Numerical Simulation of Flow Field around a Darrieus Vertical axis wind turbine to Estimate Rotational wakes Size

Milad Babadi Soltanzadeh¹, Babak Mehmendoust Isfahani², and Davoud Toghrai Semiromi³

1) Ms Student, Mechanical Engineering Department, Islamic Azad University, Khomeini Shahr Branch. milad.babadi@iaukhsh.ac.ir
2) Assistance Prof, Mechanical Engineering Department, Islamic Azad University, Khomeini Shahr Branch. mehmandoust@iaukhsh.ac.ir
3) Assistance Prof, Mechanical Engineering Department, Islamic Azad University, Khomeini Shahr Branch. toghraee@iaukhsh.ac.ir

Abstract: Due to the increasing energy cost and reducing fossil resources, also with respect to environmental pollutions, sustainable energy usage is inevitable. In the last decades wind power is allocated to a particular share of sustainable energies. Design and optimization of wind systems requires aerodynamics analysis. Computational fluid dynamics is a guaranteed method for aerodynamics analysis of wind turbines. In this work at first we introduce briefly Darrieus vertical axis wind turbines and discussed their advantages and disadvantages compared to the horizontal axis wind turbines. Then tried to use a computational fluid dynamics method to simulate flow field around the Darrieus vertical axis wind turbine with three straight blades that use NACA0016 airfoil as profile of blades. To do this, we used RNG K-ε turbulence model to solve RANS equations of fluid motion. Simulation has been performed for three values of the rotational speeds. Velocity distribution of wakes is obtained in some horizontal directions.

Keywords: Computational Fluid Dynamics, Darrieus Wind Turbine, Sustainable Energy, Wind Energy, Vertical axis wind turbine.

I. INTRODUCTION

With increasing concerns about the environment, research on environmentally friendly renewable energy sources has increased. Focusing on these resources is due to increased environmental pollution (chemical and thermal pollution), increasing global demand for energy and reducing fossil energy resources. Renewable energy includes solar, biomass, geothermal, hydroelectric and wind energy. Wind energy is one of the most important that have put a various choices in front of researchers. This energy currently has the fastest growth rate among the other renewable resources.

Devices that are used for obtaining wind energy are known as wind turbines. A wind turbine is a device that converts the kinetic energy of wind into rotational energy of the rotor shaft [5]. They include two main groups, horizontal axis and vertical axis turbines. Vertical axis wind turbines are the first devices to convert wind energy to rational energy of shaft. Types of vertical axis turbine configurations include Darrieus, Savonius, Zephyr and windmills of Sistan [4].

Darrieus turbines have been installed at the first time in 1931. There are various configurations for Darrieus turbine blades that can be include straight-blade models, complex spiral curved blades and troposkein curved blades. Number of blades can vary from one to five (depending on economic considerations) [5]. Darrieus vertical axis wind turbines are the most efficient among the vertical axis wind turbines, but the main problems of them are low torque in starting and the lack of structural integrity [4].

Also the pitch angle of the blades can be fixed or variable. Changing the blade pitch angle can increase the torque to help better starting. Fixed pitch blades have a simpler structure, but produces less torque [4].

Currently, large scale vertical axis wind turbines is economically less attractive, but they can be used for areas far from the grid and Areas that are not suitable for wind farms of horizontal axis wind turbines. For this reasons, small scale vertical axis wind turbines have become more common in the world.

The main problems of vertical axis turbine are low start torque, rotor lift force, low efficiency and integrated structure.
Instead, advantages compared to horizontal axis turbines are as follows

- Not sensitive to the wind direction.
- The vertical axis blades have extra life because of the force of gravity and inertia is constant.
- Manufacturing costs are lower compared to the horizontal axis.
- Less sensitive to crossover turbulent flow and mechanical ability to work under storms and hurricanes conditions as they are safe.
- Due to the lower rotational speed than a horizontal axis turbine, vertical axis wind turbines are more silent [2], [5].

For this reasons, small vertical axis turbines can be installed close to the ground, on the roofs of houses in urban and rural environments and anywhere that turbulence is higher than wind farms sites.

Progression in the technology of vertical axis wind turbines need to increase the coefficient of performance, ability to start better, optimize the aerodynamic characteristics of airfoil used, study effects of solidity and blade pitch angle.

Fig. 1 illustrates average wind speed at a height of 40 m above ground level in different regions of Iran. Areas highlighted in red, light red and orange are suitable for wind farms of the horizontal axis wind turbines. Vertical axis turbines can be used in areas where wind speeds are lower and covered more area of the map.

