Two-Dimensional Unsteady Flow Calculations of a Five Bladed Vertical Axis Wind Turbine

†*Nishant Mishra[1], Praveen Laws[1], Rajagopal Vinod Bethi[1], Santanu Mitra[1]

[1]Department of Mechanical Engineering, Shiv Nadar University, India

*Presenting author, †Corresponding author: nishant.mishra@snu.edu.in

Abstract

With simple construction and mechanism, Vertical Axis Wind Turbine (VAWT) is the most promising renewable energy resource in the modern era. A two-dimensional analysis on the aerodynamic behavior and performance of five bladed VAWT is investigated. The study is systematically focused mainly on parameters such as turbulence modeling, temporal discretization and mesh refinement to figure out the certainty of the unsteady flow behavior and stability of the scheme. All these parameters have a direct impact on the exactness of the simulation result, which is analyzed and presented. Later on, CFD validation is done by comparing with experimental five bladed data based on the aerodynamic behavior and performance. Lastly, it is observed from the numerical analysis that turbulence modeling, time step and mesh refinement has a high impact on the accuracy of the simulation results in understating the unsteady flow phenomena around and through the vertical axis wind turbine. It is very important to develop a fluid-solid integrated model towards understanding of energy production by varying different parameters of wind machine. It is believed that the present 2D work can be extended to 3D without much difficulty and can produce useful results before developing a prototype and making it viable for green energy production.

Keywords: Vertical axis wind turbine, Turbulence Modeling, Time Stepping, Mesh Refinement, performance.

Introduction

To comprehend unsteady flow physics of Darrieus turbine the free vortex approach turn out to give a better picture to figure out the wake formulation around the rotor. These wake formations plays a vital role for calculating the force and moment on the blade. Forces on the turbine are examined by the pressure acts on the blades and the moment using wake vortices impulse. Further this method is suitable only for rotor with single straight bladed with no inclusion of stall effect and extravagant [1]. Later on a model which incorporates dynamic stall effect in free vortex wake has been modeled to validate three dimensional vertical axis wind turbine. A time marching schemes is developed which computes the strength of the bound vortex and wake using Kutta-Joukowski law. In order to probe the effect of dynamic stall an indicial stall model is used which derives the stall effect with a physical approximation [2]. To figure the improved efficiency of vertical axis wind turbine a high fidelity model is developed to examine unsteady, 3D, viscous flow around the rotor. Solving RANS equation using spectral discretization, the model tend to study about solidity and tip speed. The exactness of the system relay on Fourier representation in time because unsteady flow always remains periodic thus gaining computational time by just extracting few number of modes [3].

Also, the complex flow behavior around the turbine due to unsteady flow field and varying angle of attack causes dynamic stall on the blades at low tip speed ratio tends to the formation of eddies. This hysteresis eddies traced from the blade surfaces are due to the impact of vortex shedding, main reason behind the lift reduction in the trailing edge and strong increase of drag

[4]. Further, to simply the physics of flow around the turbine using discrete vortex method, with an assumption that the boundary layer separation is not considered and applicable only for higher tip speed ratio. The method is built upon conformal maps, later solved using Fourier transformation with kutta condition to get the final solution. The model can be further developed to calculate additions effects such as skin friction drag, number blade effect and unsteady calculation of force [5].

The rotation of the turbine in CFD is imitated using the sliding mesh technique, data transfers transpire between the two interfaces. Suitable coupling through interface is made between the steady fixed domain and the unsteady rotating domain. Theses coupling data will be updated at each time step and stable exchange of fluxes between the two domains. When the turbine rotates the blades experience different angle of attack, so at each time step new set of transient aerodynamic parameters will be updated and calculated at each time step [6].The authentication of CFD data are validated against the flow field observed from a Particle image velocimetry (PIV), the system measure the upwind and downwind stall to predict the power coefficient [7].

CFD simulation on the effect of increasing the number of blades show that five bladed vertical axis wind turbine provides higher power coefficient at lower tip speed ratio which is of 20% greater than that of a three and four bladed turbine. The frequency of oscillation torque get reduced leads to an increase in torque coefficient at lower tip speed ratio. To add on increase in blades achieves a marginal reduction of peak radial force (Fr) which may cause structural damage [8].

Another major problem which hits the vertical axis wind turbine is its no self-starting aerodynamics, 2D weak coupling model is generated to analyze the interplay between the rotor and the inducing air. Discussion is made on parameters such as solidity and fixed pitch angle which reveals solidity in the range of $\sigma = 0.6 - 1.0$ provides a minimum self-start at t=0.8s. Delay in the start is influenced by the separated flow vortex with minimum lift [9]. Coupled pressure based solvers with k-ε turbulence modeling turn out to be comparably efficient solver setting which favors with accurate result in modeling periodic flow field. So considering all the research fact above understanding and characterizing the unsteady flow nature around the turbine is complex [10]. The fluid dynamics around the blade surface falls on the effect of tip speed ratio, affected by flow separation and wake created by its own and also from other rotating blades. All these unsteady physics get coupled and affects the performance of the turbine. Understanding all these above discussed details research in this works is spotlighted on culling most advisable time step, mesh refinement and turbulence modeling to harvest an exceptional performance of the turbine. Finally all these desired properties are conjoined to estimate the self-starting behavior and power coefficient actualize from different wind speed and tip speed ratio.

