FLUENT - Vertical Axis Wind Turbine (Part 2)

This page has been moved to https://courses.ansys.com/index.php/courses/vertical-axis-wind-turbine-part-2/
Click in the link above if you are not automatically redirected in 10 seconds.

Author: Julio Sampaio Gabriel de Pieri, Cornell University

Vertical Axis Wind Turbine - Part 2

Created using ANSYS 16.2

This tutorial has two parts. The first part will analyse the flow using Moving Frame of Reference, while the second part will use the Sliding Mesh feature.

> To access Part 1 of the tutorial, click here

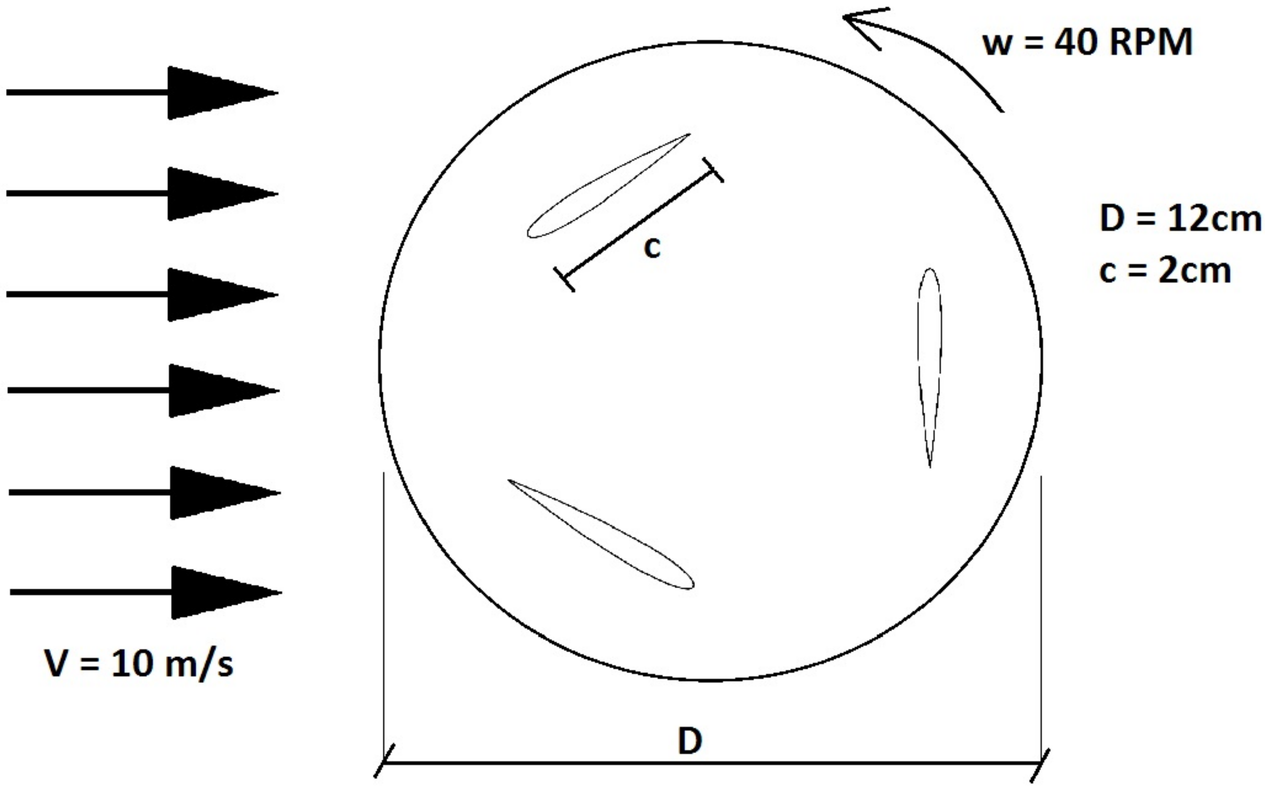
The setup for this part 2 of the tutorial has mostly been done during part 1. So it's highly recommended that you go through the first part, as **we will start from the same project file**.

Learning Goals

In this tutorial, you will learn to:

- Determine the flow behavior over a Vertical Axis Wind Turbine
- Apply concepts of Moving Frame of Reference (part 1) and Sliding Mesh (part 2) in FLUENT to simplify a transient problem into a steady state situation

Problem Specification



Consider an uniform flow of V = 10m/s passing through a Vertical Axis Wind Turbine (VAWT) as sketched above. The VAWT has a diameter of 12cm and 3 equally spaced blades, each one with a chord length of 2cm. For simplification, consider that it spins with a constant angular velocity of 40 RPM*. The center of each blade is located 0.04m from the center of the hub.

Note that this is a *Darrieus* VAWT, which is Lift based, in contrast to the *Savonius* VAWT, which is Drag based. This is an intensive field of research, and at Cornell we have the Fluid Dynamics Research Laboratory, directed by Prof. Charles Williamson. In the last section we will compare results with the experimental data obtain by the lab.

* Note that this is an important simplification. Here we're considering that the turbine is already spinning independently of the flow, what is clearly not true, since the flow is responsible for spinning the turbine. There is only one stable combination of incoming flow velocity and RPM, given the geometry and mass of the turbine. However, modeling the movement of a geometry caused by the flow is considerably difficult, and require use of a method called "6DOF solver". A tutorial on that is intended to be made in the future. For now, let's stick simpler analysis, first a steady state picture of the problem (part 1 of this tutorial), and later a transient analysis (this tutorial).

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

Go to all FLUENT Learning Modules