AN INVESTIGATION OF THE FLOW BEHAVIOR OVER A VERTICAL AXIAL WIND TURBINE (VAWT) APPLYING THE CONCEPT OF A MOVING FRAME OF REFERENCE

Cvetelina Velkova

Nikola Vaptsarov Naval Academy, Varna 9026, Bulgaria
E-mail: cvetelina.velkova1985@gmail.com

Abstract: The purpose of the study is to present a design, investigation and modeling of flow behavior of vertical axial wind turbine blades (VAWT). It has been developed a suitable numerical model using ANSYS software. The made numerical experiment clarified the steady state picture of the flow around VAWT with the help of function-Moving Frame of Reference that is available at ANSYS Fluent software. The main value of that study is that here the task like modeling of the behavior of any kind of complex geometry like VAWT caused by the flow which is extremely difficult is overcome by clarifying first of the steady flow field around VAWT.

Keywords: Dynamics, ANSYS, Fluent, VAWT, Moving Frame of Reference

1.   Introduction

Wind power is considered as a plentiful renewable energy whose technology and capacity is increasing over the years, allowing a breakthrough for harnessing energy from the wind [1].

A lot of innovative solutions are made by the researches over the topic during last years in order to improve the efficiency of wind turbines, [2, 3, 4]

Still unknown, Darrieus vertical axis wind turbines, Fig. 1 appears more suitable than the "classic" ones in different fields such as building integration, the extreme zones (observatories, mountain refuge), [5].

Although less efficient compared with three-bladed wind turbines, this kind of wind turbine can overcome the limits due to the size of the blades and their rotation speed.
The total size is smaller, and in some cases, when the motor is located at its base, this type of wind turbine can be cheaper.

The Darrieus wind turbine can run with a wind speed up to 220 km/h and in any direction.

The main shortcoming of this type of wind turbine is their difficult start, because the weight of the rotor on its base generates frictions.

Advantages:

-       The generator can be placed on the ground (depending on model)

-       Easily integrated into buildings

Disadvantages

-       Difficult start unlike the *Savonius wind turbine*

-       Low efficient

[3],[5].

Fig. 1 Darrieus Wind Turbine, [5]

It is made an important simplification at the presented study using the tutorial of Cornell University Web, [2]. Here it is been considering that the turbine is already spinning independently of the flow, which is clearly not true, since the flow is responsible for spinning the turbine. Combining the incoming flow velocity and RPM, and the given geometry and mass of the turbine is the only one stable combination. Although, modelling the movement of a geometry caused by the flow is considerably difficult. This study presents the simpler analysis by clarifying the steady state picture of the flow behaviour over VAWT [2].

2. VAWT –Problem Specification

Figure 2 shows Darrieus VAWT in details, it has been consider a uniform flow of V = 10m/s passing through a Vertical Axis Wind Turbine (VAWT) as sketched above, [2]. The VAWT has a diameter of 12cm and 3 equally spaced blades, each one with a chord length of 2cm, [2].
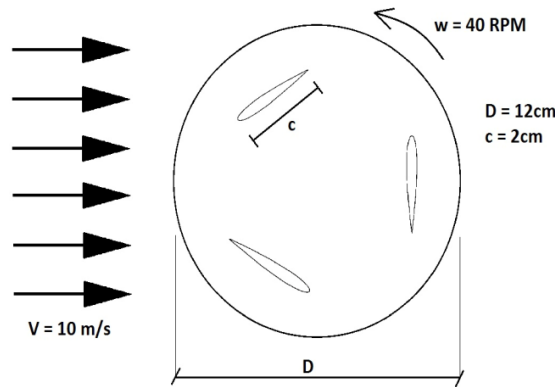

Fig. 2 Darrieus VAWT – details, [2]

For simplification, it has been considered that it spins with 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 [2].

3. Numerical Model of VAWT

The developed numerical model of VAWT under study is based on the relevant models of the Cornell University web [2].

3.1. Mathematical Model:

The governing equations solved here are conservation of mass and the Navier-Stokes equations, taken in a frame of reference moving with the turbine:

- **Conservation Mass**:

(1)
$$\frac{\partial p}{\partial t} + \nabla \rho \vec{V_r} = 0$$

- **Navie-Stokes equations**, simplified for constant angular velocity:

(2)
$$\frac{\partial}{\partial x}\left(\rho \vec{V_r}\right) + \nabla\left(\rho \vec{V_r}\vec{V_r}\right) + \rho\left(\overrightarrow{a_{coriolis}} + \overrightarrow{a_{centripeal}}\right) = -\nabla p + \nabla \overline{\overline{\tau}}_r$$

Where:

$\overrightarrow{a_{coriolis}} = 2\omega x \overrightarrow{V_r}$ - Coriolis acceleration

$\overrightarrow{a_{centripetal}} = \overrightarrow{\omega^2} x \vec{r}$ - Centripetal acceleration

Simplifying Navier-Stokes's equation using constant angular velocity is good, because Fluent solver can easily calculate this in a moving frame of reference without dealing with sliding mesh.