Methodology

Design Parameters

A five bladed 2D model of vertical axis wind turbine was designed in SolidWorks and reviewed in ANSYS Fluent, 2D model was preferred to cut down the computation time and to clearly understand the unsteady aerodynamics around the turbine blades. Numerical analysis is carried out with the turbine blades and neglecting hub and the supporting arm. Straight-bladed Darrieus rotor is characterized byNACA0018symmetric blades with a chord length(c) of 0.2 m, diameter of the turbine (D) 2.5m with an aspect ratio of 12.5 has been taken into account [11]. The geometry of the turbine is branched by a rectangular outer domain and circular inner domain. The rectangular domain host the inlet, far field wall and outlet, rotor is place in the center of the domain. Both inlet and outlet are placed at a distance of 25m from the center of the rotor in order to capture the wake vortices formulation and the width of the domain is 20m. The inner circular domain with an interface boundary condition has a diameter of 3m mounts the turbine blades. To mimic the rotation of the turbine sliding mesh technique in FLUENT is adopted, which is a time-periodic technique suitable of analyzing rotating parts. The model is developed ideal for moving frame of reference problems or rotating object in a domain. The rotor mesh slides with respect to the fixed mesh surface in the interface region. The mesh from both the domains should match each other to transfer data through interpolation [12].

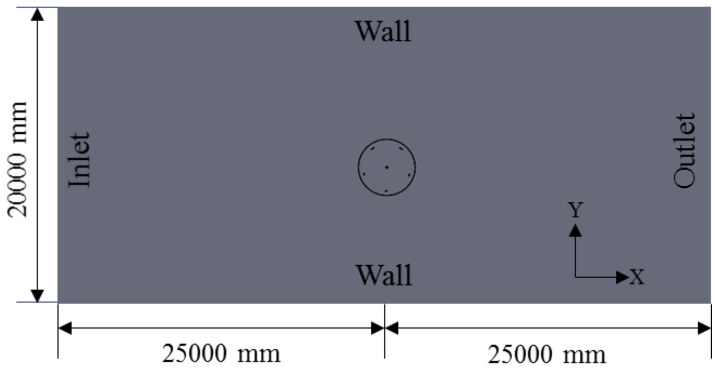

Figure 1.Turbine geometry and boundary condition

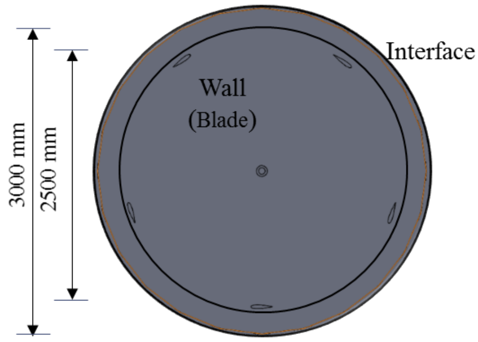

Figure 2. Rotating domain geometry

Table 1. Specification of the geometry

Design Aspects	
Number of blades	5
Domain	50 X 20 m
Blade position	72 degree apart
Chord Length (c)	0.2 m
diameter of the turbine (D)	2.5m
Interface	3m

Simulation and Discretization Parameters

A uniform velocity distribution is set along the x-direction at the velocity inlet. No-slip (U=0) condition obliged on the blade surface, top and bottom of the domain and in the outlet pressure outlet ($\Delta p=0$) boundary condition is imposed. Standard atmospheric air density and viscosity measures were chosen for the flow properties. A 2D, coupled, pressure based, transient, incompressible solver setting has been urged for the study. Pressure-Based solver has been applied with Gauss node based Interpolation Methods is used to discretize pressure gradients with second order accuracy which is more suitable for triangular mesh, same discretization is adopted for the turbulence equation. The pressure-velocity is set as coupled algorithm implicit discretization, which is highly effective scheme towards single phase flow and performs superior than the other segregated solvers [13]. The scheme is potential strong while dealing with large time step.

