CFD analysis on the influence of angle of attack on vertical axis wind turbine aerodynamics

Prateek Srivastava¹, Sachin Kansal¹, Ashish Talwalkar¹, R Harish¹*

¹ School of Mechanical Engineering, Vellore Institute of Technology Chennai, Tamil Nadu-600127, India
* E-mail address: harish.r@vit.ac.in

Abstract
The Angle of Attack (AOA) in a Vertical Axis Wind Turbine (VAWT) plays an important role in determining the forces and the power generated by the wind turbine. It is difficult to find the suitable AOA due to the complex and constantly changing wind flow patterns. In this paper, we have performed CFD simulations using Ansys Fluent software, based on the constantly changing AOA. The CFD simulations were conducted by selecting a suitable range of AOA and the velocity of the wind. The selected range of AOA varied from 5 degrees to 25 degrees with increments of 5 degrees and the range of the air velocities varied from 7m/s to 21m/s with increments of 7m/s. The tests were also performed using the X-Foil software. The results obtained from the CFD simulations, done by using the Ansys Fluent Software and from the X-Foil software, were then compared to give a more accurate and optimized AOA and velocity value. This optimization of the AOA could enhance the overall performance of the Vertical Axis Wind turbine.

Keywords: Vertical Axis Wind Turbine (VAWT); Computational Fluid Dynamics (CFD); Angle of Attack (AOA); Ansys Fluent; X-foil.

1. Introduction
There has been a significant amount of increase in the number of researches performed on the use of Vertical Axis Wind Turbines (VAWTs) [1]. Today the major portion of the market has been captured by the Horizontal Axis Wind Turbines (HAWTs) and it has been estimated that the Vertical Axis Wind Turbines (VAWTs) will play an important and significant role in the coming decades [1]. Currently the Vertical Axis Wind Turbines face the problem of lower efficiency as compared to the Horizontal Axis Wind Turbines but over the past few years there has been an extensive research carried out on increasing the efficiency and improving the aerodynamics of the Vertical Axis Wind Turbines [1]. For horizontal axis wind turbine many studies have tried to reduce the cut in speed by redesigning the blade and by optimizing the power coefficient, blade design constraints, cost and blade mass both experimentally and analytically [2]. Harmonic balance and Navier stroke technology are compared for HAWT and VAWT and it was found that for HAWT harmonic balance solver is 10 times faster than Navier stroke technology, but for VAWT harmonic balance unsteady flow analysis is lower than in comparison with HAWT and Harmonic balance solution features undesired oscillations due to wide harmonic content and high
level of stall predisposition of this flow field type [3]. Vertical Axis Wind Turbines (VAWTs) are classified into two types namely the Savonius and Darrieus designs. The Darrieus design depends on the lift produced by the aerodynamically profiled airfoil blades while the Savonius design depends on the drag force produced because of the bucket-shaped vanes [1]. The Darrieus design has a much higher power coefficient and are the most suitable ones for any large-scale usage. The Darrieus designs also have a better efficiency than the Savoinus design because of the aerodynamic shape [1]. In the Darrieus Design of the VAWT's the performance is highly dependent on the incident Angle of Attack (AOA). Hence for the optimization of the turbine design it is extremely critical to obtain an accurate estimation of the incident flow and the AOA [1].

