

Wind Turbine Blade FSI (Part 1) - Pre-Analysis & Start-Up

Authors: Sebastien Lachance-Barrett (Cornell University) & Edwin Corona (University of Waterloo)

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Numerical Solution

6. Numerical Results

7. Verification & Validation

Pre-Analysis & Start-Up

Pre-Analysis

Mathematical Model

Governing Equations

The governing equations are the continuity and Navier-Stokes equations. These equations are written in a frame of reference rotating with the blade. This has the advantage of making our simulation not require a moving mesh to account for the rotation of the blade.

The equations that we will use look as follows:

Conservation of mass:

$$\frac{d\rho}{dt} + \nabla \cdot (\rho \vec{v}_r) = 0$$

Conservation of Momentum (Navier-Stokes):

$$\nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho (\vec{\omega} \times \vec{v}_r + \vec{\omega} \times \vec{\omega} \times \vec{r}) = -\nabla p + \nabla \cdot \vec{\tau}_r$$

Where

$$\vec{v}_r$$

is the relative velocity (the velocity viewed from the moving frame) and

$$\vec{\omega}$$

is the angular velocity.

Note the additional terms for the Coriolis force (

$$\vec{\omega} \times \vec{v}_r$$

) and the centripetal acceleration (

$$\vec{\omega} \times \vec{\omega} \times \vec{r}$$

) in the Navier-Stokes equations. In Fluent, we'll turn on the additional terms for a moving frame of reference and input

$$\vec{\omega} = -2.22 \hat{k}$$

.

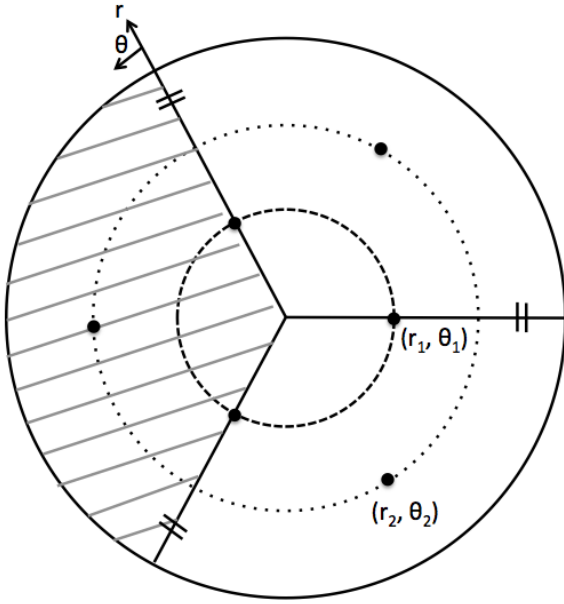
For more information about flows in a moving frame of reference, visit *ANSYS Help View > Fluent > Theory Guide > 2. Flow in a Moving Frame of Reference* and *ANSYS Help Viewer > Fluent > User's Guide > 9. Modeling Flows with Moving Reference Frames*.

Important: We use the Reynolds Averaged form of continuity and momentum and use the SST k-omega turbulence model to close the equation set.

Boundary Conditions

We model only 1/3 of the full domain using periodicity assumptions:

$$\vec{v}(x_1, \theta) = \vec{v}(x_1, \theta_1 - 120^\circ)$$



$$\begin{aligned}\vec{v}(r_1, \theta) &= \vec{v}(r_1, \theta_1 - 120^\circ n) \quad \text{For } n = 1, 2, 3... \\ &= \vec{v}(r_1, 240^\circ - 120^\circ(1)) = \vec{v}(r_1, 120^\circ) \\ &= \vec{v}(r_1, 240^\circ - 120^\circ(2)) = \vec{v}(r_1, 0^\circ)\end{aligned}$$

$$\begin{aligned}\vec{v}(r_2, \theta) &= \vec{v}(r_2, \theta_2 - 120^\circ n) \quad \text{For } n = 1, 2, 3... \\ &= \vec{v}(r_2, 180^\circ - 120^\circ(1)) = \vec{v}(r_2, 60^\circ)\end{aligned}$$

This therefore proves that the velocity distribution at theta of 0 and 120 degrees are the same. If we denote theta_1 to represent one of the periodic boundaries for the 1/3 domain and theta_2 being the other boundary, then