Turbulence modeling and solver setting

The Navier-strokes equation is solved by finite volume method, to measure the unsteady flow properties around the turbine blades. Selecting an equitable turbulence model to figure an accurate result depends on the domain, nature of the flow and shape of the model. A fully developed turbulent flow encounters with high velocity fluctuation, infinite degree of freedom, highly nonlinear, three dimensional and riotous [14].So selecting an elegant model reduces computation time and benefits accurate result. Classical turbulence model based on Reynolds Averaged Navier-Stokes (RANS) advisable for 2D flow simulation tabbed for the study are: Standard, RNG, Realizable k-ε turbulence model Standard k-ω turbulence model and SST model. The above discussed model are two equation models except SST model with two PDEs to represent the turbulent kinetic energy and the turbulent dissipation rate with a turbulence intensity of 0.5% [15]. So five set of simulation are formulated and analyzed to resolve the exact model.

Time step sensitive review

Subsequently three case were setup to optimize the sensitivity of time step in which the flow property change with time in a simulation. Time step is more sensitive while solving the governing equation for rotating motion and for turbo machinery [16].Rotating motion hatch unsteady flow and is time periodic, so acceptable time step has to been chosen to get preferred convergence, lesser computational time and good accuracy of results. Taking this in account three different time steps has been inspected $\Delta t = 0.001$, $\Delta t = 0.003$, $\Delta t = 0.005$. And the Max iteration for each time step is set to 20, finally appropriate time step is chosen for getting accurate result and lesser computational resource.

Grid Refinement Survey

Complex aerodynamic flows around the turbine can be solved only with reliable mesh generation which adversely won't affect the factualness of the investigation. Improper grid resolutions leads to numerical error and incorrect results in the simulation [17], so mesh quality definitely has a huge influence on simulation accuracy. Grid dependency test was carried for three different mesh qualities: course with one lakh elements, medium with five lakhs element and fine with one million elements. Y-plus value less than 30 has been adopted for all the study with wall functions. To accurately visualize the wake formulation and flow separation from the turbine blades dense fine unstructured mesh triangular mesh refined on the surface of blades. The region of detect are both the blades and interface were meshed with edge sizing while the outer domain with structured rectangular mesh. Maximum skewness around 80 and aspect ratio around 23 has been maintained in the study.

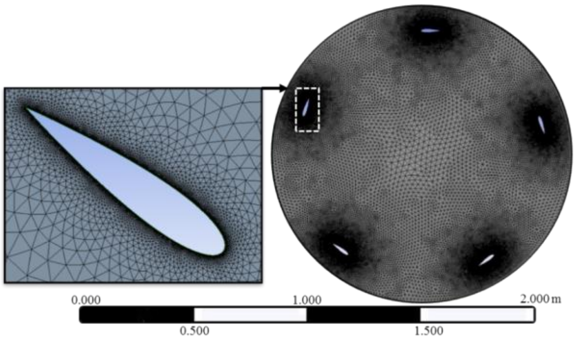

Figure 3. Detailed mesh discretization around the blade and rotor interface

Table 2. Detail of grid elements and simulation time

Refinement	Number of elements	Simulation time
Coarse	169430	6 h and 43min
Medium	510482	13h and 38 min
Fine	1036249	25 h and 54 min

Result and discussion

All the above observations are examined with a speed of U_{∞} = 5m/s and tip speed ratio λ = 3, the reason to pick the particular tip speed is that from several published data's the performance of the turbine is hit its peak at between λ = 3. To interpret the correct selection of grid is a huge obstacle, to point out the ideal mesh quality three different mesh refinements were made namely coarse, medium and fine. Finally the primary goal was to select a pertinent mesh resolution to reduce cost, resource of computation and mainly veracity result. Figure shows torque generated along the unit length of single blade for one complete revolution.

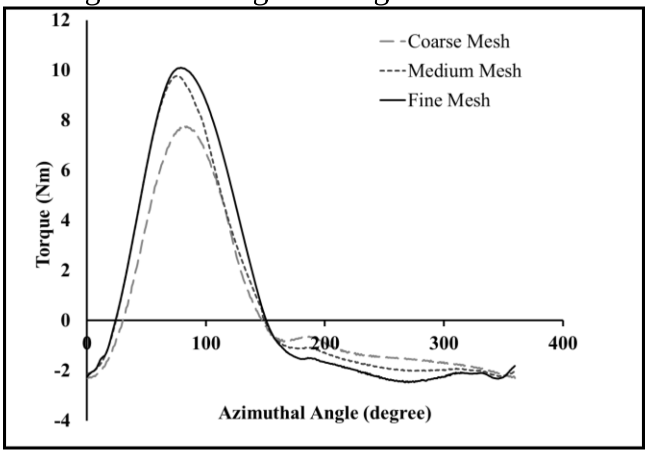

Figure 4. Grid refinement survey for 2D turbine

Varying results obtained from the three different mesh models, comparing all the three meshes both fine and medium mesh show very close results with little difference. The coarse grid provides much vague results which is unsuitable for further simulation. But better arrangement has been seen in both medium and fine mesh form 0 degree to 60 degree later on little dissimilarity subsequently. In general from the both the medium and fine mesh figures the more accurate positive peak torque between 0^{o} to 150^{o}, further negative torque below 150^{o} till 360^{o}. Positive torque falls on the upstream side while negative torque is generated in the downstream. Eventually from all the three mesh study to get an accurate result from a computational analysis reckon on the number of mesh elements. From the overall view the mesh study with one million elements is consider for all the forthcoming simulation as it is accurately predicting the torque despite it takes computational time.

