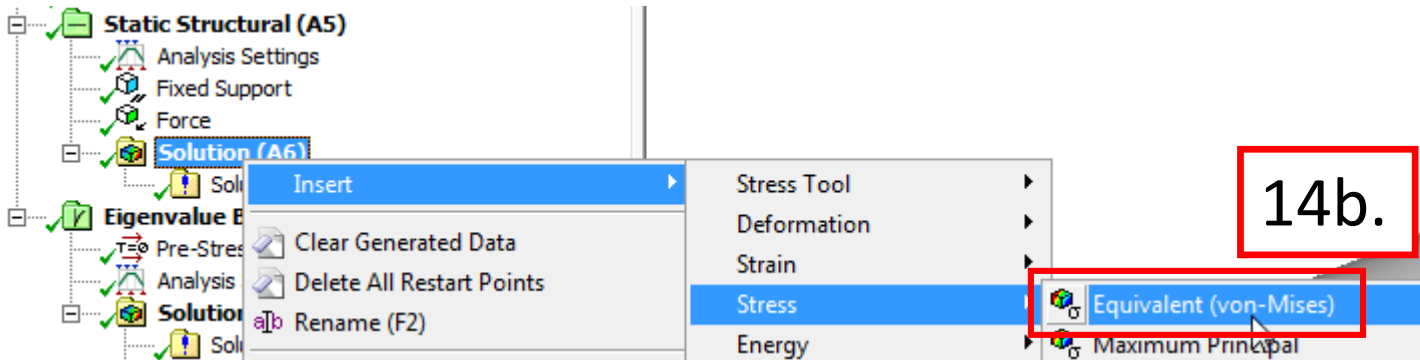
Verification

Our final step in the buckling analysis is to verify that the pipe material will not fail by yielding. The results that we have obtained up to this point say nothing about how our expected load will affect the stresses and deflections in the structure. As a verification step, we will verify that the expected load of 10,000 lbf will not cause excessive stresses or deflections.

Verification

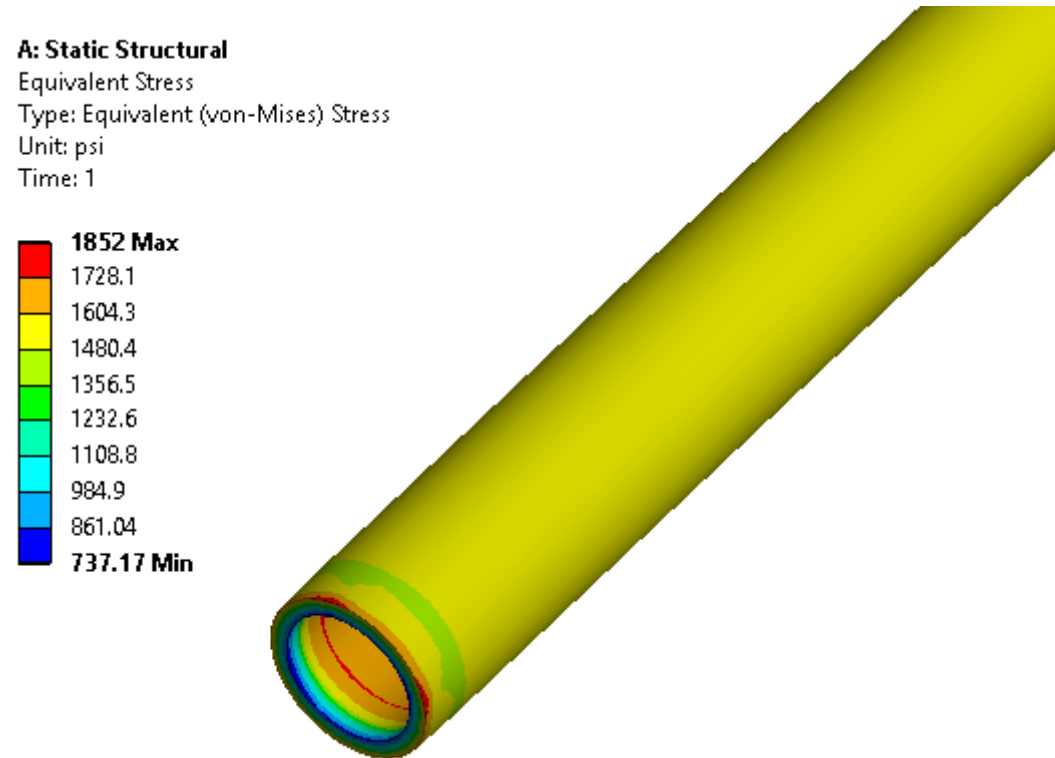
14. Review stresses for the 10,000-lbf load:

- Highlight the Solution branch under the Static Structural environment branch.
- RMB > Insert > Stress > Equivalent (von-Mises).
- RMB > Insert > Deformation > Total.
- Solve.



Verification

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





Workshop 08.1: Eigenvalue Buckling with Linear Pre-Stress

END