The problem that has been solved is turbulent, so that numerical solution procedure is made in ANSYS Fluent using the Finite-Volume Method (FVM).

3.2. Boundary Conditions

First, it is needed to create a region a few times larger than the main geometry of the turbine. This region is where the presence of the turbine disturbs the flow. This can be seen in Fig. 3 as the outer circle in the following figure. Note that we could have made any geometry for this "far-field" zone, but to simplify the boundaries a circle was chosen.
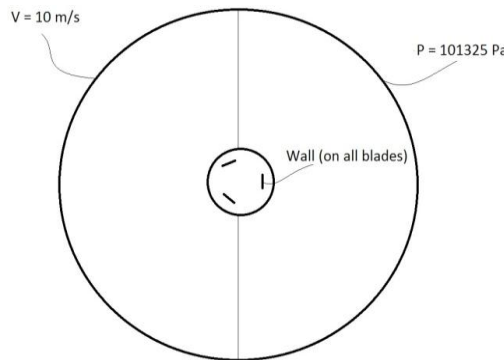

Fig. 3 Boundary Conditions, [2]

The next boundary conditions are shown in Fig. 3:

- Inlet (far-field): constant velocity in the x-direction of 10m/s, with a turbulent intensity of 5% and a turbulent viscosity ratio of 1.

- Outlet (far-field): an absolute pressure of 101325 Pa, or 1 atm.

- Blades: wall, so no velocity. (No-slip condition).

3.3. Validation of Numerical Results by Hand-Calculation.

The expression for calculating Tip Speed Ration (TSR), [2]:

(3)
$$TSR = \frac{rx\omega}{U} = \frac{\text{Velocity at the blade tip}}{\text{Incoming wind velocity}}$$

Meaning *r* as the distance from the center to the mid point of a blade: *r=0.04m*.The angular velocity is *40rpm* which corresponds to *4.1888rad/s*. Therefore, the expected velocity at the center of the blades is 0.1676m/s and therefore the TSR is 0.01676.

The problem is solved numerically with the additional assumption that the airfoils are flat plates of dimensions 0.1x2 cm, the plates are placed such that when they are at the upper-most part of its trajectory, they form an angle of 20deg with the horizontal axis.

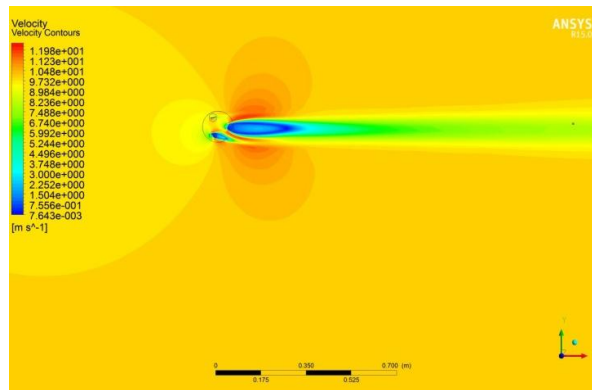## 4. Results and Analysis

4.1. Velocity Contours

Fig. 4 Velocity Contours

Note that Fig. 4 illustrates very clearly the effect of when the blade is perpendicular to the flow: a huge recirculation bubble is made. This will negatively affect other turbines placed downstream of this one. This is a very important thing to consider when designing an array of VAWTs. Fig. 4 is a single snapshot of the spinning of the turbine or a particular position. A transient analysis with a complete animation will be created in the future.
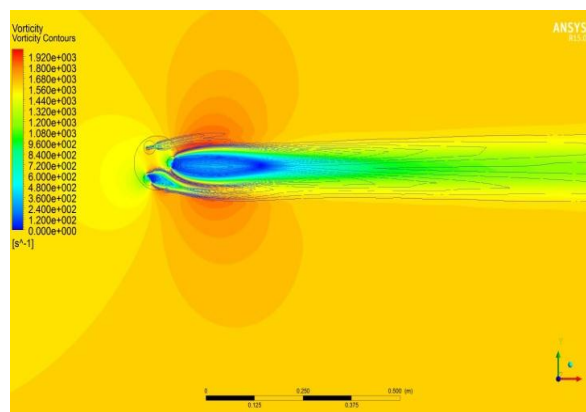
4.2. Vortices Contours

Fig. 5 Vortices Contours

Fluent has an ability to show also the vortices contours. It's important to analyze the vortices distribution downstream of the VAWT.