II. DARrieUS Wind Turbine Aerodynamics

Solidity is one of the basic parameters of the Darrieus wind turbines. This parameter controls the rotational speed of the turbine to achieve maximum performance. High solidity usually lead to low tip speed ratio, which is one of another main parameters that cause reduced turbines performance coefficient. The high blade tip speed ratio provides a stronger interaction with the upstream wakes [2].

Both rigidity and tip speed ratio are dimensionless ratios, respectively, the equations (1) and (2) are introduced them.

\[ \sigma = \frac{NC}{R} \]  
\[ \lambda = \frac{\omega R}{V_\infty} \]

In these equations \( N \) is the number of blades, \( C \) is chord length of blade (airfoil), \( R \) is the radius of the rotor, \( \omega \) is the rotational speed of the rotor in radians per second and \( V_\infty \) is the free stream velocity. Fig. 2 illustrates the composition and shape of the Darrieus wind turbines. Fig. 3 shows the velocity triangles for the first blade when it rotates \( \theta \) degree from reference condition. \( V \) is the absolute velocity of the fluid stream within the affected area by the rotation of the rotor. \( U \) is the linear velocity of the blade that is \( \omega R \) and tangent to the circular path of the blade. \( W \) is the relative velocity to the blade. Blades are straight and NACA0016 airfoil is used. Air free stream moves toward the rotor, the rotor will absorb fraction of the kinetic energy of the air stream and leaving stream carry less energy than inflow (principle of conservation of energy). In other words, the wind turbines will act as a wind
shade, obtained fraction of energy and a low-speed turbulent wake of air is formed in the back of turbine.

![Simulated Darrieus wind turbine with three blades configuration](image)

**Fig. 2** Simulated Darrieus wind turbine with three blades configuration

**Fig. 3** Velocity triangles for first blade

III. **NUMERICAL SIMULATION OF FLOW FIELD**

To predict the behavior of wind turbines should be evaluated aerodynamic loads and flow field around turbines [4]. In fact, the complexity of three-dimensional vortex as well as dynamic stall should be considered. Experimental wind tunnel testing can help to better understand the aerodynamics of vertical axis wind turbines, but the cost of equipment and setups is really expensive. Also, results should be achieved in a manner that measuring devices have little effect on the flow field. These results should be modified to post-processing. Instead of experimental devices, computational fluid dynamics methods can provide details about the flow field without complex instruments usage. It also enables us to do regular studies like dimensional analysis without spending too much [5]. Although wind turbines work under unsteady three-dimensional flow but two-dimensional studies can be provide good results with an overview of the fundamentals [6]. Two-dimensional analysis disables to simulate tip vortex and usually examine rotor torque more than the maximum torque that experimental tests show.

Unlike horizontal axis wind turbines, vertical axis wind turbine is usually works under very high angles of attack conditions and the angle of attack is much more stalling angle. This phenomenon is more evident at low tip speed ratios. In the downstream areas (θ greater than 180 degrees), the blades are in the wake shadow region of the upstream blades, so finding the exact angle of attack of the blades is very difficult.

Stall occurs when the boundary layer flow is separated from the blade, in this case the lift force began to decline and rotational turbulent wakes are created. The blades in this area are under highly turbulent unsteady flow and non-linear behavior of the aerodynamic forces exerted on the blades. High turbulence in this region (wake shadow region) creates inaccuracy in calculating the downstream blade forces [5].

To simulate the flow field RNG K-ε turbulence model is used in this paper for solving the RANS equations. This model is derived using statistical methods and it’s accurate for fast-strained flows. Also because of the consider swirl on the turbulence, it provides good results in vortex flows and turbomachinery flow modeling.

**Fig. 4** illustrates flow field that solved. The rotor dimensions shown in this Figure is exaggerated because the domain size is much larger than rotor diameter. The grid near the rotor can be seeing in the Fig. 5. Finer grid near the rotor blades used to calculate field much more in line with reality results. Inaccuracy and errors in the grid production and the errors in the discrete equations of the flow controller equations play the role of free turbulence in computational fluid dynamics space.

![Flow simulation Domain (with exaggerate)](image)

**Fig. 4** Flow simulation Domain (with exaggerate)
In turbomachineries due to rotational motion of the rotor rotational flow is generated. It is better to use moving reference frame in this cases because the rotational motion of fluid is made because of the solid boundary rotation. This flow is inherently unsteady because rotor blades sweep a volume (surface in two-dimensional problems) of the environment but if there was not a stationary wall near the rotational boundary it’s good to allocate a coordinate system to moving (rotational) frame and solved the problem as steady flow. In this new coordinate system grids rotate with frame (rotor) and there is not any relative velocity between blades and cells. Since Darrieus vertical axis wind turbine is a turbomachinery we could use this method to simulate the flow field. Fig. 6 illustrates the region that new frame has been attributed to it.

