

Numerical Investigation of the Aerodynamics of Three Blades Vertical Axis Wind Turbine having Movable Vanes

Kadhim H. Suffer^{1, a}, Ryspek. U.^{2, b}

^{1, 2, 4}School of Manufacturing Engineering

^{1, 2}University Malaysia Perlis (UniMAP), Perlis, Malaysia

^akadhim_askar@yahoo.com

^bryspek@unimap.edu.my

Ghulam A. Quadir^{3, c}, Khairul A. Ismail^{4, d}

³School of Mechatronic Engineering

^{3, 4}University Malaysia Perlis (UniMAP), Perlis, Malaysia

^cgaquadir@unimap.edu.my

^dk.azwan@unimap.edu.my

Abstract—The analysis of the aerodynamics for the vane type vertical axis wind turbine (VVAWT) with different blades and vanes position is carried out using computational fluid dynamics (CFD). In the present design, there are three blades each having three movable vanes. 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 dynamic CFD software GAMBIT and FLUENT are used. In this numerical analysis the Shear Stress Transport (SST) $k-\omega$ turbulence model is used which is better than the other turbulence models available as suggested by some researchers. The computed results show good agreement with published experimental results.

Keywords—wind energy; vertical axis wind turbine; computational fluid dynamics; performance analysis; $k-\omega$ turbulence model

I. INTRODUCTION

Energy is the key to modern life and provides the basis necessary for sustained economic development. The emission of CO₂ into the atmosphere contributes to the global climate change, particularly global warming. To tackle this problem, many of the developed nations are continuing to invest heavily in renewable energy sources. The future will be a mix of energy technologies with renewable sources such as solar, wind and biomass playing an increasingly important role in the new global energy economy [1]. Wind turbine is a machine for converting the kinetic energy available in the wind into mechanical energy. Modern wind turbines are divided into two main types based on their rotational axis: the horizontal axis wind turbine (HAWT) and the vertical axis wind turbine (VAWT). The advantages of VAWT are 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. The technical characteristics of known designs of wind turbines show that there is a necessity to enhancement the out power of wind to be harnessed to the maximum, and with the ability to be used in wide areas of application. The design of VAWT blades, to achieve

satisfactory level of performance; starts with knowledge of the aerodynamic forces acting on the blades. In this article, the of a Vane type Vertical Axis Wind Turbine VVAWT blade design from the aspect of aerodynamics is investigated. The aerodynamic characteristics of VVAWT can be improved by numerical simulation using Computational Fluid Dynamic (CFD). Many wind turbine researches are focused on accurately predicting efficiency. Various computational models exist, each with their own strengths and weaknesses that attempt to accurately predict the performance of a wind turbine. Being able to numerically predict wind turbine performance offers a possibility to reduce the number of experimental tests that are needed.

Reference [2] presented aerodynamics analysis of small horizontal axis wind turbine blades of 2D and 3D models using ANSYS-FLUENT software. The numerical solution was carried out by simultaneously solving the three-dimensional continuity, momentum and the Navier-Stokes equations in a rotating reference frame using a standard non-linear $k-\omega$ solver so that the rotational effect can be studied. The computational domain was designed as cylindrical shaped and sliding mesh method was used to generate mesh. These CFD model values were then validated using published calibrated lift and drag coefficients available in the literature.

Reference [3] developed a novel Savonius vertical axis wind turbine (VAWT) consisting of several asymmetric vertically-stacked stages. A computational investigation of the torque characteristics of such VAWT was carried out using the commercial CFD software, CFXdesign 2010. CFXdesign is an unstructured finite element CFD code that applies prismatic layers near the wall. The $k-\epsilon$ turbulence model and renormalized group (RNG) model which apply wall functions were used. A sliding mesh interface exists between the rotating and nonrotating regions of the domain. To validate, the steady and rotating simulation results of the novel Savonius rotor were compared with experimental data [4]. They highlighted the use of RANs for the development of vertical axis wind turbines based upon predicted results.

Reference [5] examined the performance of a Savonius wind turbine with constant cross-sections by means of quasi

2D flow predictions using ANSYS CFX. Simulation was done for three geometries: the Classical Savonius design, Bach-type one and Elliptical rotor. The Shear Stress Transport (SST) $k-\omega$ turbulence model was used based on earlier similar studies [6, 7, 8]. This turbulence model is known to give accurate predictions of flow separation under adverse pressure by an implementation of the transport effects on the formulation of the eddy viscosity. All mentioned designs were tested experimentally at different values of tip speed ratio. The results were presented in terms of coefficients of power, coefficients of torque and torque variation with the variation of angle of incidence. The wake is observed and examined by means of FFT analysis of pressure fluctuations at the point located downstream of the rotor.

