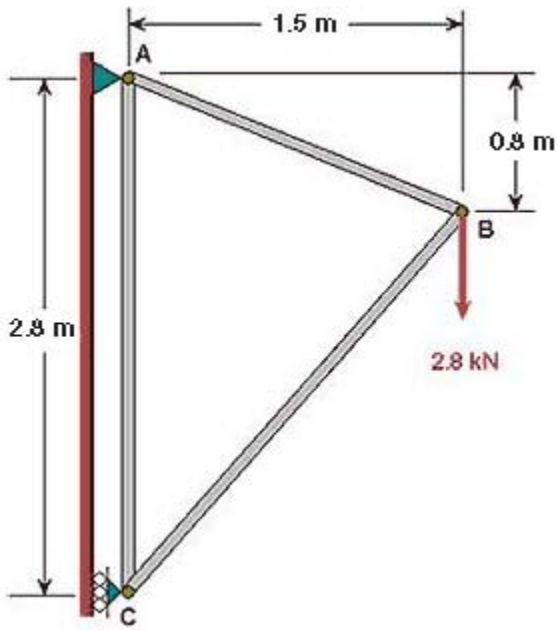
ANSYS - Truss Problem Specification

Author: Rajesh Bhaskaran, Cornell University

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results
Problem Set 1
Problem Set 2

Problem Specification

Determine the force in each member of the following truss using ANSYS 12.1. Indicate if the member is in tension or compression. The cross-sectional area of each member is 0.01 m, the Young's modulus is 200×10^9 N/m^2 and Poisson ratio is 0.3.



The solution in ANSYS for this and subsequent tutorials is divided into the nine steps listed in the grey area above. Note that you'll need to follow these same nine steps for solving almost any problem in ANSYS or a comparable finite-element analysis package.

Go to Step 1: Start-up and preliminary set-up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules