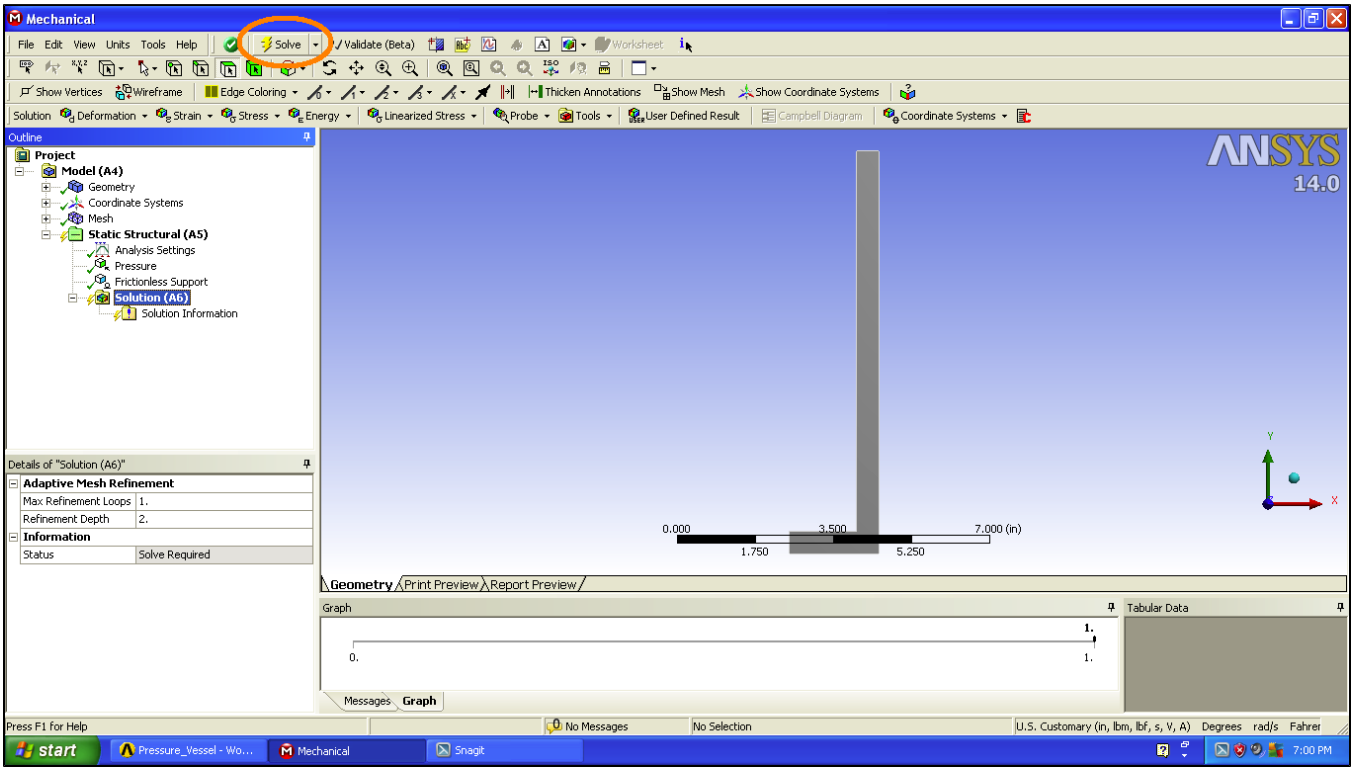
Pressure Vessel - Numerical Solution

Authors: Sebastien Lachance-Barrett and Vincent Prantil

Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

Numerical Solution

To obtain the numerical solution, click solve. ANSYS formed the stiffness matrix for each element, assembled the global stiffness matrix and inverted it to get the nodal displacements. This is the bulk of the computation that ANSYS performs. All the results that we will look at next such as the deformed shape and the stresses are derived from these nodal displacements.



Go to Step 6: Numerical Results

Go to all ANSYS Learning Modules