3D Signpost - Numerical Solution

Authors: Rajesh Bhaskaran and Vincent Prantil

Numerical Solution

To obtain the numerical solution, click Solve.
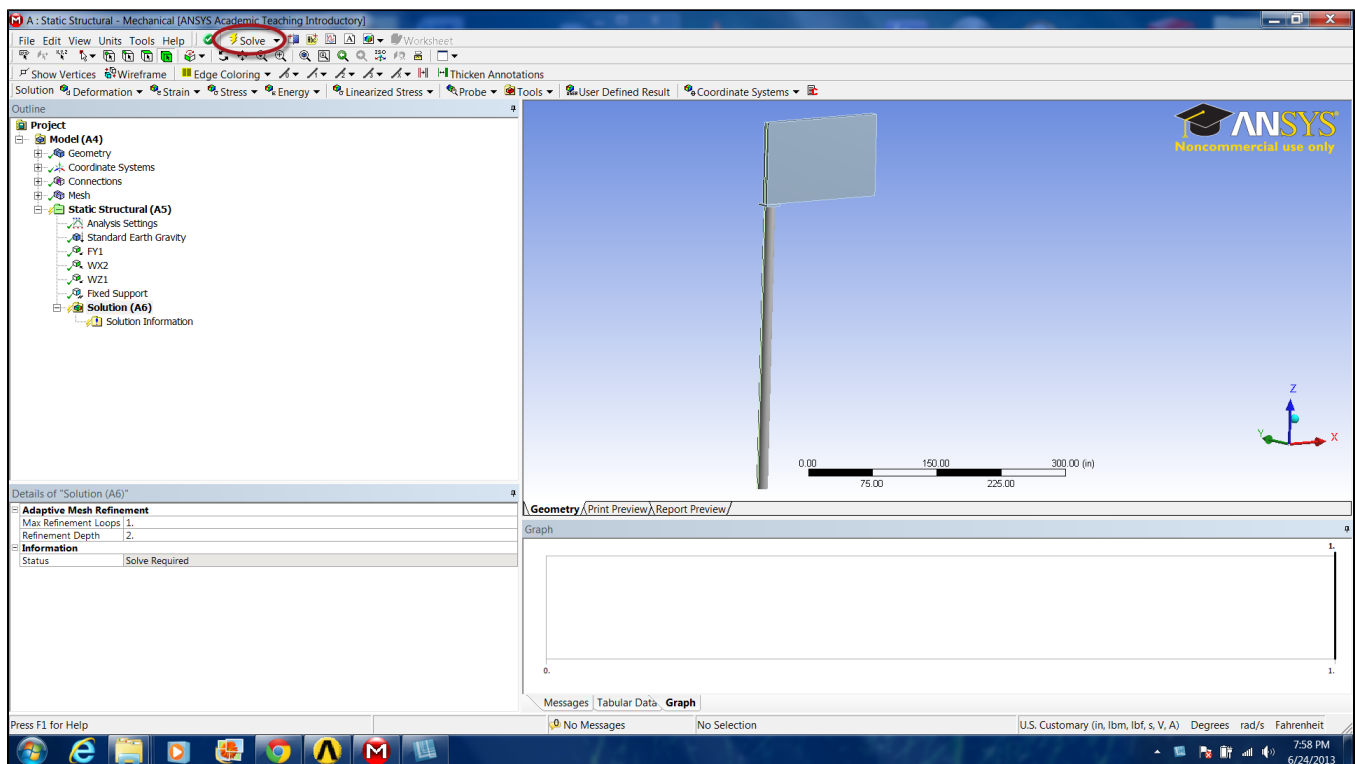
ANSYS created the stiffness matrix for each element, assembled the global stiffness matrix, then inverted the matrix to get the nodal displacements. This is the bulk of the computation that ANSYS performs. The following results are derived from these nodal displacements.

Go to Step 6: Numerical Results

Go to all ANSYS Learning Modules