ANSYS - Tensile Bar (Results-Interpretation)

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Pre-Analysis & Start-Up
2. Numerical Results
3. Verification and Validation
Exercises
Comments

Tensile Bar (Results-Interpretation)

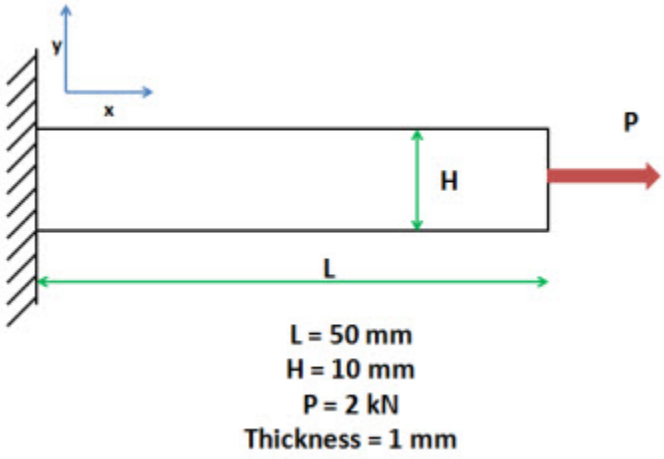
Created using ANSYS 13.0

Learning Goal

This ANSYS tutorial focuses on the *interpretation and verification* of FEA results (rather than on obtaining an FEA solution from scratch). The ANSYS solution files are provided as a download. We read the solution into *ANSYS Mechanical* and then move directly to reviewing the results critically. We are particularly interested in the comparison of FEA results with hand calculations. The objective is to help you learn the process of interpreting FEA results critically. There are other tutorials to help you learn the process of developing an ANSYS solution from scratch.

Problem Specification

A steel bar is mounted in a rigid wall and axially loaded at the end by a force P = 2 kN as shown in the figure below. The bar dimensions are indicated in the figure. The bar is so thin that there is no significant stress variation through the thickness. Neglect gravity.



The material properties are:

- Young's modulus *E* = 200 GPa
- Poisson ratio = 0.3

In this exercise, you are presented with the numerical solution to the above problem obtained from finite-element analysis (FEA) using ANSYS software. Compare FEA results for the stress distribution presented to you with the corresponding analytical solution. Justify agreements and discrepancies between the two approaches (FEA vs. Analytical).

Go to Step 1: Pre-Analysis & Start-Up

Go to all ANSYS Learning Modules