2D Beam - Physics Setup

Authors: Rajesh Bhaskaran and Vincent Prantil

Physics Setup

The following video shows how to specify the physics of the problem: plane stress approximation, material properties (Young's modulus and Poisson ratio) and boundary conditions. These settings get fed into the element formulation when obtaining the numerical solution later.

Summary of steps in the above video:

1. In ANSYS Mechanical, double-check units, behavior, thickness value, and material assignment:
    a. Select Units > U.S. Customary (in, lbm, lbf, F, s, V, A).
    b. In Project Tree, highlight Geometry. Under Details of ‘Geometry’, verify 2D Behavior is set to Plane Stress.
    c. In Project Tree, expand Geometry. Highlight Surface Body. Under Details of ‘Surface Body’, verify Thickness is set to 3in in accordance with problem statement.
    d. Verify Material > Assignment > Structural Steel in accordance with problem statement.
2. To apply displacement constraints:
    a. In Project Tree, highlight Static Structural.
    b. In toolbar, select Supports > Fixed Support.
    c. Using Vertex Selection Tool, hold down CTRL button and select bottom two vertices on rectangle. Click Geometry > Apply.
3. To apply loads:
    a. In Project Tree, highlight Static Structural.
    b. In toolbar, select Loads > Force.
    c. Using Vertex Selection Tool, select point of load application on rectangle. Click Geometry > Apply.
    d. Click Define By > Components. Change Y Component value to -10000
4. Save Project.

Go to Step 5: Numerical Solution

Go to all ANSYS Learning Modules