Numerical Simulation of a Vertical Axis Wind Turbine Having Cavity Vanes

Kadhim H. Suffer[a], Ryspek Usubamatov[b],
[a,b,d]*School of Manufacturing Engineering*
University Malaysia Perlis, Perlis, Malaysia
[a]kadhim_askar@yahoo.com
[b]ryspek@unimap.edu.my

G. A. Quadir[c], Khairul Ismail[d]
[c]*School of Mechatronic Engineering*
University Malaysia Perlis, Perlis, Malaysia
[c]gaquadir@unimap.edu.my
[d] k.azwan@unimap.edu.my

***Abstract*— It is well known that Vertical Axis Wind Turbines (VAWTs) are more suitable for urban areas than Horizontal Axis Wind Turbines (HAWTs). However, very little has been published in this area to assess good their performance and sustainability either for open or urban areas. The main goal of this research is to investigate numerically the aerodynamic performance of a vertical axis wind turbine having cavity vanes. In this design, the power generated depends on the drag force generated by the individual blades and interactions between them in a rotating configuration. For numerical investigation, commercially available computational fluid dynamics CFD software GAMBIT and FLUENT were used. The Shear Stress Transport (SST) *k-ω* turbulence model used which is considered to be better than other turbulence models available as suggested by some researchers. The computed results show good agreement with published results.**

Keywords--wind energy; vertical axis wind turbine; computational fluid dynamic; performance analysis; turbulent model.

I. INTRODUCTION

Energy is the key to modern life and provides the basis necessary for sustained economic development. With the continued rapid development of economy, the request for energy increased, energy problem in many countries have become more and more critical. The depletion of fossil energy resources and global warming trends has led to the recognition of a low carbon economy as an international strategy for sustainable development. The most environmental friendly types of energy are wind power, solar power, wave power and marine current power. Among several green and renewable energy sources, Wind power is one of the alternative energies because of cleanliness and emissions-free power generation technology. Wind turbines are typical devices that convert the kinetic energy of wind into electricity.

At present, there are two categories of modern wind turbines, namely Horizontal Axis Wind Turbine (HAWT) and Vertical Axis Wind Turbine (VAWT), which are used mainly for electricity generation [1]. There are many advantages of using VAWT. It is ability to accept wind from any direction without yawing, ability to provide direct rotary drive to a fixed load can capture ground level winds and its components (generator and gear box) can be mounted at ground level. VAWT is further divided into lift driven VAWT (Darrieus type) and drag driven VAWT (Savonius type). Darrieus type VAWT proves to be more efficient than Savonius type turbine. The theoretical maximum power coefficient of wind turbine of any design operating in an open atmosphere is lower than 59.3% [2], which is known as the Betz limit. It is reached by setting up the ratio between the maximum power extracted from the turbine rotor and the energy in the wind. However, the real word limit is well below the Betz limit with values of 0.35-0.45 for the best designed wind turbine.

Designing a wind turbine system that can generate power with high efficiency requires a thorough understanding of the principles of aerodynamics and structural dynamics of the rotor system. Simple analysis of the available wind turbine designs shows that the designs are not perfect, and the wind force is not harnessed in full-scale due to geometrical and technical problems. Therefore the study of wind turbine has been of interest to many researchers. Researchers are working to improve the aerodynamic characteristics of VAWT which include laboratory measurement to full scale numerical simulation and theoretical prediction for flow around the turbine.

Computational fluid dynamics (CFD), as a promising technique in wind engineering research, has attracted growing interest in the past two decades. Compared with the conventional experimental approaches, CFD could provide more details about flow field without any need for complicated control and measurement systems. It also provides an inexpensive solution to performing systematic analyses, such as the parametric studies required for performance optimization of VAWTs. Subject to the constraint of computational resources and time, CFD is a very powerful tool to solve a wide range of industrial and non-industrial fluid flow problems. Physical characteristics of the fluid motion around the rotor could be described through the fundamental flow governing equations, usually in the partial differential forms. In order to solve the governing equations, high level computer programming languages or commercial available CFD software's are used [3]. Where the finite difference, finite element and finite volume methods where used. Fundamentally, the finite volume method is originated from the special finite difference formulation. It is the formal integration of the governing equations of the fluid flow over all the (finite) control volumes of the computational domain, discretization of the terms in the integrated equation representing the flow over the Turbine rotor. This converts the integral equations into a system of algebraic equations. Finally, the algebraic equations are solved by different iterative methods. ANSYS-Fluent and ANSYS-CFX are the widely accepted finite volume software packages to solve complex flow phenomena [4-6].

