FLUENT - Wind Blade Analysis for Wind Power

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

Author(s): Zachary Brothers, Cornell University

Wind Blade Analysis Problem Specification

Created using ANSYS 2020 R1

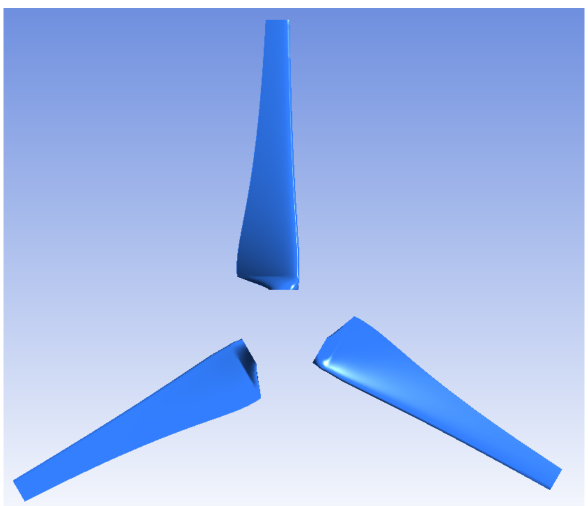
Learning Goals

In this tutorial, you will learn how to:

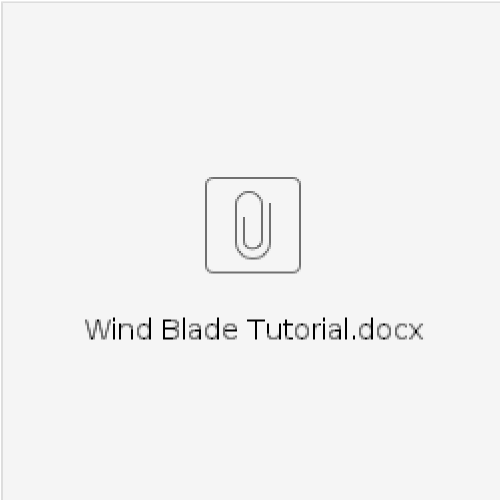
- Develop a CFD solution for wind turbine rotor using ANSYS Fluent
- Develop a a 3D mesh, solve for the flow field, and plot results

Problem Specification

This tutorial shows you how to simulate the flow around a given “standard” wind turbine rotor using Ansys Fluent. The rotor geometry is shown in the figure below. The hub is neglected for simplicity since it doesn't contribute to the power generated by the rotor. Only one blade is considered in the simulation set-up by using periodic boundary conditions. The simulation yields the velocity and pressure fields around the blade and the power generated.



The following document lists the steps of this tutorial with general directions:



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

Go to all FLUENT Learning Modules