Reference [9] 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. Straight blade fixed pitch VAWT with NACA0012 airfoil as a blade profile was considered to assess its performance. Analytical simulation is done for two-dimensional unsteady flow around the same VAWT model using Double Multiple Streamtube (DMST) model. ANSYS software is used to create 2D model of the turbine and the mesh. The RNG $k-\epsilon$ model was adapted for the turbulence closure. Comparison of the analytical results using DMST model and the CFD simulation results 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.

Reference [10] developed a CFD based computational model to analyze the aerodynamic performance of their novel designed vertical axis wind turbine (VAWT) and compared with their experiment work. The extent to which a CFD model employing the simplest turbulence representation can provide a useful input was investigated 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. The CFD model reflected the experimental data well 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.

Reference [11] performed a computational analysis of diffuser-augmented turbines by using high resolution computations of the Reynolds Averaged Navier–Stokes equations supplemented with a transition model. The amplification of mass flow through a shroud is found to increase nearly linearly with radial lift force. The nonlinear effects were examined in terms of the location of the leading edge stagnation point. Flow fields were examined in detail considering the regions with flow separation, the development of averaged velocity profiles, and the interaction between the helical turbine wake and shroud boundary layer.

Reference [12] performed the numerical simulations of the flow-field around a three straight blades VAWT using the finite-volume ROSITA (ROTORcraft software ITALy) solver developed at Politecnico di Milano for the solution of the

Reynolds-Averaged Navier–Stokes (RANS) equations with the Spalart–Allmaras turbulence model [13]. Numerical results gave a relevant aerodynamic feature of the flow around VAWT. The comparison with the experimental data validated the general features of the flow.

II. PRESENT SIMULATION

The main goal of this current research is to simulate numerically the three blades vane type vertical axis wind turbine (VVAWT) with different blade shape under movable vanes (open) conditions to investigate its aerodynamic performance. The present turbine rotor design consists of three blades of cavity shape each one having three vanes with angles 120° between each blade. This design increases drag force substantially. Solidworks2013 software is used to design turbine geometry as shown in Fig. 1. The top section view of the computational domain with the boundary conditions is

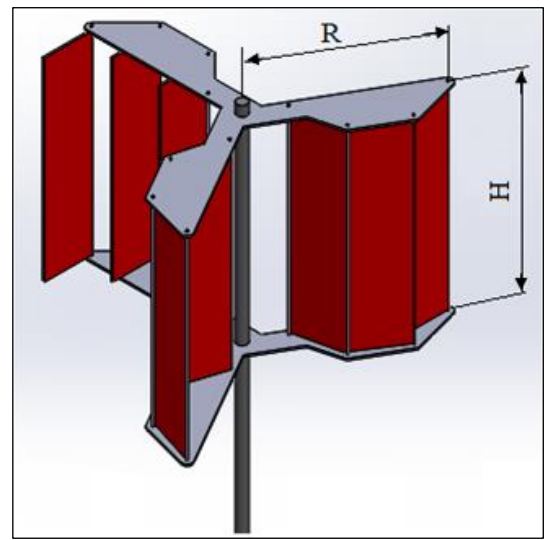


Fig. 1. Three blade newly designed vane type vertical axis wind turbine geometry.

shown in Fig. 2. The computational domain is of dimension (300 x300x300 mm), the rotor radius (R) and the blade height (H) are 75 mm and 100 mm respectively with cavity depth of 17 mm. The inlet is defined as a velocity inlet and the simulation is carried out for different inlet velocities 14, 20, and 25 m/s but the result is presented for 14 m/s only, while the outlet is set as a pressure outlet, keeping the pressure constant. The no slip shear condition is applied on the turbine blade, which sets the relative velocity of blades to zero. For numerical investigation, CFD software GAMBIT is used for meshing the geometry and specifying the boundary conditions, whereas ANSYS FLUENT 14.5 software (which employs finite volume method) is used to solve the Navier-Stokes equations. The Shear Stress Transport (SST) $k-\omega$ turbulence model is used in the analysis mainly because it is better than the other turbulence models available as reported earlier [14]. The numerical simulations were carried out for fixed blade position representing its angular positions of, 0° , 30° , 45° , 60° , 90° , and 120° .