Once the simulation is done when rotor is in \( \theta = 45 \). In this case, flow domain is solved for the three rotational speed \( \omega \), respectively, 5, 8 and 12 rad/s. In each case the tip speed ratio \( \lambda \) is equal to 1.5, 2.4 and 3.6 respectively. In all cases free stream velocity consider \( V_\infty = 10 \text{m/s} \).

Note that in all cases the tip speed ratio is lower than 4, so turbine blades operate in conditions of high stall. Table (I) provide details of the rotor.

### Table I. Rotor detail

<table>
<thead>
<tr>
<th>Airfoil</th>
<th>NACA0016</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor diameter</td>
<td>6m</td>
</tr>
<tr>
<td>Number of blades</td>
<td>3</td>
</tr>
<tr>
<td>Chord length</td>
<td>1m</td>
</tr>
<tr>
<td>Solidity</td>
<td>1</td>
</tr>
</tbody>
</table>

### IV. RESULTS AND DISCUSSIONS

Fig. 7 illustrates velocity contours in the whole domain for each case. As previous studies have shown, the rotational speed and the rotor tip speed ratio have great impact on geometry and length of rotational wakes. More rotational speed leads to less wake velocity. However, low rotational speed leads to low tip speed ratio, which make stall and generating wakes.

Fig. 8 shows velocity contours near the rotor for \( \omega = 5 \text{rad/s} \). As is obvious, first blade angle of attack is such that blade is under intense stall condition and flow is totally separated. Due to the separation a rotational wake generated in region (A) that showed in Fig. 8. The velocity is very low in this region and according to the Fig. 10 has a high intense of turbulence. The biggest rotational wake is related to the third blade that covered region (B) in Fig. 8. This area also has a very intense turbulence. High velocity area near the second blade is due to the direction of free stream velocity and velocity of the rotational flow that generated by rotor, in this area both of them have a same direction so they aggravate each other and make a high velocity area.
Fig. 7 Magnitude velocity contours in whole domain, (a): $\omega=5$ rad/s, (b): $\omega=8$ rad/s, (c): $\omega=12$ rad/s

Fig. 8 Magnitude velocity contours near the rotor for $\omega=5$ rad/s

Fig. 9 Pressure contours near the rotor for $\omega=5$ rad/s

Fig. 10 Kinetic energy of turbulence contours near the rotor for $\omega=5$ rad/s

Fig. 11 is the best simulation approach that illustrates velocity triangles in Fig. 3. This image represents the path lines and shows the impact of rotor rotational speed on the free stream velocity direction and determined rotational wakes structures very well. Free up-stream with value of $V_\infty$ and horizontal direction moves toward the rotor and become shelvy gradually in area close to the rotor. Then enter the area affected by the rotor and $V_\infty$ transform to $V$ that is absolute velocity of the flow in the rotor area. It is available to calculate angle of attack for first blade that showed in Fig. 11 with respecting to the linear speed of blade and velocities directions. The region (A) rotational wake in Fig. 8 path lines is obvious in Fig. 11 very clear.

Fig. 12 to Fig. 15 illustrate velocity magnitude distribution on the horizontal lines $y = 0$, $y = 2$, $y = 4$, $y = -2$, $y = -4$. This method could provide an adequate understanding of the size and length of the rotational wakes. Fig. 12 shows that the greatest impact of the wakes on the line passing through the axis of the rotor for rotational speed of 5 rad/s is to 25 meters far from rotor axis. After that, flow began the healing process and rotational wake completely disappears after almost 75 meters. Fig. 13 also indicates the same results that on other horizontal lines also strong affects up to 25 meters and then flow started to heal. This analysis method can be used for other rotational speeds, and the length and size of each wake could be analyzed this way.
V. CONCLUSION

In this paper, a numerical computational fluid dynamics method are used to simulate flow field around a Darrieus vertical axis wind turbine with three straight blades that used NACA 00016 symmetric airfoil for estimating wake size and study impact of rotational speed of rotor on the wake lengths. Select a moving reference frame help us to prepare good results that are accommodate with real overall view. Simulation done for tree rotational speeds and velocity, pressure and turbulence contours obtained in whole domain. Also, distribution of velocity magnitude along some horizontal direction are plotted to predict wake length in each direction and provide an opportunity to study impact of rotational speed on wake size. As the path lines obtained we are able to compare numerical simulation with analytical velocity triangles. Results showed that increasing of rotational speed leads to longer and much more turbulent wakes.

REFERENCES