Time step is another variable which may perturb the numerical result for a little extend, so three different time steps were carefully chosen for the time periodic study notable Δt = 0.001s, Δt = 0.003s and Δt = 0.005s. Torque generated from unit length of the blade is observed from each time step as shown in fig 6 along unit length of the blade which points out Δt = 0.001s is suitable for the study as the generated data in that particular time step is more precise. But time step, Δt calculation for turbo machinery and rotating simulations are proposed as [16]

$$\text{Time step, } \Delta t = \frac{1}{10}\frac{\text{Number of blades}}{\text{Rotational Velocity}} \tag{1}$$

The above equation yields a time step off, Δt = 0.033s for a ω = 12rad/s and radius, r = 1.25m. Validations is simulated for the calculated time step which predicts the minimal value of peak torque and has a huge dissimilarity when compared with the other probed time steps. So for better accuracy the time step is set to Δt = 0.001s throughout the entire 2D simulation.

Five different RANS based turbulence modelling were considered in this numerical analysis namely standard k–ε model, Realizable k-epsilon model, RNG k-epsilon model, standard k–

ω modeland SST model. For this case the fine mesh and time step Δt = 0.001s were selected from the above study. Fig 6 shows the torque force generated from a single blade for one rotation for each turbulence models analyzed. From the plot its shows that both standard k–ω model and SST model under predicts the torque in the upstream flow while both Realizable k-epsilon model and RNG k-epsilon model shows a good argument with higher torque generation in the upstream flow, also lesser dissimilarities in the downstream.
From the previous study it shows both realizable k-epsilon model and RNG k-epsilon model predicts more accurate and similar results and also simulates the detailed vortices during dynamic stall[18]. So realizable k-epsilon model has been chosen as the baseline model for all the postliminary simulations. The main prospect in choosing realizable k-epsilon model is that it performs well for large boundary layer separated flows and also for swirl or rotating flow withadverse pressure gradients which can be visualized during turbine operation.

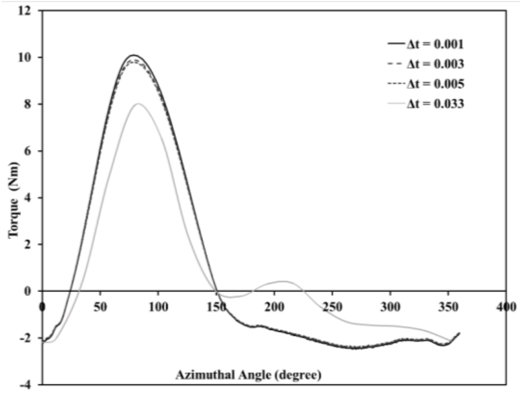

Figure 5. Time step sensitive study for the turbine

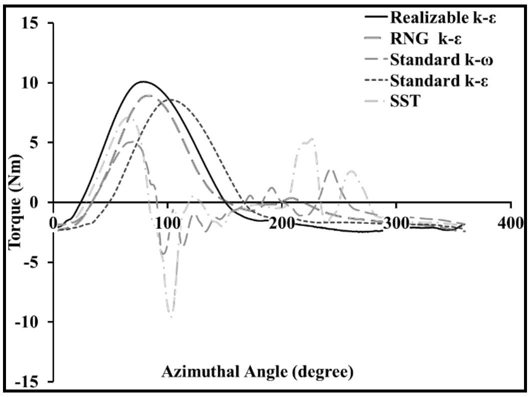

Figure 6. Turbulence modeling study for the turbine

Flow driven Characterization

Acceleration of the turbine about its own axis influenced by the induced air flow is simulated for free load condition considering the moment of inertia of the turbine, I and Aerodynamic moment, C_m generated by the incident air.
From Newton's second law of motion, the revolutionary motion of the flow drive turbine is measured as:

$$I\alpha = C_m \tag{2}$$

Where α is the angular acceleration of the turbine.
From equation (2) the revolutionary motion of the flow drive turbine is calculated as:

$$\omega = \int_0^t \frac{C_m}{I}\, dt \tag{3}$$

Where, ω angular velocity of the turbine
Similarly as the turbine starts to rotate the experience varying azimuthal angle, hence the equation is transformed to

