ANSYS - Cantilever Beam Modal Analysis

Author: John Singleton, Cornell University

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

Important Note

Prior, to attempting this tutorial, you must complete the Cantilever Beam Tutorial. The Cantilever Beam Tutorial covers the static structural analysis for the same geometry. If you have not yet completed the Cantilever Beam Tutorial, please do so now.

Cantilever Beam Modal Analysis

Created using ANSYS 13.0

Problem Specification

Consider an aluminum beam that is clamped at one end, with the following dimensions.

Length	4 m
Width	0.346 m
Height	0.346 m

The aluminum used for the beam has the following material properties.

Density	2,700 kg /m^3
Youngs Modulus	70x10^9 Pa
Poisson Ratio	0.35

Using ANSYS Workbench find the first six natural frequencies of the beam and the mode shapes.

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

Go to all ANSYS Learning Modules