Simple straight blade Darrieus type VAWT was studied and it was seen that the NACA Symmetric airfoils are not suitable for small scale modeling instead of that one should use high lift and drag coefficients to utilize high lift and low drag asymmetric airfoils for low-speed operations [4]. Wind tunnel approach was used to evaluate the efficiency of VAWT, to study the effects in urban environment. The setup to the Darrieus VAWT under values of turbulence over 5% in comparison to free stream and it found out that turbulence do have negative significant effect on turbine performance [5]. VAWT (straight blade Darrieus) operating at low TSR are addressed using numerical approach via finite discretization of 2D Unsteady RANS equations on a multiple sliding mesh and validated via experimentation. To increase the efficiency pitch angle was varied sinusoidally [6]. There has been an intensive research on improving the efficiency of the straight bladed VAWT by controlling the AOA of the blade during the rotation of the blade around the Vertical Axis [1]. There is a complex aerodynamic phenomenon around the blades due to the interactions between the wind and the VAWT rotating blades and several studies have showed the instantaneous power and torque generation over a rotating cycle [1]. One of the studies has used the MATLAB software in which the torque and speed of VAWT was compared. It was found that although VAWT has less speed the torque generated is high. Due to Beltz Limit (wind turbine efficiency) only 59.3% of wind energy is only used. And it was found that turbine charging time will be more than 6 hours [7]. The actuator Cylinder flow model is defined as ideal VAWT rotor and it seems to exceed the beltz limit and is found close to uniform loading. For fixed pitch VAWT the maximum obtainable power coefficient is limited by the cyclic variation of inflow angle and relative velocity leading to a loading that seems to deviate from the uniform loading. An increase in airfoil drag by 0.001 leads to 1% decrease in power coefficient [8]. To find out the aerodynamic reasons for differences in power generation in different turbine design of the VAWT's it is necessary to carry out an in-depth more detailed aerodynamic analysis of the AOA [1]. BSL and SST models were compared, and it was found that BSL model changes gradually to the k-epsilon model near boundary-layer edge. Whereas SST model is able to incorporate the transportation of principle turbulent shear stress in adverse pressure gradients [9]. The Effect on the curved streamline region on incoming blade is studied [10]. This suggests that the extra incidence seen on the airfoils referred as “virtual incidence angle”. Unsteady CFD simulation of different airfoils and their AOA are compared to the theoretical results [10]. This study shows the in-depth analysis of the NACA0015 airfoil with different AOA ranging from 5, 10, 15, 20 and 25 degrees corresponding with velocities ranging from 7m/s to 21m/s with increments of 7m/s [1]. The simulations are carried out by using the ANSYS Fluent Software and the results are compared and validated with X-Foil software for static conditions.

| **Nomenclature & Abbreviations** |
|----------------------------------|
| $C_l$ = Coefficient of Lift       | HAWT = Horizontal Axis Wind Turbine |
| $C_p$ = Pressure Coefficient      | VAWT = Vertical Axis Wind Turbine   |
| $C_d$ = Coefficient of Drag       | SST = Shear Stress Transport        |
| AOA = Angle of Attack (deg)       | TSR = Tip Speed Ratio               |
| CFD = Computational Fluid Dynamics| $\alpha$ = Angle of Attack (deg)   |

### 2. Methodology
In this paper we have identified two significant reference points on which the CFD Simulations are carried out. The CFD tests were run on the ANSYS Fluent Software and then validated using the X-Foil Software.
For the simulations done in the ANSYS Fluent Software we have used the SST-k ω equations. This equation was chosen over the SST k-ε equation because the k-ω model is sensitive to the freestream value of ω which is added at the inlet. The near wall damping function (f) in the k-ε model is quite unreliable and does not give accurate results. In the SST model a viscosity limiter is also added. This limiter results in better agreement with the experimental measurements of the separated flow. The F2 is a blending function and if F2 or S is large then the viscosity value reduces.

\[ \mu_t = \frac{\rho k}{\omega} \] (1)

The equation (1) gives us the representation of the original SST model

\[ \mu_t = \frac{a_1 \rho k}{\max (a_1 \omega, S F_2)} \] (2)

The Equation (2) gives us the representation of the modified SST model here F2 is the blending function and S indicates the Shear Stress

\[ \frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho \mathbf{U} e) = \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla e \right) + C_1 e P_k \frac{\epsilon}{\kappa} - C_2 \rho \frac{\epsilon^2}{\kappa} \] (3)

Equation 3 is a representation of the k-epsilon (k-ε) equation.

\[ \frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho \mathbf{U} \omega) = \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla \omega \right) + \frac{\gamma}{v_l} P_k - \beta \rho \omega^2 \] (4)

Equation 4 is a representation of the k-omega (k-ω) equation.