$$\theta = \int_0^t \omega\, dt \tag{4}$$

Hence the above loop the passive rotation of the turbine for free load condition is achieved, Fluid driver structure coupling defines the torque generated by the turbine, calculating the shear stress and the pressure developed by the aerodynamic moment and also constraining the turbine rotational direction.
The torque generated by the turbine under free load condition for all blades and single blade is show in fig 7 and fig 8. The term free load defines the capability of turbine to accelerated about its own axis by the impact of the induced air without any counter torque been supplied [19]. The moment of inertia and mass of aluminum is considered for the study with an

induced flow velocity of U_∞= 5m/s with passive motion. Peak fluctuating torque around 11Nm has been obtained from the turbine during the accelerating period but after t=2s starts the ideal equilibrium period with a peak value of torque around 5Nm. Turbine attains its equilibrium state much prior if the induced flow velocity is higher but at lower wind speed the turbine poses "start and stop loop" because the turbine won't be able to overcome the inertia leads to low self-starting performance of straight bladed turbine [20]. This shows the passive model study accurately predicts the self-start efficiency of the turbine under no-condition.

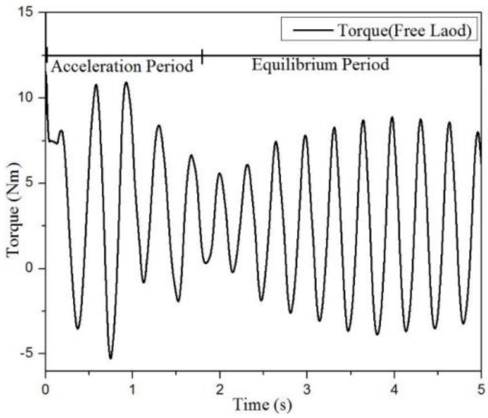

Figure 7. Flow driven torque generated by the turbine

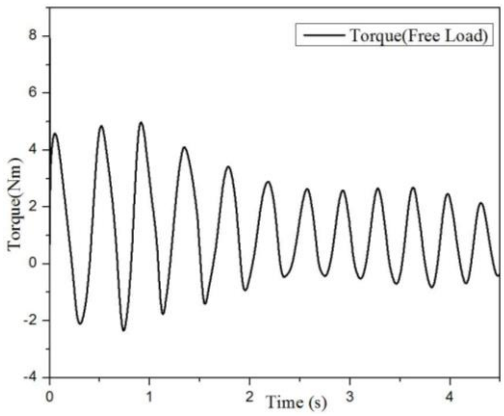

Figure 8. Flow driven torque generated on single blade

CFD validation

Realistic study of straight bladed turbines was compared against the numerical setup, the main purview was justify whether simulation falls in the right direction with that of experimental data. So more admissible empirical study has been considered, experimental setup figured by Qing 'an LI [21] was more promising and most significant for validation. The main benchmarks considered are: Straight five bladed Darrieus turbine, airfoil chord, symmetrical airfoil, diameter and solidity. The following considerations are listed below in the table 3.

Table 3. Comparison between the two turbines

Framework	2D simulation	Experimental
Number of blade	5	5
Airfoil profile	NACA 0018	NACA 0021
Airfoil chord (m)	0.2	0.265
Turbine diameter (m)	2.5	2
Blade pitch angle (β)	0 degree	10 degree
Solidity(σ)	0.8	1.325

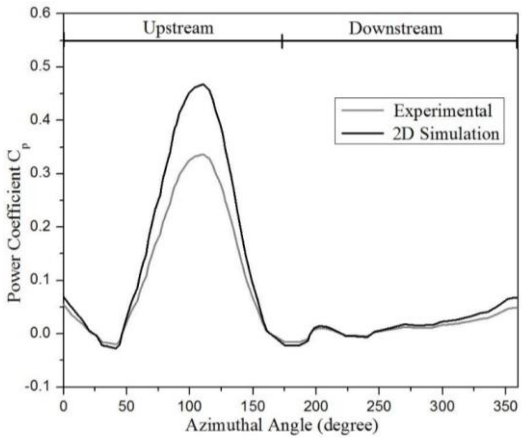

Figure 9. Power coefficient comparison between two turbines

Numerical simulation is performed with the same aerodynamic conditions applied in the experimental set up with a constant wind velocity of 8m/s and tip speed ratio 1.39. The figure 9 shows the validation for coefficient of power (C_p) versus Azimuthal angle for unit length of single blade, Comparison results of both the plot shows a promising trend with a discrepancy factor of 2, with a more similar curve. Both the experiment and numerical results show a maximum coefficient of power (C_p) at $\theta \approx 100^o$. Peak C_p of around 0.45 for the computational simulation and around 0.32 for the experimental study. The slight dissimilarity between the results are due to the effect of blade profile, blade pitch angle and also the effect of wing tip vortices visualized in the experimental study which is not observed in the 2D simulation. All the above numerical simulation is able to predict the physical behavior of the vertical axis wind turbine.

