

AN INVESTIGATION OF THE FLOW BEHAVIOUR OVER A VERTICAL AXIAL WIND TURBINE (VAWT) USING SLIDING MESH FEATURE

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 the 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 using the Sliding Mesh feature 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 simplifying the transient problem into steady state situation, [2].

Keywords: Dynamics, ANSYS, Fluent, VAWT, Sliding Mesh

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, 4, 5].

Still unknown, Darrieus vertical axis wind turbines, Fig. 1 appear more suitable than the "classic" ones in different fields such as building integration, the extreme zones (observatories, etc.)

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

[6].

This study is a next stage of conducted already numerical experiment of the complex flow behavior around VAWT using the concept of Moving Frame of Reference (MFR), [3]. Cited [2] and using the developed approach it is made an important simplification, here it is considered that the turbine is already spinning independently of the flow, which is not correct, since the flow is responsible for spinning the turbine. Quoting [2], modeling the movement of a geometry caused by the flow is considerably difficult, and requires use of a method called “6DOF solver”, (DOF- degree of freedom). This is a trend for the development of this study in a future. In the current stage of that study a simpler analysis is made, first a steady state picture of a problem, [3], and here it is using Sliding Mesh feature available in ANSYS Fluent so that to can simplify the transient problem, which is the complex flow behavior around VAWT into a steady state one, [2].



Fig.1 Darrieus VAWT, [6]

2. VAWT –Problem Specification

Figure 2 shows Darrieus VAWT in details, it has been consider a uniform flow of $V = 10\text{m/s}$ 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 40rpm . 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].

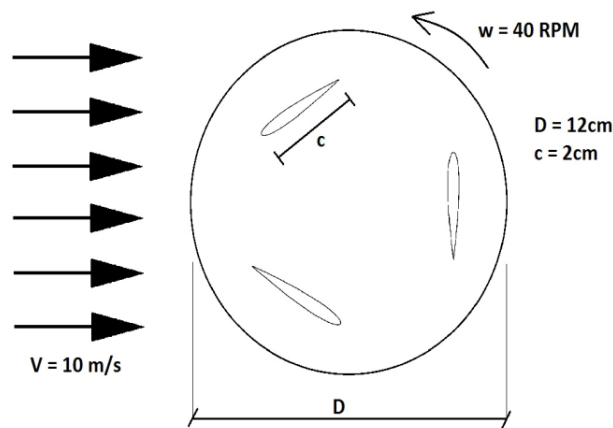


Fig. 2 Darrieus VAWT – details, [2]

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:**

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

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

$$(2) \quad \frac{\partial}{\partial x} (\rho \vec{V}_r) + \nabla (\rho \vec{V}_r \vec{V}_r) + \rho (\vec{a}_{\text{coriolis}} + \vec{a}_{\text{centripetal}}) = -\nabla p + \nabla \bar{\tau}_r$$

Where:

$$\overrightarrow{a_{coriolis}} = 2\omega x \overrightarrow{V}_r - \text{Coriolis acceleration}$$

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

Simplifying Navier-Stokes's equation using constant angular velocity is good, because Fluent solver can easily calculate this with Sliding Mesh features.

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

To solve this in Fluent, it is need 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 it could have made any geometry for this "far-field" zone, but to simplify the boundaries a circle was chosen.

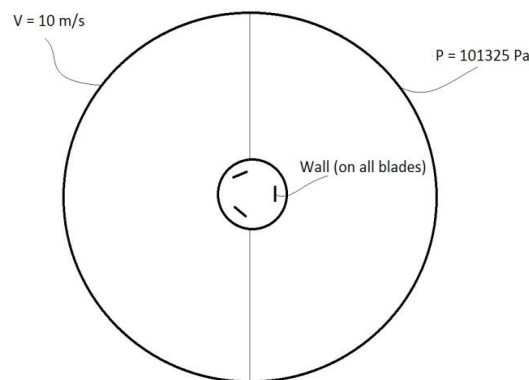


Fig. 3 Boundary Conditions, [2]

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): absolute pressure of 101325 Pa, or 1 atm.
- Blades: wall, so no velocity. (No-slip condition).