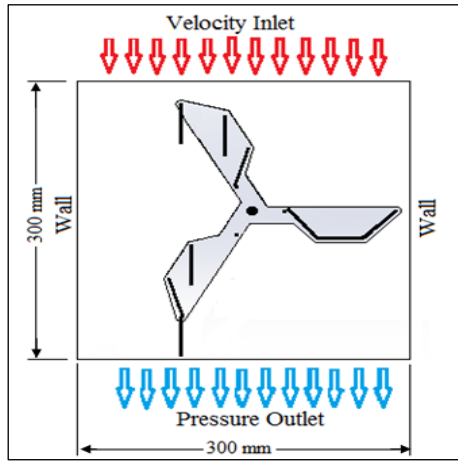


Fig. 2. Top section view of the computational domain with boundary conditions.

III. RESULTS AND DISCUSSION

Fig. 3 shows the residual plot, which has very small spikes in turbulent kinetic energy and specific dissipation rate for the residual being set as 10^{-5} .

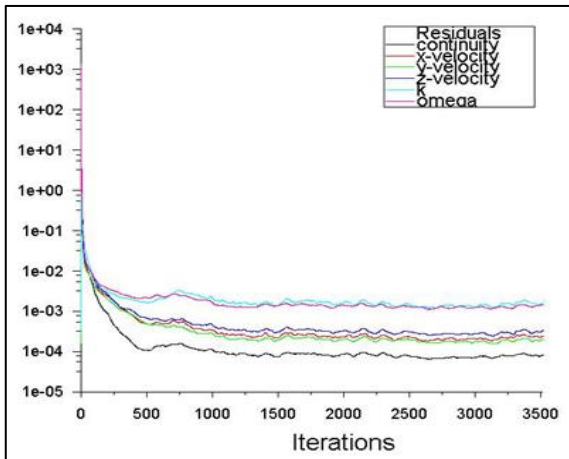


Fig. 3. Convergence history for continuity, momentum, and turbulence equations.

Fig.4 shows the variation of predicted drag coefficient (Cd) with different blade angular positions. It is evident that Cd is maximum (1.364) at zero blade angle, and minimum (1.071) at 60° blade angle and again maximum at 120° blade angle. At 120° blade angle, actually the next blade takes the position of its first blade because the angle between the turbine blades is 120°.

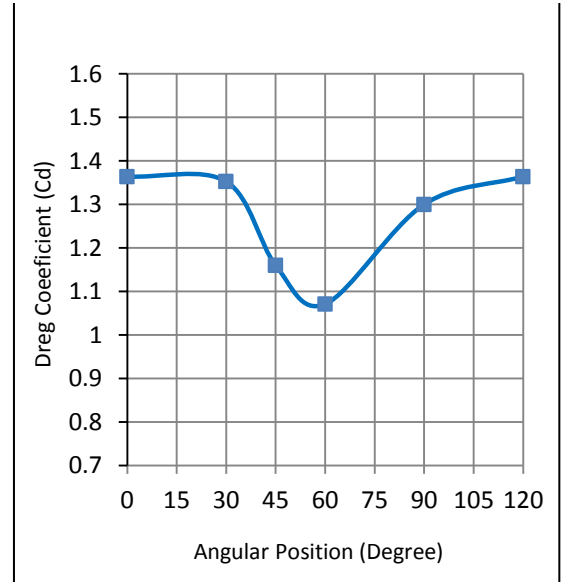


Fig. 4. Relation between drag coefficient and first blade angular positions

Fig. 5 shows the contour of static pressure distribution in and around turbine blade at different angular positions and Fig. 6 shows the static pressure contours on the three blades rotor surfaces with different blade angular positions.

From Fig. 5 it can be observed that the maximum static pressure is (1.73E+02 Pa). This occurs on the surfaces of the turbine in front of air flow direction. The minimum static pressure is (-9.14E+01 Pa), which occurs on the back surfaces of the turbine in front of air flow direction.

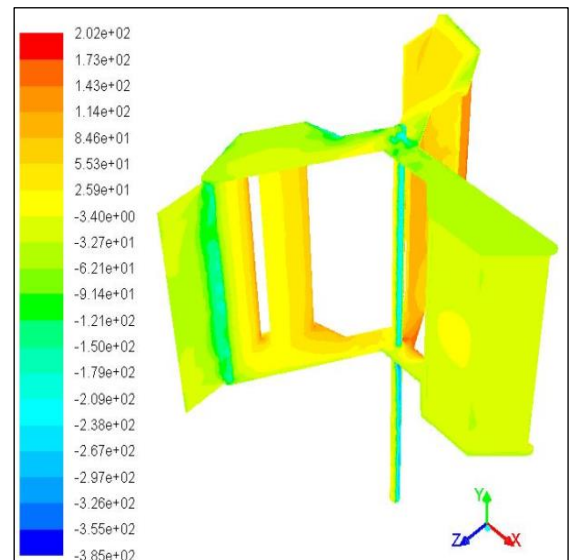


Fig. 5. Contour of static pressure distribution on three blades turbine surfaces with different angular position, at left closed vanes, right open vanes.