Dynamic stall

Another parameter to look into while investigating unsteady aerodynamic behavior of the turbine is to compute the lift and drag coefficients, as the turbine rotates the flow parameters such as the Reynolds number and angle of attack vary which tend to occur dynamic stall on blade surface with drag and lift hysteresis[16]. The lift and drag coefficients are probed over one rotation of the turbine for five different tip speed ratios from $\lambda = 2$ to $\lambda = 4$ as shown in figure 10 and 11. It figures that the tip speed ratio λ is directly proportional to that of lift and drag coefficients on the airfoil. As the tip speed ratio increases both lift and drag increases durably. Stall effect on the blades are characterized by vortex formation in the leading edge which leads to separation of flow and finally get detached to the trailing edge [22]. This progression generates huge hysteresis of flow with unsteady lift and drag coefficients.

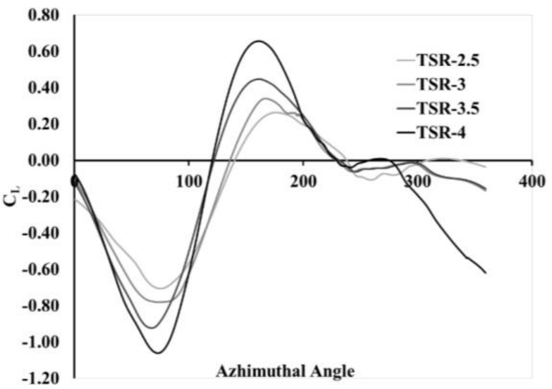

Figure 10. Lift coefficient vs. azimuthal angle for one rotation

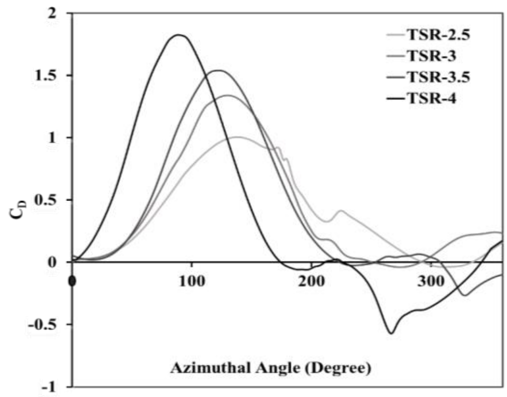

Figure 11. Drag coefficient vs. azimuthal angle for one rotation

Table 4.Stall at different TSR

Λ	$C_{L\ stall}$	$\theta_{\ stall}$
2.5	-0.69	59.956
3	-0.77	60.073
3.5	-0.96	63.438
4	-1.45	74.902

Performance Analysis

The power coefficients computed in this numerical simulation for different tip speed ratio were based on acknowledging the results obtained from the mesh, time step and turbulence modeling research. According to simulation results fine mesh with time step $\Delta t = 0.001s$ and realizable k-epsilon model were considered for the Performance analysis. The power coefficient analysis is formulated for four different tip speed ratio with three different wind velocity U_∞ 4, 6 and 8 (m/s). A total of 12 simulations were carried out with a computational of 3 days for each simulation with 10 interactions loop per time step.

Fig 12 shows the average power coefficient obtained from three different wind flows, all the three curves follows the same trend for the considered wind speeds. The maximum power coefficient is achieved at $\lambda = 3$ for the three different incident wind velocity, with a maximum C_p around 0.3 at 8 m/s. This figures that higher the wind velocity the value of C_p increases for the particular tip speed ratio, further increase in tip speed ratio the graph shows a sudden drift in the C_p and it tend to decrease at $\lambda = 3.5$ and generates negative torque at $\lambda = 4$.

Fig 13 shows average torque coefficient at three different wind velocity as a function of Tip speed ratio. The plot shows a good argument with similar trend for all the three wind velocity, it is seen that as the tip speed increases the turbine achieves its maximum steady peak at $\lambda =$ 3and later on decreases linearly as the tip speed ratio increases. As the wind velocity increases there will be a steady increase in the turbine torque. Even here all the three curves follows the similar trend , coefficient of torque increases with TSR and later decreases after reaching a peak at $\lambda = 3$.

Table 5. Flow conditions for the performance analysis

Velocity U_∞ (m/s)	$\lambda = 2.5$	$\lambda = 3$	$\lambda = 3.5$	$\lambda = 4$
	Angular Velocity (ω) rad/s			
4	8	9.6	11.2	12.8
6	12	14.4	16.8	19.2
8	16	19.2	22.4	25.6