$$\vec{v}(r_j, \theta_1) = \vec{v}(r_j, \theta_2)$$

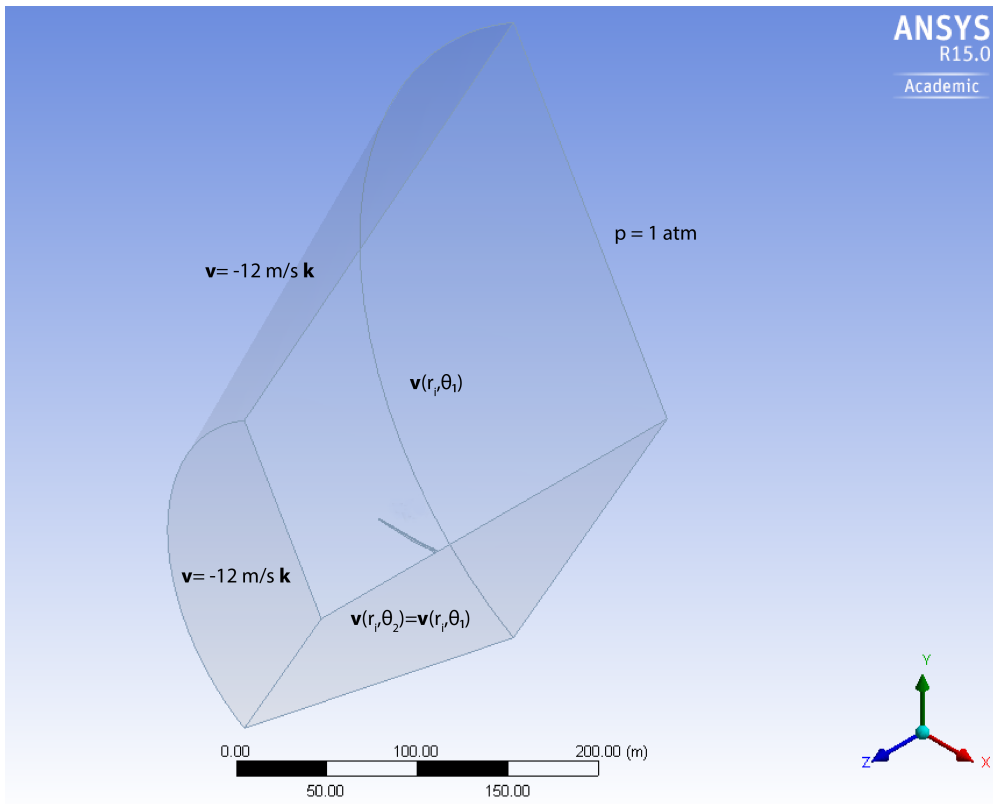
The boundary conditions on the fluid domain are as follow:

Inlet: Velocity of 12 m/s with turbulent intensity of 5% and turbulent viscosity ratio of 10

Outlet: Pressure of 1 atm

Blade: No-slip

Side Boundaries: Periodic



Numerical Solution Procedure in ANSYS

FLUENT converts these differential equations into a set of algebraic equations. Inverting these algebraic equations gives the value of (u , v , w , p , k , ω) at the cell centers. Everything else is derived from the cell centers values (post-processing). In our mesh, we'll have around 400,000 cells. The total number of unknowns and hence algebraic equations is:

$$400,000 * 6 = 2.4 \text{ million.}$$

This huge set of algebraic equations is inverted through an iterative process. The matrix to be inverted is huge but sparse.

In FLUENT, we will use the pressure-based solver.

Hand-Calculations of Expected Results

One simple hand-calculation that we can do now before even starting our simulation is to find theoretical wind velocity at the tip. We can then later compare our answer with what we get from our simulation to verify that they agree.

The velocity, v , on the blade should follow the formula

$$\mathbf{v} = \mathbf{r} \times \boldsymbol{\omega}$$

Plugging in our angular velocity of -2.22 rad/s and using the blade length of 43.2 meters plus 1 meter to account for the distance from the root to the hub, we get

$$\mathbf{v} = -2.22 \text{ rad/s } \hat{\mathbf{k}} \times -44.2 \text{ m } \hat{\mathbf{i}}$$

$$\mathbf{v} = 98.12 \text{ m/s } \hat{\mathbf{j}}$$

Additionally, by using the simple one-dimensional momentum theory, we can estimate the power coefficient which is the fraction of harnessed power to total power in the wind for the given turbine swept area. This analysis uses the following assumptions:

- The flow is steady, homogenous and incompressible.
- There is no frictional drag.
- There is an infinite number of blades.
- There is uniform thrust over the disc or rotor area.
- The wake is non-rotating.
- The static pressure far upstream and downstream of the rotor is equal to the undisturbed ambient pressure.

According to the M.Eng report presented in the problem statement, this blade is meant to resemble GE 1.5 XLE wind turbine blade. The [specification sheet of this turbine](#) states the rated power of this turbine to be 1.5 MW, the rated wind speed to be 11.5 m/s and the rotor diameter to be 82.5 m.

Thus, at rated wind speed,

$$C_p = \frac{P_{rated}}{P_{in}} = \frac{P_{rated}}{0.5 \rho A V_{rated}^3} = \frac{P_{rated}}{0.5 \left(1.225 \frac{\text{kg}}{\text{m}^3} \right) \left(\frac{\pi (82.5 \text{ m})^2}{4} \right) (11.5 \frac{\text{m}}{\text{s}})^3} = 0.30$$

The resulting power coefficient of 0.30 is very reasonable. We will compare it to power coefficient obtained from the simulation in the Verification & Validation section.

Start-Up

Please follow along to start this project! It is recommended to have these videos run side by side with your ANSYS project, with the video taking 1/3 of the screen space and the ANSYS window taking 2/3 of the screen space. An even better method is to use two monitors. This would allow running both the tutorial videos and ANSYS in full-screen. For example, the tutorial would be opened up on your laptop and ANSYS would be running on a lab computer. If you use the Cornell lab computers then make sure to bring some earbuds!

Summary of the above video:

1. Open up ANSYS Workbench
2. Drag Fluid Flow (Fluent) from Analysis Systems into Project Schematic
3. Rename Project
4. Save Project

[Go to Step 2: Geometry](#)

[Go to all FLUENT Learning Modules](#)