Wind Blade Analysis for Wind Power - Numerical Results

This page has been moved to https://courses.ansys.com/index.php/courses/wind-blade-analysis-for-wind-power-using-ansys-fluent/lessons/numerical-results-lesson-7-22/
Click on the link above if you are not automatically redirected in 10 seconds.

Author(s): Zachary Brothers, 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

Numerical Results

Depending on the mesh used, values for the torque on the blade can range from 0.02 to 0.04 N-m for the simulation as it was shown in the videos. Newer versions of ANSYS may use a different cell count, mesh quality, and solver algorithm which accounts for this discrepancy.

In CFD-Post, when calculating the torque, newer versions of ANSYS do not let you select "Air" as the fluid. The two options under the "Fluid" dropdown are "All Fluids" and "Mixed" - the recommended selection is "All Fluids", but both selections should give you the same final answer for the torque.

Go to Step 7: Verification & Validation

Go to all FLUENT Learning Modules