4. Results and Analysis

4.1. Moment monitor

After about reaching 6000 iterations is obtained solution convergence and the calculation is ok. First let's review moment monitor – Cm.

Observing Fig. 4 it can be noticed that when Cm is above zero that means that the flow is exerting a positive moment on the VAWT's rotor but when it's negative it's on the negative

direction. The positive value for C_m is the one that will generate power, the one that will accelerate the VAWT.

On the other hand the negative value of C_m is the one that is contrary to the movement of VAWT in terms. Fig.4 shows that our average value for C_m will be slightly above zero. So that means that the VAWT's rotor in fact has some extra moment so the angular velocity could be increased the little bit in the cell zone and then the VAWT will still spinning. It is possible to account for that moment and let the turbine find its own steady angular velocity the last based on the flow.

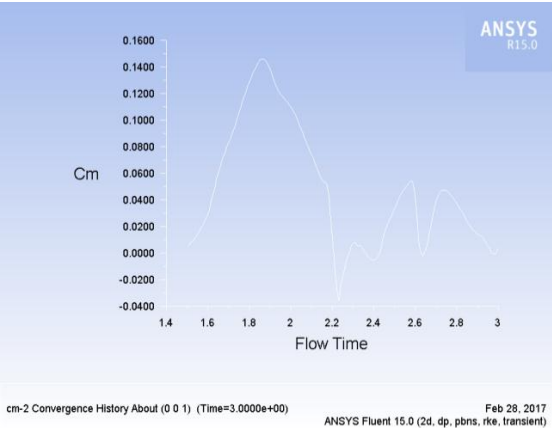


Fig. 4 Moment Monitor

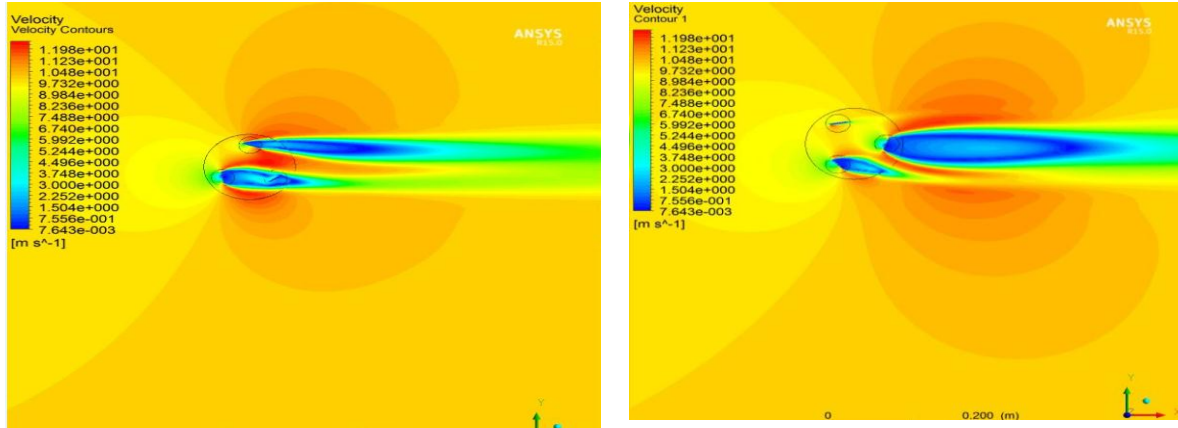
4.2. Comparison of Velocity Contours

Using the numerical results obtained into the first stage of the study, [3] both numerical results obtained with the Sliding Mesh feature and Moving Frame of Reference (MFR) for VAWT can be compared.

Let's first have a look at Fig. 5 which represents the made comparison of the Velocity Contours obtained with both features available at ANSYS Fluent.

