2D Beam - Numerical Results

Authors: Rajesh Bhaskaran and Vincent Prantil

Numerical Results

The following video shows how to plot the deformed shape and use it to check if the boundary conditions (displacement constraints and point load) have been applied correctly.

Summary of steps in the above video:

1. To view the shape of deformation:
   a. In ANSYS Mechanical, highlight Solution in Outline Tree.
   b. Select Deformation > Total.
   c. Click Solve.
2. To view and compare the undeformed and deformed shapes of the beam:
   a. Select Edges > Show Undeformed WireFrame.
3. To check the behavior of nodes at our displacement constraint corner locations:
   a. Draw a rectangle around each constrained corner using the zoom tool and the right mouse button. After attaining a closer view, confirm that the constrained corners have not moved.
4. To find the magnitude of the max deformation:
   a. The max deformation magnitude will be shown in the colorbar in the graphics window.
   b. Select Maximum. A tag will appear in the graphics window at the location of maximum deformation.
5. To view the unscaled deformation of the beam:
   a. Click on the dropdown menu next to Result in the Toolbar that says: 2.8e+002 (Auto Scale)
   b. Select 1.0 (True Scale)
6. To save the image to a .png image file:
   a. Select New Figure or Image > Image to File …
   b. Browse to a desired folder location and save the image to your liking.
7. To animate the displacement:
   a. Press the Play button in the Graph window.
8. Save Project.

As we saw in the previous video, the maximum deformation is 0.0178". From this, we can calculate the normalized maximum deformation which, if you recall, is defined as follows:

$$\frac{v_{max}}{P\,L^3/(E\,I)}$$

The normalized maximum deformation comes out to be 0.0066. This result is compared with various alternative analysis methods in 2D, 1D, and 3D in the following figure. This is figure 4.12 in *Lying by Approximation: The Truth About Finite Element Analysis* by Prantil, Papadopoulos and Gessler. The alternative analysis methods are described in the book.
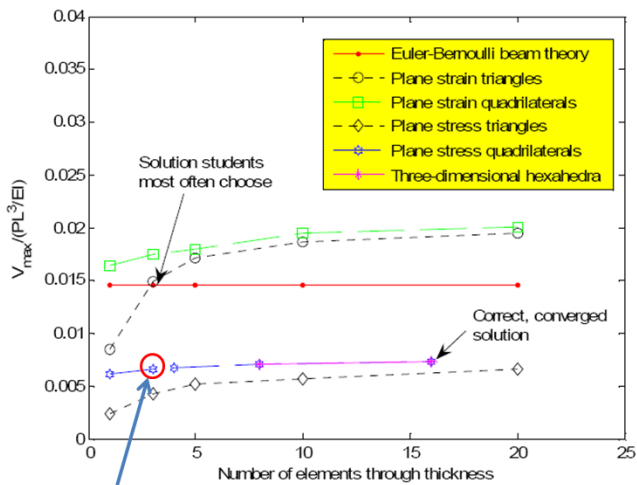
2D Plane stress solution

Go to Step 7: Verification & Validation

Go to all ANSYS Learning Modules