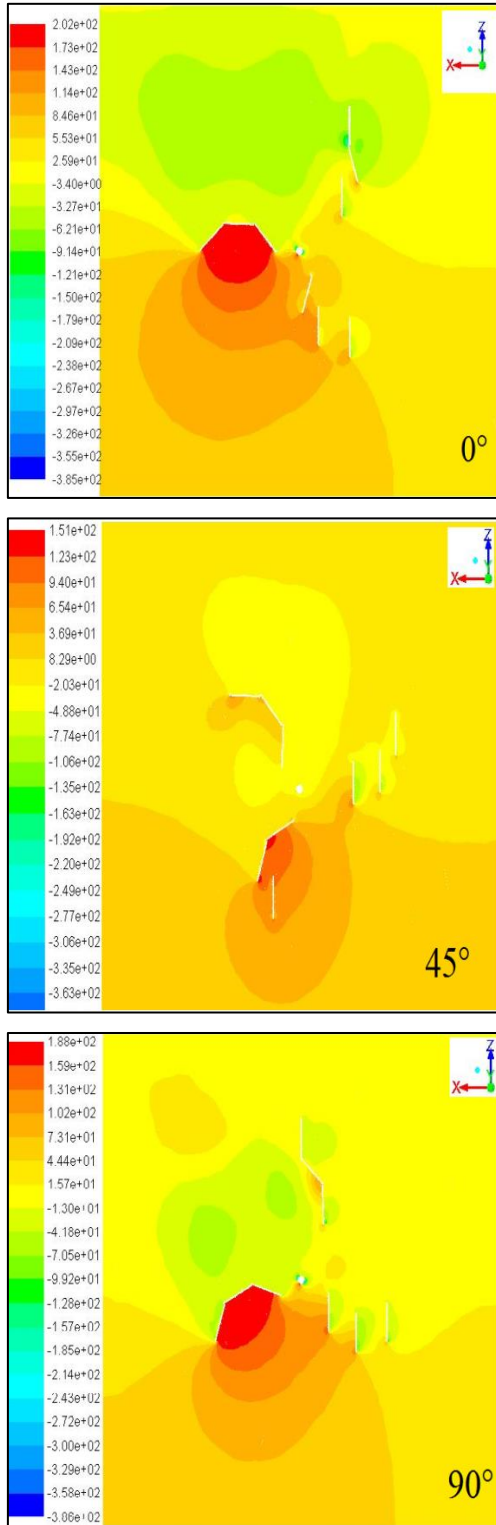


Fig. 6. Contour of static pressure distribution in and around turbine blade at different angular positions for three blades.

From Fig. 6 it can be seen that pressure drop occurs across the rotor from upstream to downstream side. This pressure drop indicates power extracted by the rotor causing its rotation. The maximum static pressure drop (2.02×10^2 Pa) is found in the case of blade angular position of 0° , whereas the minimum (-7.7×10^1 Pa) in the case when the blade angular position is 45° . The positive pressure on the frontal surface of the blades in front of air stream, and a negative pressure on the other sides of the back surface for the blades are clearly seen in Fig. 6.

Fig. 7 shows the velocity contours on the three blades rotor surfaces with different blade angular positions. The velocity in the region of wind turbine rotor is much larger than that in the upstream or air flow. 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 [15, 16, and 17].

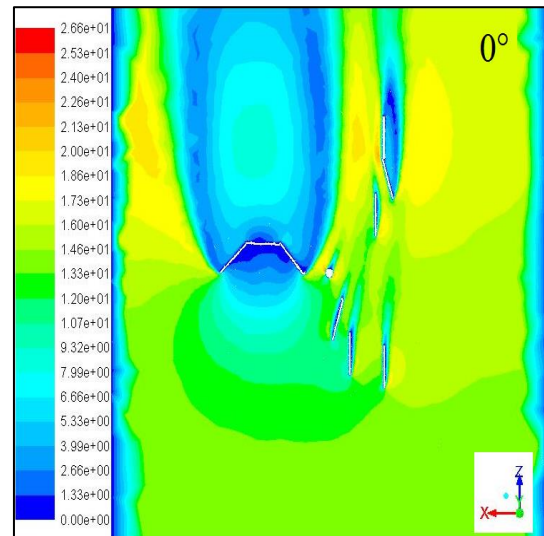


Fig. 7. Contour of velocity distribution in and around turbine blades at different angular positions.