Kacprzak et al. [7] examined the performance of a Savonius wind turbine with constant cross-sections by means of quasi 2D flow predictions executed in ANSYS CFX. They simulated the Classical and so-called Bach-type Savonius rotor. Comparison revealed the importance of applying a Laminar-turbulent transition model. Additionally, turbine design with an Elliptical blade shape was modeled. All mentioned designs were analyzed at different values of tip speed ratio, and the results were presented in terms of coefficients of power, coefficients of torque and torque variation with the angle of incidence. The wake is observed and examined by means of FFT analysis of pressure fluctuations at the point located downstreamof the rotor.

Nobile et al. [8] studied two-dimensional unsteady flow simulations of a vertical axis wind turbine in their CFD analysis with different mesh resolution, turbulence model and time step size. They found that the mesh resolution and the turbulence model affect the result accuracy while the time step size examined has small impact on the numerical results.

Morshed et al. [9] carried out feasibility studies on conventional and modified vertical-axis Savonius wind turbine feasibility improved by, first performed a series of wind tunnel investigations on semi-cylindrical three-bladed Savonius scale models with different overlap ratios and also without overlap at different Reynolds numbers. They also performed computational fluid dynamics (CFD) simulations using GAMBIT and FLUENT software's to analyze the static rotor aerodynamics of those models. The results from the experimental part of their research show a significant effect of overlap ratio and Reynolds number on the improvement of aerodynamic performance of the Savonius wind turbine.

Biadgo et al. [10] studied the numerical and analytical investigations of VAWT to highlight the progress made in the development of aerodynamic models for VAWT with particular emphasis on streamtube approach to accurately predict the efficiency. Numerical and analytical investigations are conducted on straight blade fixed pitch VAWT using NACA0012 airfoil as a blade profile to assess its performance. Numerical simulation is done for two-dimensional unsteady flow around the same VAWT model using ANSYS FLUENT by solving Reynolds averaged Navier-Stokes equations. Comparison of the analytical results using double multiple streamtube (DMST) model with computational fluid dynamics (CFD) simulation has been done. Both the CFD and DMST results have shown minimum and/or negative torque at lower tip speed ratios for the modeled turbine, which implies the inability of NACA0012 to self-start.

Widodo et al. [11] designed and analyzed the Savonius rotor blade to generate 5 kW power output. The Savonius rotor was designed with the rotor diameter of 3.5 m and the rotor height of 7 m. The 3D model of Savonius rotor blade was created by using SolidWorks software. Computational Fluid Dynamics (CFD) analysis and structural Finite Element Analysis (FEA) were carried out. CFD analysis was performed to obtain the pressure difference between concave and convex region of the blade while FEA was done to obtain the structural response of the blade due to the wind load applied in terms of stresses and its displacements.

McLaren [12] reported on a numerical and experimental investigation of the unsteady loading on a small scale, high solidity, H-type Darrieus vertical axis wind turbine considering two-dimensional, unsteady Reynolds averaged Navier-Stokes simulations. They found the dominant effect of dynamic stall on the power production and vibration excitation of the turbine. To validate the numerical model, a series of full-scale experimental wind tunnel tests were performed to determine the aerodynamic loading on the turbine airfoils, vibration response behavior, and wake velocity. Comparison of the two-dimensional numerical model results to the experimental measurements revealed a considerable over-prediction of the turbine aerodynamic force and power coefficients, and wake velocity. In doing so, the two-dimensional numerical model results could be properly scaled to represent the three-dimensional flow behavior of the turbine prototype. Ultimately, a validated VAWT design tool was developed by them.