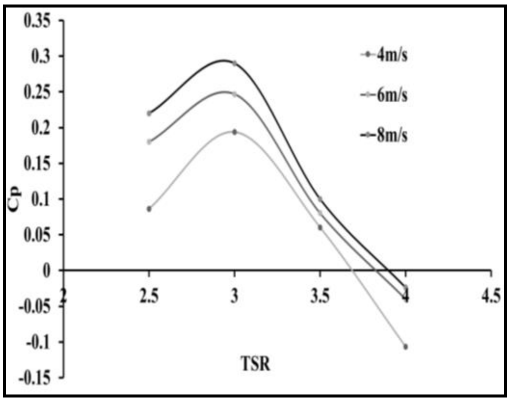

Figure 12.Power coefficient versus TSR at different wind velocity

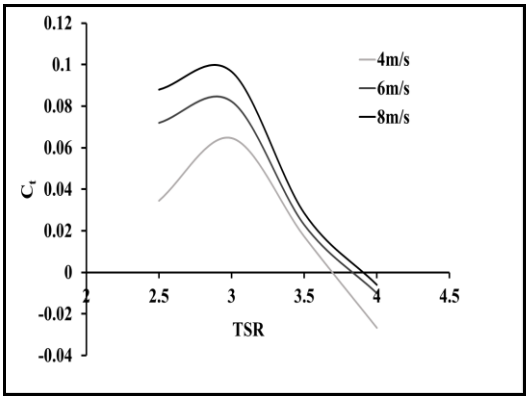

Figure 13. Torque coefficient versus TSR at different wind velocity

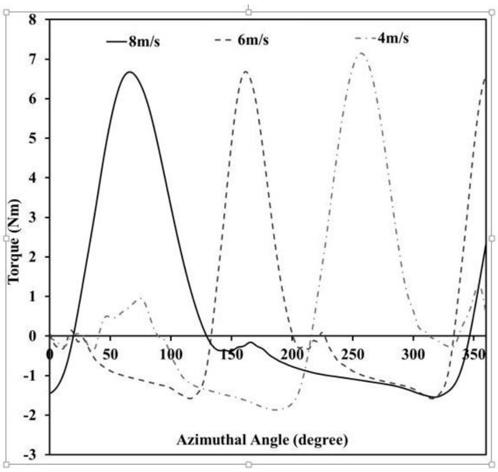

Figure 14. Torque generated at λ = 3 for three different wind speed on single revolution

Fig. 14 shows the torque obtained at λ = 3 for one complete revolution, as the velocity increase positive peak torque is achieved earlier. At 4 m/s the positive peak is obtained around 250^o but as the velocity increases the peak torque is shifted more forward so at 8m/s peak torque is obtained around 70^o. On an average the torque falls in the positive note also it states that the turbine won't be able to self-start at minimum velocity of airflow and also at certain azimuthal angle. As we have seen from the plot negative torques falls at lower induced wind flow and also minimum torque at certain azimuthal angle. Overall all the three curves follows similar trend with slight variation.

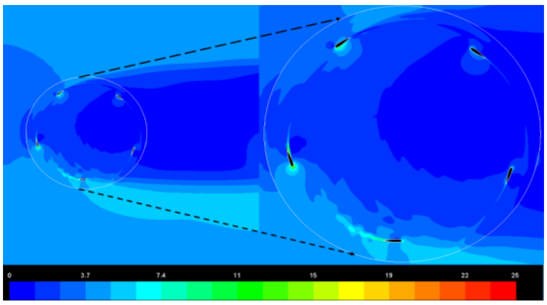

Figure 15. Velocity plot for TSR = 3 at 4m/s

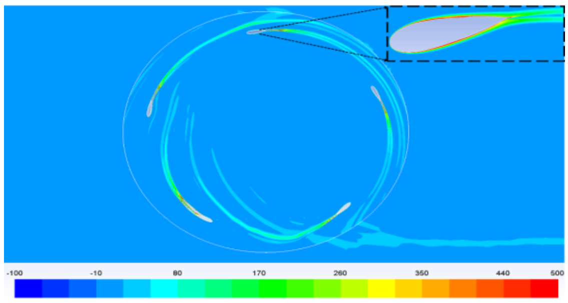

Figure 16. Vorticity magnitude for TSR = 3 at 4m/s

Conclusion

In this research a detailed examination is carried out to simulate the unsteady flow around a rotatory turbine. Characterization is performed in such a way is to select a reasonable grid resolution, convenient time step and favorable turbulence modeling for a sequential numerical simulation in order to conclude with accurate result. Aerodynamic coefficients generated in the simulation of the rotating turbine relay on the validated turbulence modeling and fine mesh resolution. Irregularities in probing the aerodynamic coefficients cause the simulation with weaken peak torque and invalid performance result. All these combined parameters predict that the wind energy harvested by the turbine lay on the upstream with net negative torque in the downstream. This also leads to the low self-staring behavior of the turbine as it poses gradual fluctuation in net torque and the turbine has to overcome the mass and inertia force acts on it. CFD validations is performed against a five bladed turbine with similar configuration shows a similar trend in result. Form the performance point of view maximum

Cp and torque is generated at higher wind speed and also at higher tip speed ratio and minimum power at low TSR which again proves that the turbine is not a self-starting.

