ANSYS - Cantilever Beam

Author: John Singleton, Cornell University

Cantilever Beam

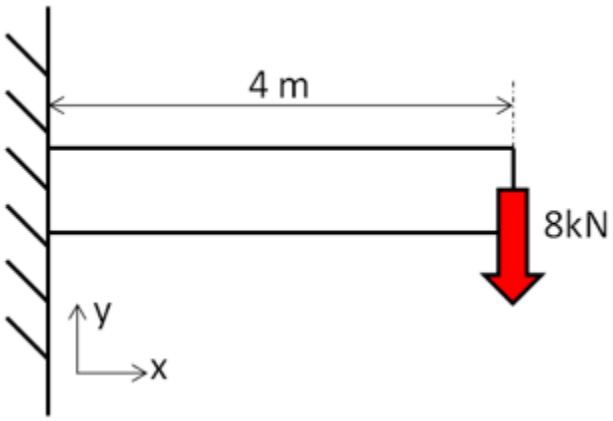
This module is from our free online simulations course at edX.org (sign up here). The edX interface provides a better user experience and the content has been updated, so we recommend that you go through the module there rather than here. Also, you will be able to see answers to the questions embedded in the module there.

Created using ANSYS 17.1

Problem Specification

Consider the beam in the figure below. It is clamped on the left side and has a point force of 8kN acting downward on the right end of the beam. The beam has a length of 4 meters, width of 0.346 meters and height of 0.346 meters (cross-section is a square). Additionally, the beam is composed of a material which has a Young's Modulus of 2.8x10^10 Pa. Using ANSYS, calculate the following:

1. Deformation of the beam
2. Maximum bending stress along the beam
3. Bending moment along the beam



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

Go to all ANSYS Learning Modules