Arturo et al. [13] presented an experimental and numerical aerodynamic analysis of Savonius VAWT using the Dynamic Mesh Method (DMM) which allows moving bodies and adjusts the mesh according to the loads acting on the surface of the body. They also used Fluent® to simulate unsteady two-dimensional computational model**.**

Gupta et al. [14] studied CFD analysis of a two- bucket Savonius rotor for various overlap conditions. The analysis was carried out to model the complex flow physics around the rotating rotor. For this purpose, data were taken from the experiments conducted earlier on the rotor in a subsonic wind tunnel for five overlap conditions, namely 16.2%, 20%, 25%, 30% & 35%. The pressure drop across the rotor from upstream to downstream side was the maximum in case of 16.2% overlap indicating maximum power extraction from the wind at that overlap condition. Vortices formed on the concave side of the advancing bucket due to flow separations from adverse pressure gradient were less for overlaps up to 16.2%, which increased with higher overlaps leading to reduction in positive torque by the advancing bucket. Moreover, for 16.2% overlap, maximum pressure difference was obtained across the returning bucket, thereby increasing the overall aerodynamic torque and power for the rotor.

Cao [15] presented aerodynamics analysis of small horizontal axis wind turbine blades of 2D and 3D models using ANSYS-FLUENT software. He used the Spalart-Allmaras turbulent viscosity and calculated the dimensionless lift, drag and pitching moment coefficients for wind-turbine blade at different angles of attack. These CFD model values were then validated using published calibrated lift and drag coefficients available in the literature.

Alessandro et al. [16] developed an unsteady aerodynamics analysis of a Savonius wind rotor using the Sliding Mesh Method (SMM) where the Navier-Stokes equations were solved using the finite volume code FLUENT®, The solid body motion was treated to solve the second cardinal equation of the dynamics by means of a custom Matlab® numerical algorithm able to import CFD data, calculate the angular velocity and export this variable as an input to the CFD code. Their work is one of the newest numerical approaches developed to predict the aerodynamic performance of VAWT and reproduce, many characteristics of the complex phenomena of the Savonius rotor.

Rolland et al. [17] developed a CFD based computational model to analyze the aerodynamic performance of their novel designed vertical axis wind turbine (VAWT) and compared with the experiment work, They investigated the extent to which a CFD model employing the simplest turbulence representation can provide a useful input to evaluate the impact of several key operational parameters: wind speed, rotor speed, yaw angle and blade pitch angle. A sliding mesh technique was implemented in the CFD code. This enabled a geometrically exact simulation of the rotor motion with time. The CFD model reflects the experimental data well overall and numerical values are shown to give very good correlation within the working range of each studied parameter with respect to over-all power output and also the pressure distribution. The authors however, acknowledge they need of further refinement on turbulence modeling for wide range of application.

Sukanta et al. [18] reported a review of various computational methods addressing the influence of various operating parameters and also discussed different augmentation techniques. They observed that with the selection of a proper computational methodology, the design, performance, and efficiency of a Savonius rotor can be enhanced significantly.

Nathan et al. [19] highlighted their design and analysis work performed to increase the aerodynamic efficiency of a unique and patented VAWT in order to optimize it for implementation in remote rural villages. The design goal was to develop arms with aerodynamic properties that complemented the function of the blades at the appropriate phases of a single revolution.

Huimin et al. [20] reported the numerical simulation for the aerodynamic performance of VAWT at different wind velocities with two dimensional unsteady flow field of the vertical axis wind turbine using Reynolds average Navier-Stokes equations and Realizable k-ε model. Their results showed that: (1) the velocity in the region of wind turbine´s rotation was much larger than that of upstream. There is a wake dispersion region in the downstream of the wind turbine, and the length of the wake dispersion region increased with the increase of the wind velocity; (2) the eddy is much larger in the upper blade´s back of the wind turbine´s rotational part, while it is less in the lower blade´s back of the wind turbine´s rotational part; (3) At the same rotational speed, the condition of lower wind velocity has larger total torque coefficient. With the increase of the wind velocity, the variation of the wind turbine´s total torque coefficient tends to smooth. The calculated results pointed out the direction for the follow-up study.

