ANSYS - Wind Turbine Blade

Author: Ben Mullen, 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

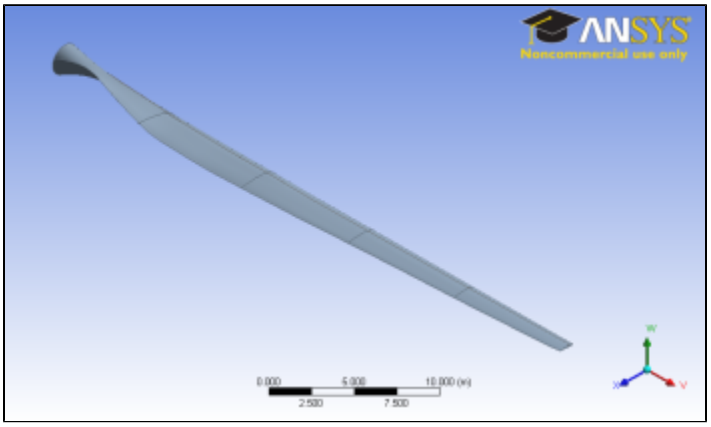
Wind Turbine Blade

Created using ANSYS 13.0

⚠ This tutorial is not being updated any more. We recommend that you follow this newer tutorial on fluid-structure analysis of a wind turbine blade. Thank you!

Problem Specification

This tutorial is based on this M.Eng project report completed at Cornell University in 2011. In this exercise, we will examine the stresses and deformation of a wind turbine blade under a force load.



Click here to enlarge imageThe blade is composed of an outer surface and an inner spar. The spar is 0.02 meters thick and the outer surface is of varying thickness. A table containing the material properties of the two parts is below. Structural steel will be used for both the blade and the spar.

	Spar	Wind Blade
Young's Modulus (Pa)	2e11	2e11
Poisson's Ratio	0.30	0.30
Thickness (m)	0.02	0.03 - 0.00048485*X

We are now ready to open ANSYS Workbench

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

Go to all ANSYS Learning Modules