Observing Fig. 5 the velocity contours around VAWT obtained with both features, (see Fig 5 a) with MFR, see Fig. 5 b) with Sliding Mesh) look pretty similar but we can see from Fig. 5 b) with Sliding Mesh that the velocity contours are slightly curved to the side compared to the Fig 5 a) with MFR. Last one means that the made numerical simulation with Sliding Mesh into this stage of that study predicts a little bit better what happens on the way behind the blades. Using Sliding Mesh techniques is referred to be even more actually accurate for the obtained and analyzing of the VAWT's flow field. And the last statement is the reason of conducted this stage of the study of investigating VAWT's flow field using both techniques available in Fluent: MFR and Sliding Mesh feature.

So even though the numerical simulations realized with MFR features and the papers written on their basis are useful and qualitative education of the velocity field the transient analysis for the flow behavior around VAWT using Sliding Mesh is referred to be even more actually accurate for the obtained and analyzing of the VAWT's flow field.



a) Velocity Contours, MFR b) Velocity Contours, Sliding Mesh

Fig. 5 Velocity Contours

4.3. Velocity Vectors

ANSYS Fluent gives an ability to observe velocity vectors around modeling object. Fig. 6 shows the obtained velocity vectors around VAWT with Sliding Mesh Features. It can be seen the velocity field distribution around whole VAWT.

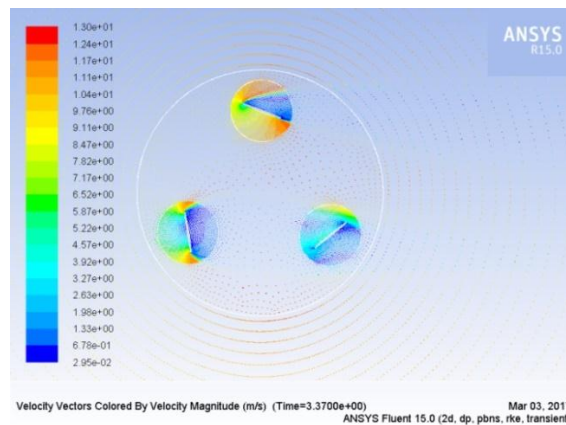


Fig. 6 Velocity Vectors obtained with Sliding Mesh Features

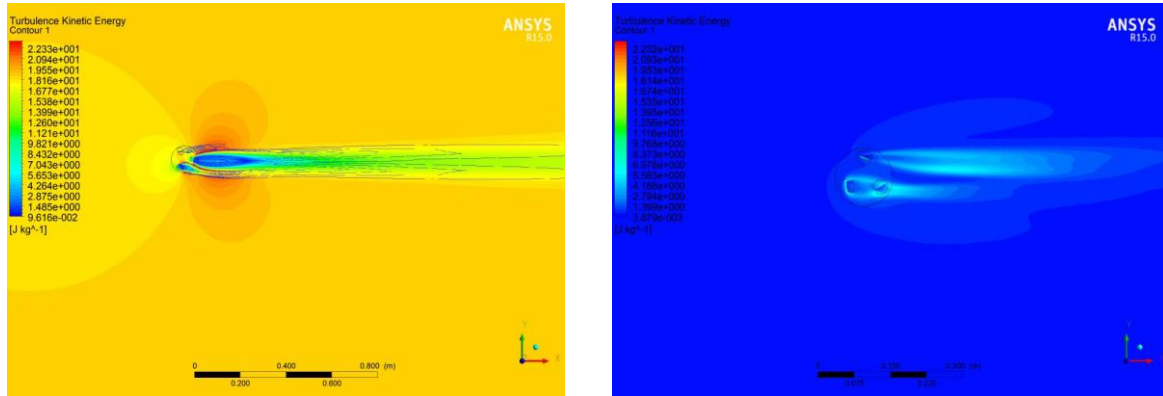
4.4. Turbulence Kinetic Energy (KE)

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

Observing Fig. 7 it can be concluded that making the numerical simulations for the analyzing of the flow field around VAWT using MFR and Sliding Mesh features its gives the

same value for turbulent KE around the turbine. The difference between obtained Turbulent KE with both features is in the distributions of kinetic energy around VAWT.