Konrad et al. [21] studied the performance of the Savonius wind turbine with constant cross-sections by means of quasi 2D flow. They numerically analysed three geometries of Savonius wind turbine rotors, namely Classical, Bach-type and Elliptical designs. They presented the results in terms of coefficients of power, coefficients of torque and torque variation with the variation of angle of incidence. The predictions were done using ANSYS CFX. Simulations were performed in a way that allowed comparison with wind tunnel data [19-21], where two designs were simulated. Their comparison revealed the importance of applying a laminar turbulent transition model. The most characteristic flow structures were identified and compared. Finally, the wake was observed and examined.

II. PRESENT SIMULATION

The main goal of this research is to carry out numerical simulation of the vertical axis wind turbine with cavity vanes to investigate its aerodynamic performance. The present design of vertical axis turbine consists of three frames with angles 120° between each one. Solidworks 2013 software is used to design turbine geometry as shown in Fig. 1. Three frames have cavity shaped vanes that increase drag force substantially. For numerical investigation, CFD software GAMBIT is used for meshing geometry and specifying boundary conditions as shown in Fig. 2, whereas ANSYS FLUENT 14.5 software is used to solve Navier-Stokes equations using finite volume method.

The Shear Stress Transport (SST) k-ω turbulence model is used in the analysis mainly because it is considered to be better than the other turbulence models as reported earlier [19- 21]. The numerical simulations were carried out for fixed blade position representing its five angular positions. The results obtained from numerical simulation are compared with data available in the literature.

III. RESULTS AND DISCUSSION

The inlet was defined as a velocity inlet of 20 m/s inflow velocity while the outlet was set as a pressure outlet, keeping the pressure constant. The no slip shear condition was applied on the turbine blade, which sets the relative velocity of blades to zero. The residual plot is shown in Fig. 4, which has very small spikes in turbulent kinetic energy and specific dissipation rate for the residual being set as 10^{-5}.

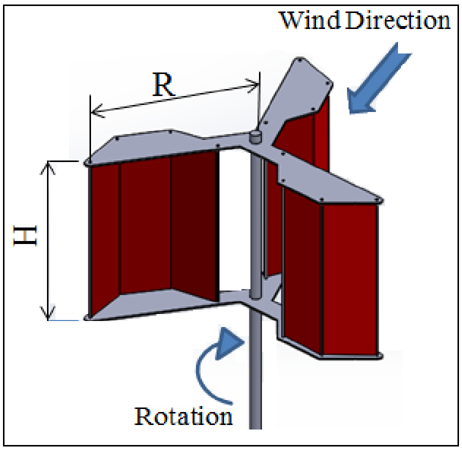

Figure 1. Three blade newly designed vane type vertical axis wind turbine geometry.

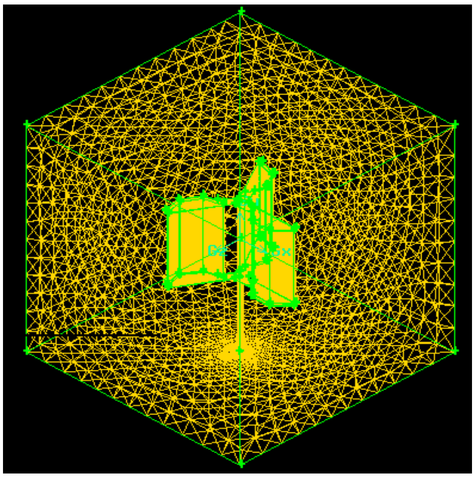

Figure 2. Mesh for the three blades newly designed vane type vertical axis wind turbine geometry.