The Shear Stress Transport turbulence model or the SST model is a widely used two-dimensional eddy-viscosity turbulence model which has a significant use in Computational Fluid Dynamics. This model combines both the k-omega turbulence model and the k-epsilon turbulence model. The k-omega turbulence equation is used in the inner region of the boundary flow layer while the k-epsilon model is used in case of free shear flow.

The following study is divided into two parts: -
1) The CFD Simulations were performed on the airfoil with different AOA and were examined for C_d, C_l, C_p, Stream Function, Lift Force and Drag Forces. These results were then validated with the results obtained from the X-Foil Software.
2) A Vertical Axis Wind Turbine (VAWT) with 2 blades is modeled and the CFD simulations are performed for half rotation from 0-180 degrees by using ANSYS Fluent Software. The TSR, C_p, Turbulent Energy are studied with an incremental increase in angle by 30 degrees.
CFD Simulations using ANSYS Fluent Software and X-Foil Software

Geometry. The airfoil chosen for this study was the NACA0015 airfoil. The airfoil has a chord length of 1 m. The Computational Domain created around the airfoil has the following dimensions: - The Semicircle in front of the airfoil has radius 5 m and the rectangular shape joining the semicircle has length 10 m and breadth 5 m. Figure 1 shows us the front view of the geometry constructed with the airfoil placed at the center.

Figure 1. Geometry Isometric View

Meshing. The following model is the meshed using Structured Face Meshing. We have used quad type mesh because it shows reduction in numerical diffusion when mesh is aligned with the flow. It can provide higher quality of solution with a smaller number of nodes than triangular mesh. The Structured Face Meshing is used as the ANSYS Software can generate an irregular mesh which will give a less accurate or irregular results. Hence, to obtain more accurate results we are using the Structured Face Mesh to obtain a smooth uniform meshing. The mesh has minimum length scale of 3.932X10^{-5} m and a maximum length scale of 0.4331514 m. The mesh has 66604 cells, 133842 faces and 67238 nodes with 1 Partition for AOA 5. Figure 2 shows us the meshing of the whole domain whereas Figure 3 shows us the meshing near the airfoil section.
**CFD Simulations and Contour Plots.** After the mesh was obtained the setup was then ready for the simulations in the ANSYS Fluent Software. The setup was placed under the SST-k omega turbulence model where the inlet velocity was varied from 7m/s to 21m/s with increments of 7m/s to obtain the contour plots and the pressure coefficient values by running iterations in the ANSYS Fluent Software.

**Figure 2.** Mesh for AOA 5 degrees

**Figure 3.** Mesh for AOA 5 degrees
X-foil Software. The results obtained after the CFD simulations run on the ANSYS Fluent Software were then compared with the results obtained by using the X-Foil Software. In the X-Foil Software the airfoils are pre-loaded and have to be selected from the database. The NACA0015 airfoil was selected, and system was run in viscous conditions and the AOA and the Reynolds number were varied according to the different velocities. This variation was done to obtain the pressure coefficient values. The values of pressure coefficient obtained from X-foil and ANSYS Fluent were then compared by plotting a graph.

\[ Re = \frac{\rho V L}{\mu} \]  

Equation 5 represents the Reynolds number equation which was used for the X-Foil simulations.

Modeling of VAWT with 2 blades

Geometry. For the VAWT the previously studied NACA0015 airfoil geometry is used to design 2 blade turbine which are placed 10 units apart from each other and have a 1-unit diameter center shaft. The spherical domain is of the radius 7.5 units, and this has been created to obtain more precise results inside the C-type domain with semi-circle of radius 25 units and length 70 units and breadth 50 units of the rectangular part.