References

[1] Wilson, R.E., (1980) Wind-turbine aerodynamics, *Journal of Wind Engineering and Industrial Aerodynamics*, *5*(3-4), pp.357-372.

[2] Dumitrescu, h. And cardos, v., (1981) A free wake method for vertical-axis wind turbine performance prediction, *Romanian journal of technical sciences-applied mechanics*, 1(54), pp.87-100.

[3] Lauffer, J.P., Carne, T.G. and Nord, A.R., (1984) Mini-modal testing of wind turbines using novel excitation No. SAND-84-0984C, CONF-840178-1, Sandia *National Labs*, Albuquerque, NM (USA).

[4] Amet, E., Maaztre, T., Pellone, C. and Achard, J.L., (2009) 2D numerical simulations of blade-vortex interaction in a Darrieus turbine, *Journal of fluids engineering*, 131(11), p.111103.

[5] Österberg, David (2010) Multi-Body Unsteady Aerodynamics in 2D Applied to a Vertical-Axis Wind Turbine Using a Vortex Method.

[6] Lain, S., and C. Osorio. "Simulation and evaluation of a straight-bladed Darrieus-type cross flow marine turbine." (2010).

[7] Edwards, J.M., Danao, L.A. and Howell, R.J., (2012) Novel experimental power curve determination and computational methods for the performance analysis of vertical axis wind turbines, *Journal of Solar Energy Engineering*, 134(3), p.031008.

[8] Castelli, M.R., De Betta, S. and Benini, E., (2012) Effect of blade number on a straight-bladed vertical-axis Darreius wind turbine, *World Academy of Science*, Engineering and Technology, 61, pp.305-3011.

[9] Zhu, Jianyang, Hailin Huang, and Hao Shen, (2015) Self-starting aerodynamics analysis of vertical axis wind turbine, *Advances in Mechanical Engineering*, 7.12: 1687814015620968.

[10] Gang, D. and Kau, W.C., (2015) Unsteady Flow Numerical Simulation of Vertical Axis Wind Turbine, *Procedia Engineering*, 99, pp.734-740.

[11] M.S.U Khalid, T.Rabbani, I Akhtar, N.Durrani, and M.S Siddiqui, (2015) Reduced-Order Modeling of Torque on a Vertical-Axis Wind Turbine at Varying Tip Speed Ratios, *J. Comput. Nonlinear. Dy*., 10, no.4, p.041012.

[12] Sánchez-Caja, A., Rautaheimo, P. and Siikonen, T., (1999) Computation of the incompressible viscous flow around a tractor thruster using a sliding-mesh technique, *In Proceedings of the 7th International Conference on Numerical Ship Hydrodynamics*, France.

[13] Fluent, A., (2009) *12.0 User's guide*, User Inputs for Porous Media, 6.

[14] Tony Saad, T., (2011) *Turbulence modeling for beginners*, University of Tennessee space institute.

[15] Wilcox, D.C., (1998) *Turbulence Modelling for CFD DCW Industries Inc*, La Canada, California.

[16] Alaimo, A., Esposito, A., Messineo, A., Orlando, C. and Tumino, D., (2015) 3D CFD analysis of a vertical axis wind turbine. Energies, 8(4), pp.3013-3033.

[17] Venditti, D.A. and Darmofal, D.L., (2003) Anisotropic grid adaptation for functional outputs: application to two-dimensional viscous flows. *Journal of Computational Physics*, 187(1), pp.22-46.

[18] Durrani, N., Hameed, H., Rahman, H. and Chaudhry, S., (2011) A Detailed Aerodynamic Design and Analysis of a 2-D Vertical Axis Wind Turbine Using Sliding Mesh in CFD, *In 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition (p. 541)*.

[19] Le, T.Q., Lee, K.S., Park, J.S. and Ko, J.H., (2014) Flow-driven rotor simulation of vertical axis tidal turbines: A comparison of helical and straight blades, *International Journal of Naval Architecture and Ocean Engineering*, 6(2), pp.257-268.

[20] Liu, Z., Qu, H. and Shi, H., (2016) Numerical Study on Self-Starting Performance of Darrieus Vertical Axis Turbine for Tidal Stream Energy Conversion. *Energies*, 9(10), p.789.

[21] Qing'an LI, Furukawa, K. and Yamamoto, M., (2014) Aerodynamic Models and Wind Tunnel for Straight-bladed Vertical Axis Wind Turbines, *IOSR J Eng*, 4(06), pp.35-44.

[22] Nobile, R., Vahdati, M., Barlow, J. and Mewburn-Crook, A., (2011) November. Dynamic stall for a vertical axis wind turbine in a two-dimensional study, *World Renewable Energy Congress-Sweden*, Linköping, Sweden No. 57, pp. 4225-4232.