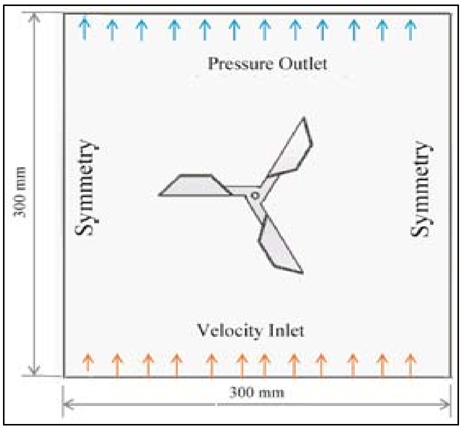

Figure 3. Top view of computational domain with boundary conditions.

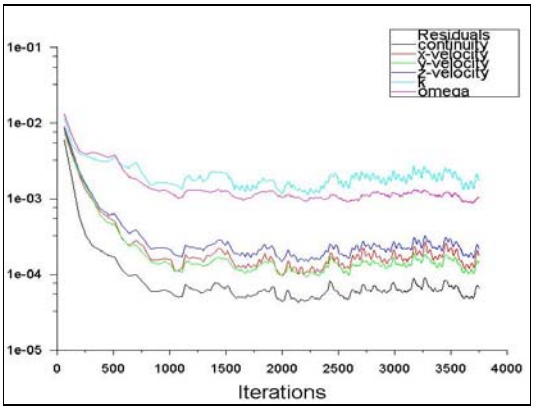

Figure 4. Scaled Residual.

Fig.5 shows the variation of predicted drag coefficient (Cd) with blade angular positions. It is evident from this figure that Cd is maximum at zero blade angle, minimum at 45° blade angle and again maximum at 120° blade angle. At 120° blade angle, actually the next blade takes the position of the first blade because the angle between the turbine blades is 120°.

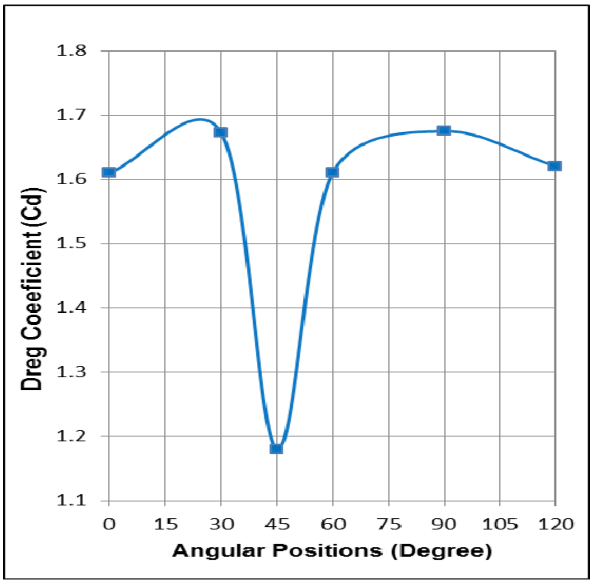

Figure 5. Relation between drag coefficient and first blade angular positions

The static pressure and velocity contours for the three blade rotor for different blade angular positions are shown in Figs. 6 and 7 respectively. Plots show the variations in velocity and pressure in various regions near the blade within the flow domain. It can be observed from the pressure contour plots that pressure drop occur across the rotor from upstream to downstream side. This pressure drop indicates power extracted by the rotor causing its rotation [22].