Meshing. The meshing was done after obtaining the geometry represented in Figure 4 and Figure 5. The geometry was then given a Structured Face mesh. In this after the meshing was completed the following properties were obtained. The mesh was having a minimum length scale of 0.00272159m and a maximum length scale of 1.273737m. The Maximum Cell Skewness was obtained to be 0.9596229. The mesh was having 347664 cells, 690314 faces, 342648 nodes and 1 partition. The Figure 6 shows us the mesh near the designed blades and Figure 7 shows us the mesh of the computational domain.
**Figure 6.** Mesh near Airfoil

**Figure 7.** Mesh of the apparatus
**CFD Simulations.** In this model we have created two blade turbines. The CFD simulations were performed on these two-blade turbines by rotating them between angles of 0-180 degrees from the positive y-axis in the second quadrant. After 180 degrees the simulations will give the same results due to symmetric conditions as the position of the two airfoil blades will be interchanged and hence to save the computational costs the simulations were performed only between 0-180 degrees with an incremental increase of 30 degrees with inlet velocity 14m/s.

3. **Results and Discussions: -**

*Results of the Static Conditions*

After running the simulations on both the ANSYS Fluent Software and on the X-Foil Software the pressure coefficient results were obtained and are compared by plotting graphs. Figure 8, Figure 9, Figure 10 shows us the comparison between the results obtained from X-Foil Software and ANSYS Fluent Software for AOA 5 degrees.

![Figure 8. AOA 5, Velocity 7m/s](image)

![Figure 9. AOA 5 degrees, Velocity 14m/s](image)
Figure 10. AOA 5 degrees, Velocity 21m/s

Figure 11 and Figure 12 shows us the comparison between the results obtained from the X-Foil Software and the Ansys Fluent Software for AOA 10 degrees.

Figure 11. AOA 10 degrees, Velocity 7m/s
Along with these graphs’ other plots like the pressure coefficient plots, streamline plots etc. were also obtained.

Here, Figure 13 represents the pressure coefficient at a velocity of 21m/s for an AOA of 20 degrees. Figure 14 represents the pressure coefficient graph at a velocity of 21m/s for an AOA of 25 degrees. In the Figure 13 and Figure 14 we can see an increase in the boundary layer separation and the increase in wake region. We can also see an increase in the pressure coefficient value due to an increase in AOA.
Here the Figure 15 represents the Stream Function at 21m/s for an AOA of 20 degrees while Figure 16 represents the Stream Function for a velocity of 21m/s for an AOA of 25 degrees. The Figure 15 and Figure 16 Stream Function plots show almost similar behavior with slight difference in the wake region for different AOA with constant velocities.

Figure 17 shows the Drag force plot for all the AOAs, in which we can observe the exponential change in behavior after AOA10 to AOA 15. Also, in Figure 18 we can see that there is a sudden increment of lift coefficient for AOA 15 between velocity 10m/s and 20m/s. This change in lift coefficient is due to an increase in pressure difference between upper surface and lower surface of the airfoil for velocity magnitude ranging from 10m/s to 20m/s.
Here in Figure 19, we can see for AOA 10 degrees at 21m/s there is observable change in the pressure contour plot. This may happen as on the leading edge of AOA 10 degrees the separation happens above the stagnation point on leading edge and hence this results in the change in the pressure coefficient plot. This can be seen in the pressure contour plot given in Figure 20.