Observing Fig. 5 it can be seen the vortices from the emitted from the tips of turbine's blades. If we recall the flow over a plate perpendicular to the direction of the flow, can be concluded that the vortices being generated at the tips of the plate. Also it can be note from Fig. 5 that vortices separations is not so intense on the turbine's blade, it is less.

5. Verification and Validation of obtained Results.

5.1. Check Mass Flow

Taking into account the value of mass flow rate 73.50 kg/s get into out the domain, and then virtually everything works. The obtained imbalance of 7.58e-9 kg/s is negligible considering the total amount of flux in the system which is good related to the obtained results.

5.2. Tip Speed Ratio (TSR)

According to reference, [2] TSR can be extracted directly from the boundary conditions, because realizing the numerical model this is essentially checked, than the velocity at the wall is zero. Hence, the purpose of this check is more to verify if the mathematical model that is exposed above is correctly inputted at Fluent solver.

Citing a source [2], to calculate the TSR it is need to extract the velocity from CFD-Post. The only way to extract the velocity of fluid particles in touch with the turbine's blades is to extract it at particular location. Here it is made by plotting the velocity distribution along the X coordinate for the whole surface of the right turbine's blade, and then extract the value at x = 0.04 (because our reference is the velocity's value at r=0.04 m), see Fig. 6.
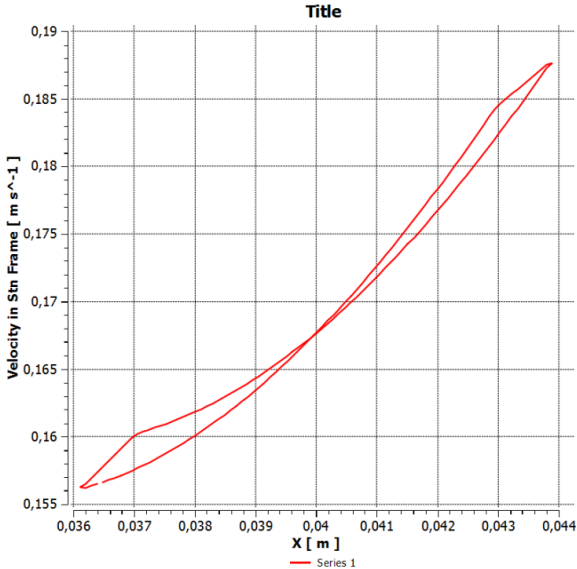

Fig. 6 Chart - velocity distribution along the X coordinate

The point of interest is the place where two red curves are coinciding and from Fig. 6, the Chart it can be read the coordinates of that point. Looking Fig. 6 it can be noticed that at the edge of the plot there is an abrupt change in velocity, as it was expected. The "closed loop" plot expect is in fact happening, but the curve collapse into a single line. The point where two curves are coinciding is slightly above the half between 0.165 and 0.170. Looking above into the made hand calculation, the expected value is about 0.1676 m/s. The obtained numerical value is the same, indicating that there might to have inputted the right mathematical model into the tool. Using the formula (3) given above for calculating the TSR is the obtained value divided to 10 m/s, the wind speed. So the obtained value of TSR is 0.01676.

5.3.   Angular Velocity

Citing [2], the obtained numerical results showed that the moment coefficient *Cm* is not zero, and this means that there is some extra torque applied on VAWT. Using [2], that is true, and it could even be used the *Cm* to calculated the power coefficient *Cp*.

6.   Conclusion

The study is performed for steady flow field around VAWT using the help of Moving Frame of Reference. Here it is determined the flow behavior over VAWT by using the simplification that the turbine is already spinning independently of the flow, of course that is not true, since the flow is responsible for spinning the turbine. But, modeling the movement of complex geometry as VAWT is extremely difficult and complex task, that's why into this study it is made a simple analysis of the flow picture into steady state.

The next stage into this study will be applying the concept of Sliding Mesh in ANSYS Fluent so that to can simplify the transient problem, which is the complex flow behavior around VAWT into a steady state situation.

Also as recommendation to improve the quality of the obtained results and flow picture it should be considered the possibility to make a mesh refinement i.e. to increase the number of cells for generated mesh around the object.

References

[1] Renewables 2014, Global Status Report. "REN21 Renewable Energy Policy Network for the 21st Century".

[2] Cornell University, Fluent Learning Modules, Vertical Axis Wind Turbine Part 1, July 7, 2016.

[3] Burton, T.; Sharpe, D.; Jenkins, N; Bossanyi, E. "*Wind Energy Handbook*". John Wiley & Sons, Chichester, UK, 2001.

[4] Martin O.L. Hansen. "*Aerodynamics of Wind Turbines*". Second edition, 2008.

[5] http://www.ecosources.info/en/topics/Darrieus_vertical_axis_wind_turbine.