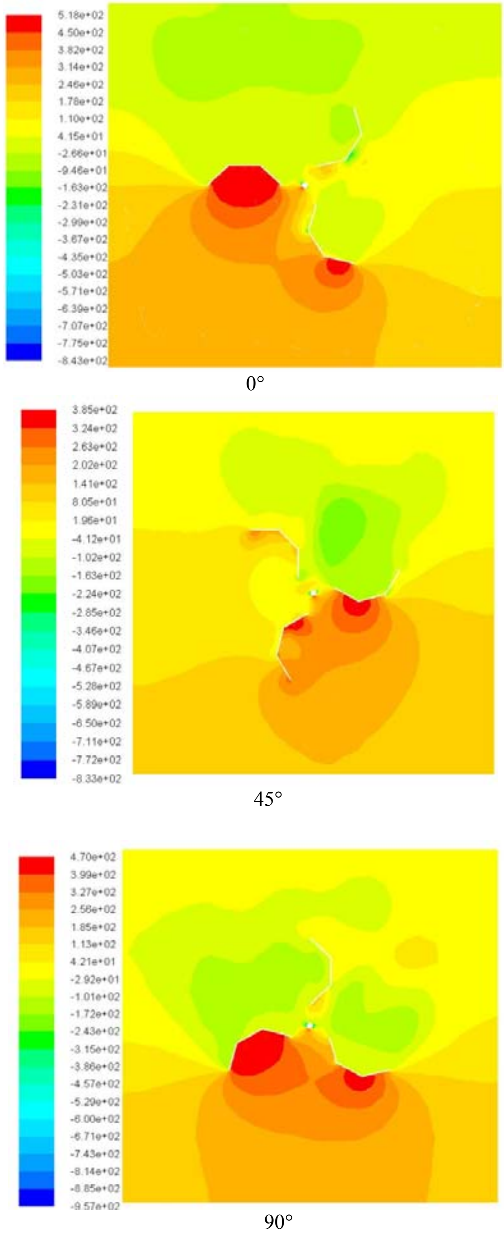

Figure 6. Contour of static pressure distribution in and around turbine blade at different angular positions.

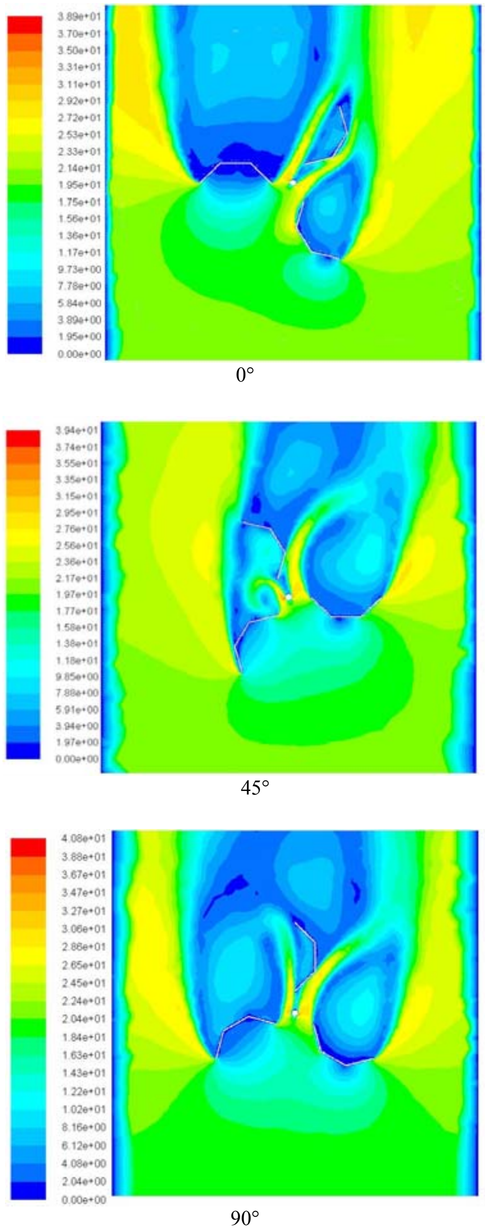

Figure 7. Contour of velocity distribution in and around turbine blade at different angular positions.

The maximum and minimum static pressure drops are found to be 14.27e+02 Pa (for the blade angular position of 90°) and 6.08e+02 Pa (for the blade angular position of 45°), respectively as shown in Fig. 6. The positive pressure is noticed on the frontal surface of the blades normal to the direction of air flow while the negative pressure is observed on the other sides (the back surface for the blades) as shown clearly in Fig. 6. This occurs due to the high flow velocity over the front side of the blade. As a result, a pressure difference acts across the two sides of blade, which provide the necessary torque for its rotation. Fig. 7 shows the velocity in the region of wind turbine´s rotor which is much larger than the upstream air flow due to the flow field distribution at the same wind velocity. However, the velocity of wake away from the rotor is smaller than the velocity of upstream air flow. This phenomenon is mainly caused by the existence of gradient between the wake velocity and the velocity of downstream free air flow. Then shear turbulence will occur between the two flows, and momentum will exchange between them. The above results agree well with experimental published results [14- 18].