**Figure 19.** Pressure Coefficient plot for 21m/s

**Figure 20.** Pressure Coefficient plot for AOA-10 velocity 21m/s
Results of the modeled blades for VAWT

The results of the following CFD Simulations are shown in the Table1:

| Table1. Results of CFD Simulations. |
|-------------------------------------|
| Theta (θ) | 0   | 30  | 60  | 90  | 120 | 150 | 180 |
| Cp Max    | 0.9027191 | 0.875844 | 7.4495 | 1.07611 | 1.07938 | 1.029257 | 0.9027191 |
| Cp Min    | -1.319544 | -1.430744 | -42.82 | -2.071416 | -3.00123 | -2.738036 | -1.319544 |
| Velocity (m/s) | 18.45491 | 18.76133 | 82.907 | 26.9139 | 26.20361 | 25.9833318.45491 |
| TSR       | 1.3182 | 1.3401 | 5.9219 | 1.9224 | 1.8717 | 1.8560 | 1.3182 |
| Turbulent Viscosity (kg/m-s) | 0.2245579 | 0.3080004 | 1.426 | 0.0999327 | 0.6211136 | 0.2621533 | 0.2245579 |
| Turbulent Kinetic Energy (m²/s²) | 6.495335 | 19.41027 | 165.5487 | 7.0268 | 15.30278 | 19.35094 | 6.495335 |

Figure 21 shows the pressure coefficient contour plot at AOA 60 degrees whereas Figure 22 shows the pressure contour for AOA 120 degrees. We can clearly see the difference between the pressure coefficient plots for AOA 60 degrees and AOA 120 degrees.
As seen in Figure 21 for AOA 60 degrees the blades have a very high magnitude as compared to rest of the results due to the non-uniform loading at the blade of the wind turbine. To minimize this peak in turbulence and to obtain uniform results we have introduced a pitch of 10 degrees on the 60-degree geometry. The Figure 23 shows us the pressure coefficient plot for 60 degrees with 10-degree pitch. Figure 24 shows the Turbulent Kinetic Energy plot for AOA 60 degrees with 10-degree pitch. Figure 25 shows the Velocity Magnitude plot for AOA 60 degrees with 10-degree pitch.
Table 2 shows the comparison between the values of the CFD Simulations before inducing the pitch and after inducing the 10-degree pitch for AOA 60 degrees.
4. Conclusions:

4.1 Conclusions for Static Conditions

In all the above static simulations we notice a similar trend, or we can say constant trend in the maximum value of Stream function in all the simulated AOAs even if we consider the relative velocities. We noticed that with velocity of 7m/s Stream function is approx. 85.7Kg/s, with 14m/s the value is approx. 717.5Kg/s and with 21m/s we notice the value to be approx. of 275.25Kg/s. So, we can say that Stream function is not much affected by the AOA as it is a function of Reynolds number.

However, if we consider the pressure coefficient nature at different AOA, we see a constant pattern with the pressure coefficient values for AOA 5 and AOA 10 but between AOA 10 and 15 we see a sudden exponential increase in the pressure coefficient value. For velocity of 21m/s we can see that the maximum value is approximately 9.0148 bar, which is higher as compared to the value for AOA 10 and 15 which is 1.010792 bar and 0.9874867 bar respectively. We can see a similar trend with velocities 7m/s and 14m/s.

With above results we concluded that at AOA 10 and at a preferred velocity of 14m/s we get better CFD results as compared to other AOA and velocities.

4.2 Conclusions for Modeled VAWT blades

Drag forces for both the airfoil for angle 30 degrees as well as 90 degrees shows a little discrepancy if we consider the angle 60 degrees drag force data. There is an ultimate increase in turbulent kinetic energy for angle 60 degrees if we consider other angles. Angle 60 degrees is showing different results than the others in the matter of velocity magnitude, pressure distribution and turbulent kinetic energy. The increase in turbulence decreases the turbine power efficiency. As a result, to reduce this we induced a pitch angle of 10 degrees which reduced the turbulent kinetic energy from 165.5487 to 22.23506. The TSR was reduced from 5.9219 to 1.82857. These parameters obtained after inducing a pitch of 10 degrees are significant and equivalent to the ones calculated for 90 degrees and 120 degrees. We can see that the introduction of pitch is necessary to obtain uniform values and better turbine power efficiency in industrial uses and is the future of Wind Energy.

