# Vertical Axis Wind Turbine (Part 2) - Pre-Analysis & Start-Up

Author: Julio Sampaio Gabriel de Pieri, Cornell University



# Pre-Analysis & Start-Up

## Pre-Analysis

In the *Pre-Analysis* step, we'll review the following:

- **Mathematical model:** (e.g.: We'll look at the governing equations + boundary conditions and the assumptions contained within the mathematical model.)
- **Numerical solution procedure in ANSYS:** (e.g.: We'll briefly overview the solution strategy used by ANSYS and contrast it to the hand calculation approach.)
- **Hand-calculations of expected results:** (e.g.: We'll use an analytical solution of the mathematical model to predict the expected stress field from ANSYS. We'll pay close attention to additional assumptions that have to be made in order to obtain an analytical solution.)

### Mathematical Model

Similar as before, but considering a rotating frame of reference for the hub.

In part 1 we only analyzed one position of the hub relative to the incoming wind, which might be an over-simplification. We also didn't account for the vortices generated by the blades passing upwind on the blades downstream.

All of these will be accounted in this Sliding Mesh method. In this case Fluent will actually move all components of the geometry and solve a transient problem.

### Numerical Solution Procedure in ANSYS

Similar as part 1, but now Fluent will iterate in each time step until it either converged, or reached the maximum iterations per time step defined by the user.

### Expected Results

We expect a similar distribution of Velocity, Vorticity and Turbulent KE and before. We also expect the Moment coefficient to be oscillatory.

## Start-Up

- Open the file from part 1 of the tutorial.
- Duplicate the project
    - Name it "Sliding Mesh"
- Click and drag "Solution" cell from MFR into the Solution cell of the new Sliding Mesh project

**Go to Step 2: Geometry**

Go to all FLUENT Learning Modules