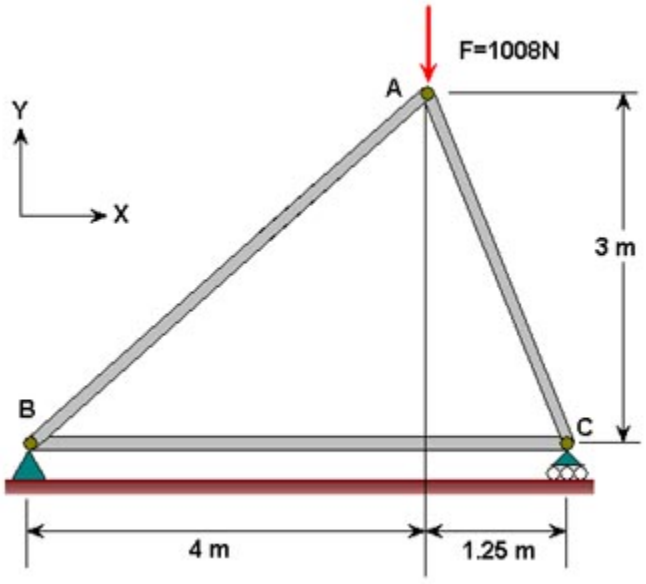
ANSYS - Truss Problem Set 2

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 Set 2

Determine the force in each member of the following truss using ANSYS. Indicate if the member is in tension or compression. Use the same LINK1 element as in the tutorial. The cross-sectional area of each member is 0.02 m^2, Young's modulus is $200x10^9$ N/m^2 and Poisson's ratio is 0.3. Verify your results by calculating the forces manually.



Results

Determine the following:

1. Listing of the reactions from the ANSYS solution.

2. Listing of the element forces from the ANSYS solution. Calculate and determine the forces in each member and whether the member is in tension or compression from this ANSYS result.

3. Using your pencil-and-paper calculations verifying the ANSYS results for the member and reaction forces.

See and rate the complete Learning Module

Go to all ANSYS Learning Modules