Fig. 8 shows the plot of static pressure contours on the surface of turbine rotor blades. It can be observed from this figure that the maximum static pressure is $5.18e^{+02}$ Pa which occurs on the surface of the turbine rotor normal to air flow direction.

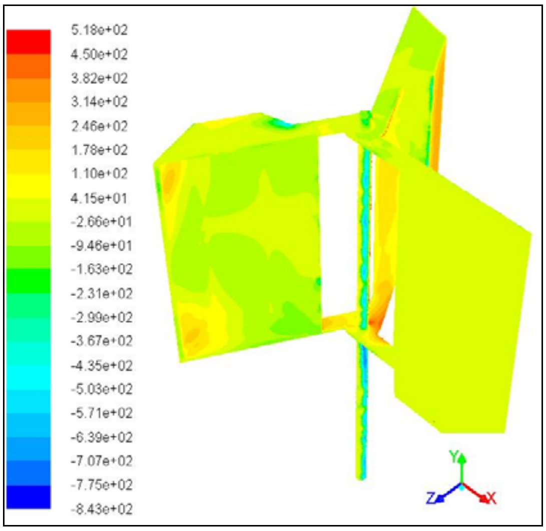

Figure 8. Contour of static pressure distribution on three blades turbine surfaces.

IV. CONCLUSSIONS

The three dimensional numerical investigation of the vertical axis wind turbine having cavity vanes is carried out using CFD softwares GAMBIT and ANSYS FLUENT. Shear Stress Transport (SST) k-ω turbulence model is used to predict the aerodynamics of the turbine. The flow field is simulated numerically at a fixed wind velocity. The predicted results show that: (1) the drag coefficient increases with the increase in turbine frontal area and decreases with the decrease in its frontal area; (2) the maximum and minimum static pressure drops are found to be 14.27e+02 Pa (in the case of blade angular position of 90°) and the 6.08e+02 Pa (when the blade angular position is 45°) respectively; (3) the maximum static pressure of 5.18e+02 Pa is found on the blade surface which is normal to the direction of air flow; (4) the velocity in the region of wind turbine´s rotor is much larger than that of the upstream air flow. There is a wake dispersion region in the downstream of the wind turbine. The predicted results agree well with published results.

REFERENCES

[1] Izli, N. Vardar and F. Kurtulmn, “A Study on Aerodynamic Properties of Nome NACA Profiles Used on Wind Turbine Blades,” Applied Science, vol. 7, 2007, pp. 426-433.

[2] S. Mathew, “Wind Energy Fundamentals Resource,” Springer, Analysis and Economic, 1st ed., 2006.

[3] J. Tu, G. Yeoh, C. Liu, “Computational fluid dynamics,” A practical Approach. India: Elsevier India Private Limited; 978-81-312-16 85-9.

[4] M. Mohamed, G. Janiga, E. Pap, D. Thevenin, “Optimal Blade Shape of a Modified Savonius Turbine Using an Obstacle Shielding the Returning Blade,” Energy Conversion and Management, vol. 52, 2011, pp. 236 – 242.

[5] K. Pope, I. Dincer, G. Naterer, “Energy and Exergy Efficiency Comparison of Horizontal and Vertical Axis Wind Turbines,” Renewable Energy, vol. 35, 2010, pp. 2102 –2113 .

[6] I. Dobrev, F. Massouh, “CFD and PIV Investigation of Unsteady Flow Throughb Savonius Wind Turbine,” Energy Procedia, vol. 6, 2011, pp. 711 – 720.

[7] K. Kacprzak, G. Liskiewicz and K. Sobczak, “Numerical Investigation of Conventional and Modified Savonius Wind Turbines,” Renewable energy, vol. 60, 2013, pp. 578-585.

