

Cantilever Beam Modal Analysis - Verification & Validation

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](#)

Verification & Validation

For our verification, we will focus on the first 3 modes. ANSYS uses a different type of beam element to compute the modes and frequencies, which provides more accurate results for relatively short, stubby beams such as the one examined in this tutorial. However, for these beams, the Euler-Bernoulli beam theory breaks down and is no longer valid for higher order modes.

Verification

Comparison with Euler-Bernoulli Theory

From our Pre-Analysis, based on Euler-Bernoulli beam theory, we calculated frequencies of 17.8, 111.5 and 312.1 Hz for the first three bending modes. The ANSYS frequencies for the first three bending modes are 17.7, 107.0 and 285.2 Hz. Note that in the ANSYS results, the third mode is NOT a bending mode. So the fourth mode reported by ANSYS is the third bending mode. These results give percent differences of 0.6%, 4.2% and 8.7% between ANSYS and theory. Thus the ANSYS results match quite well with Euler-Bernoulli beam theory. Note that the ANSYS beam element formulation used here is based on Timoshenko beam theory which includes shear-deformation effects (this is neglected in the Euler-Bernoulli beam theory).

Comparison with refined mesh

Next, let's check our results with a more refined mesh. We'll run the simulation with 25 elements instead of 10. Following the steps outlined in the Mesh Refinement section of the [Cantilever Beam Verification and Validation](#), refine the mesh.

Meshing the beam with 25 elements yielded the following modal frequencies:

Tabular Data		
	Mode	☒ Frequency [Hz]
1	1.	17.68
2	2.	107.03
3	3.	179.16
4	4.	285.07
5	5.	318.23
6	6.	524.21

These modal frequencies are all very close to those computed with a mesh of 10 elements, meaning that our solution is mesh converged.

[Go to Exercises](#)

[Go to all ANSYS Learning Modules](#)