Also it can be noticed looking at Fig. 7 that considering Turbulent KE the MFR is overestimating a little bit compared to the Sliding Mesh.



a) Turbulence KE, MFR

b) Turbulence KE, Sliding Mesh

Fig. 7 Vortices Contours

5. Verification and Validation of obtained Results.

Using source [2] the realized numerical experiment and the obtained and shown up numerical results can be verified starting the numerical simulations again for transient process by using the different values about some input parameters. Changing reference value,

- Area to 0.0012m²
- Length to 0.02 m
- Velocity to 10 m/s
- Run the calculation for 150 time steps

Citing [2] it is important to notice that we should not take the new moment monitor Cm (see Fig. 8) for granted because it's an account for the pressure and the viscous forces on the blades. Since we don't have a very fine match on the blade right above our results for drag shall be enough précised.

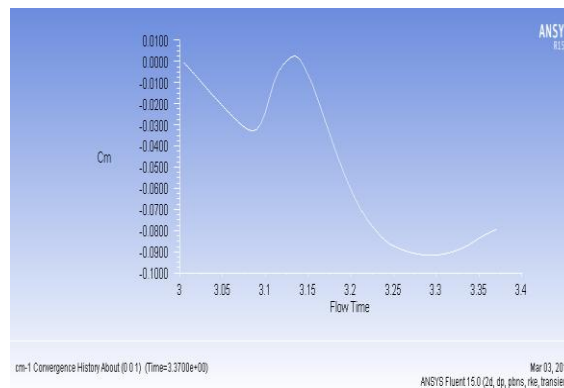


Fig. 8 Moment monitor Cm, in verification process

Hence this last part is just to illustrate a little bit in order to get a better prediction of C_m one should inflate mesh around the blades' airfoil tip get a finer mesh around them. Before was around 0.05. Now looking at Fig. 8 it can be noticed the exact same shape as before, (See Fig. 4) except for just scaled up and also it has to be remembered that it's not precise. So in that case there is a need to replace the mesh around the blades just to make sure we got a more precise moment monitor C_m . This oscillation that can be observed for Fig. 8 is due to the relative position of our blade's airfoil related to the incoming wind.

The forces acting on the blade's airfoil, the drag will generate the moment that is indirect what is being measured here.

So depends on the position of rotor's blades there are no related incoming wind believed in the drag it is always generate a positive moment or a negative one.

6. Conclusion

This study is a continuation of already realized numerical experiment using Moving Frame of Reference exposed at [3]. Here the examination of the flow behavior around VAWT is realized using Sliding Mesh feature in ANSYS Fluent so that the transient problem is simplified to a steady state situation.

It's used CFD post comparison of the numerical results for velocity contours and turbulent kinetic energy obtained with Moving Frame of Reference (MFR) and Sliding Mesh features.

Also for the validation and the verification of the obtained numerical results with that numerical simulation of the flow around VAWT is repeated with different kind of input parameters' values. By checking the moment monitor C_m (See Fig. 8) it can be seen that the suggested numerical approach, according to [2] for the modeling and the investigation of the flow behavior around VAWT has been a sufficiently precise. The last statement has to be indication that the model works properly.

As a recommendation for the next stage of that study it should be consider the possibility of modeling the flow in 3D, which will reflect in the complexity of the VAWT.

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] Velkova Cv., “An Investigation of the Flow over a Vertical Axial Wind Turbine (VAWT) applying the concept of a Moving Frame of Reference”
- [4] Burton, T.; Sharpe, D.; Jenkins, N; Bossanyi, E. “*Wind Energy Handbook*”. John Wiley & Sons, Chichester, UK, 2001.
- [5] Martin O.L. Hansen. “*Aerodynamics of Wind Turbines*”. Second edition, 2008.
- [6] http://www.ecosources.info/en/topics/Darrieus_vertical_axis_wind_turbine.