References:

[1]. Mohamed M. Elsakka, MDerek B. Inghama, Lin Maa, Mohamed Pourkashaniana - CFD analysis of the angle of attack for a vertical axis wind turbine blade
[2]. Shafiqur Rehman, Md. Mahbub Alam *, Luai M. Alhems and M. Mujahid Rafique-Horizontal Axis Wind Turbine Blade Design Methodologies for Efficiency Enhancement—A Review
[3]. Campobasso MS, Drofelnik J, Gigante F. Comparative assessment of the harmonic balance Navier-Stokes technology for horizontal and vertical axis wind turbine aerodynamics. Compute Fluids 2016;136(September):354–70.
[4]. Islam M, Ting DS-K, Fartaj A. Desirable Air foil Features for Smaller-Capacity Straight-Bladed VAWT. Wind Engineering. 2007;31(3):165-196.
[5]. Carbó Molina A., Bartoli G., De Troyer T. (2018) Generation of Uniform Turbulence Profiles in the Wind Tunnel for Urban VAWT Testing. In: Battisti L., Ricci M. (eds) Wind Energy Exploitation in Urban Environment. TUrbWind 2017. Green Energy and Technology. Springer, Cham.
[6]. Xisto C, Páscoa J, Trancossi M. Numerical analysis of design parameters with strong influence on the aerodynamic efficiency of a small-scale self-pitch VAWT. In: Volume 1: Advances in Aerospace Technology; 2015. p. 1–11
[7]. Augusta, F & Al Afif, Ahmad & Wulandari, Putri & Syahriar, Ary. (2020). Vertical axis wind turbine analysis using MATLAB. Journal of Physics Conference Series. 1517. 012065. 10.1088/1742-6596/1517/1/012065: H Aa Madsen et al 2014 J. Phys.: Conf. Ser. 555 012065

[8]. Menter, F.R. Zonal two-equation k-w turbulence model for aerodynamic flows. AIAA Paper 1993-2906, 1993.

[9]. Bianchini A, Balduzzi F, Ferrara G, Ferrari L. Virtual incidence effect on rotating air foils in Darrieus wind turbines. Energy Convers Manage 2016; 111(March):329–38.

[10]. Qing'an Li a, *, Takao Maeda b, Yasunari Kamada b, Junsuke Murata b, Toshiaki Kawabata b, Kento Shimizu b, Tatsuhiko Ogasawara b, Alisa Nakai b, Takuji Kasuy-Wind tunnel and numerical study of a straight-bladed vertical axis wind turbine in three-dimensional analysis (Part I: For predicting aerodynamic loads and performance)

[11]. P. D Abd. Aziz1, A. K. R Mohamad2, F.Z Hamidon3, N. Mohamad4, N. Salleh5, N. Mohd Yunus6 # Electrical Technology Section, Universiti Kuala Lumpur-British Malaysian Institute Batu 8, Jalan Sungai Pusu, 53100 Gombak, Selangor, Malaysia - A Simulation Study on Airfoils Using VAWT Design for Low Wind Speed Application

[12]. Suyash Nigama, Shubham Bansal, Tanmay Nema, Vansh Sharma and Raj Kumar Singh Department of Mechanical, Automotive and Production Engineering, Delhi Technological University, Delhi, India - Design and Pitch Angle Optimisation of Horizontal Axis Hydrokinetic Turbine with Constant Tip Speed Ratio

[13]. J. Morgadoa, *, R. Vizinho b, M.A.R. Silvestre a, J.C. Páscoa b-XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils

[14]. ANSYS Inc. “ANSYS FLUENT User’s Guide,” Canonsburg; 2013.

[15]. Youngren H, Drela M. Xfoil subsonic airfoil development System. Open source software available at http://web.mit.edu/drela/Public/web/xfoil; 2005.