[8] R. Nobile1, M. Vahdati, J.F. Barlow and A. M. Crook, “Unsteady flow Simulation of a Vertical Axis Wind Turbine: a Two-Dimensional Study,” Proc. EngD Conference, 2nd July, 2013, pp. 1-10.

[9] K. N. Morshed, M. Rahman, G. M. and M. Ahmed, Wind Tunnel Testing and Numerical Simulation on Aerodynamic Performance of a Three Bladed Savonius Wind Turbine,” International Journal of Energy and Environmental Engineering, 2013, pp. 4-18.

[10] A. M. Biadgo, A. Simonovic, D. Komarov and S. Stupar, “Numerical and Analytical Investigation of Vertical Axis Wind Turbine,” FME Transactions, vol. 41, 2013, pp 49-58.

[11] W. S. Widodo, A. C. Chin, H. Sihombing and M. Y. YUHAZRI, “Design and Analysis of 5 kW Savonius Rotor Blade,” Global Engineers and TechnologistsReview, Vol 2, 2012, 1-8.

[12] K. W. McLaren, “A Numerical and Experimental Study of Unsteady Loading of High Solidity Vertical Axis Wind Turbines,” Ph.D. Thesis, McMaster University, 2010.

[13] A. A. J. Arturo, G. Muñoz. and B. F. Manuel, “Experimental and numerical analysis of vertical axial wind turbine applying the dynamic mesh method”, ASME 5t International Conference on energy sustainability, August, 2011, Washington, DC, USA, pp 7-10.

[14] R. Gupta, S. S. Deka, R. Gautam and R. Das, “CFD Analysis of a two- bucket Savonius rotor for various overlap condition,” Proc. ASME 5th International Conference on Energy Sustainablity, August 7-10, 2011, Washington, USA., Pp. 7-10.

[15] H. Cao, “Aerodynamics Analysis of Small Horizontal Axis Wind Turbine Blades by Using 2D and 3D CFD Modeling,” M.Sc. Thesis, School of Computing, Engineering and Physical Sciences, University of the Central Lancashire, Preston, England, 2011.

[16] V. D’Alessandro, S. Montelpare, R. Ricci, A. Secchiaroli, “Unsteady Aerodynamics of Savonius Wind Rotor: A New Computational Approach for the Simulation of Energy Performance,” Energy, vol. 35, 2010, pp. 3349-3363.

[17] S. Rolland, W. Newton, A.J. Williams, T.N. Croft, D.T. Gethin, M. Cross, “Simulations Technique for the Design of Vertical Axis Wind Turbine Device with Experimental Validation,” Applied Energy, vol. 111, 2013, pp. 1195–1203.

[18] R. Sukanta, K. Ujjwal Saha, “Review on the Numerical Investigations into the Design and Development of Savonius Wind Rotors,” Renewable and Sustainable Energy Reviews, vol. 24, 2013, pp. 73 – 83.

[19] E. Nathan Fuller, M. David Wiens, L. Allison Johnston, and J. Jesse French, “Design and Analysis of the Aerodynamic Components for Low Altitude Implementation in Remote Rural Villages, Proc. ASME 2011 5th International Conference on Energy Sustainability, August, 2011, Washington, DC, USA, pp. 7-10.

[20] H. Wanga, J. Wanga, J. Yaoa, W. Yuanb, L. Caoa, “Analysis on the Aerodynamic Performance of Vertical Axis Wind Turbine Subjected to the Change of Wind Velocity,” Procedia Engineering, vol. 31, 2012, pp. 213 – 219.

[21] K. Konrad, L. Grzegorz, S. Krzysztof, “Numerical investigation of conventional and modified Savonius wind Turbines,” Renewable Energy vol. 60, 2013, pp. 578-585.

[22] N. Fujisawa, F. Gotoh, “Pressure measurements and flow visualization study of a Savonius rotor,” Wind Engineering and Aerodynamic, vol. 39, 1992, pp. 60.