

2D Beam - Numerical Solution

Authors: Rajesh Bhaskaran 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

The following video shows how to obtain the numerical solution where the ANSYS solver will form the stiffness matrix for each element, assemble the global stiffness matrix and invert it to get the nodal displacements.

Summary of steps in the above video:

1. In ANSYS Mechanical, highlight Solution in Outline Tree. Click Solve.
2. To see what element type ANSYS used:
 - a. Highlight Solution Information in Outline Tree.
 - b. Click anywhere under Solver Output.
 - c. Press Ctrl+F. In the search box, type: element type. You can see that the element type used is PLANE182.
3. To see more information on PLANE182
 - a. Select Help > Mechanical Help.
 - b. Click on the Search tab. In the search box, type: plane182.
 - c. Hit Enter.
 - d. PLANE182 should appear in the list of pages found. If you click on the page, information shall appear in the window to the right.
4. When finished:
 - a. Dismiss the help window and search window.
 - b. Save Project.

[Go to Step 6: Numerical Results](#)

[Go to all ANSYS Learning Modules](#)