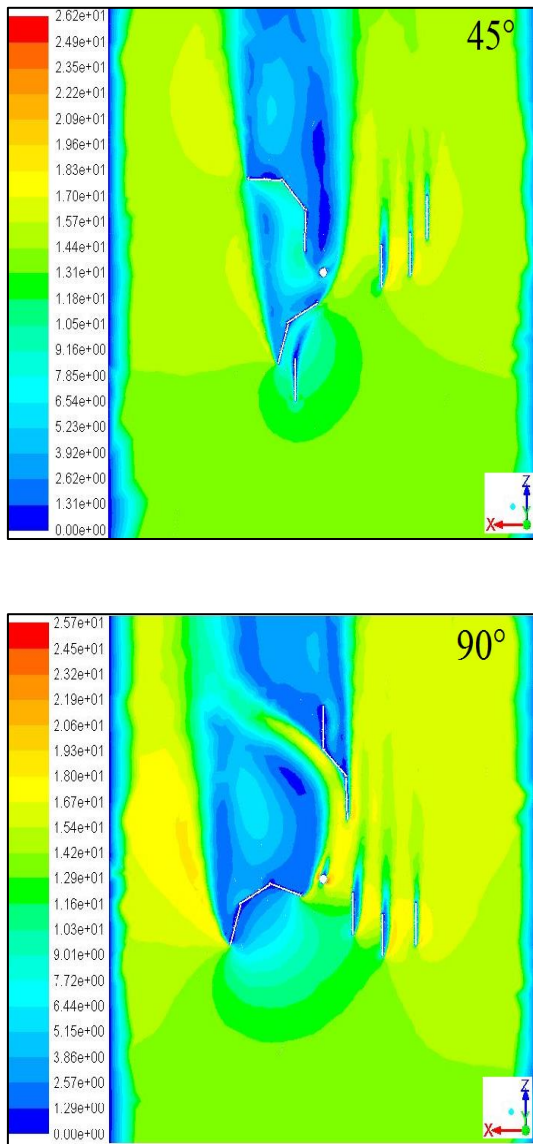


Fig. 7. (cont.) Contour of velocity distribution in and around turbine blades at different angular positions.

IV. CONCLUSIONS

The three dimensional numerical investigation is carried out using CFD software GAMBIT and ANSYS FLUENT where shear Stress Transport (SST) $k-\omega$ turbulence model is used to predict the aerodynamics of the three blades vane type vertical axis wind turbine having movable vanes. The flow field was simulated numerically at fixed vanes position and fixed wind velocities. The predicted results show that: (1) the drag coefficient increases with the increase in turbine frontal area and decreases with the decrease in frontal area, (2) the maximum static pressure drop ($2.02\text{E}+02$ Pa) is found in the case of blade angular position of 0° and minimum ($7.74\text{E}+01$ Pa) in the case when the blade angular position is 45° , (3) the velocity in the region of wind turbine's rotation was much larger than that of the upstream air flow. There is a wake dispersion region in the downstream of the wind turbine. The

results give good agreement when compared with experimental published results.

REFERENCES

- [1] R. Foster, M. Ghassem, A. Cota . "Solar energy renewable energy and the environment", CRC Press, Taylor & Francis Group. 6000 Broken Sound Parkway NW, Suite 300, Boca Raton, FL 33487-2742, 2010.
- [2] 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.
- [3] S. McTavish, D. Feszty, T. Sankar, "Steady and rotating computational fluid dynamics simulations of a novel vertical axis wind turbine for small-scale power generation". *Renewable Energy*, vol. 4, PP. 171-179, 2012.
- [4] T. Hayashi, Y. Li , Y. Hara, "Wind tunnel tests on a different phase three-stage Savonius rotor," *JSME International Journal, Series B: Fluid and Thermal Engineering*, vol. 48, PP. 9-16. 2005.
- [5] K. Kacprzak, G. Liskiewicz and K. Sobczak, "Numerical Investigation of Conventional and Modified Savonius Wind Turbines," *Renewable energy*, vol. 60, PP. 578-585, 2013.
- [6] 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, PP. 230-242, 2011.
- [7] C. Y. Kang, W. Wang, "Analysis of Aerodynamic Performance of Vertical Axis Spiral Wind Rotor," *Zhenjiang, China: Jiangsu Univ.*, 223-9, 2011.
- [8] A. Dragomirescu "Performance assessment of a small wind turbine with cross flow runner by numerical simulations," *Renewable*, vol.. 36, PP. 957-65, 2011.
- [9] A. M. Biadgo, A. Simonovic, D. Komarov and S. Stupar, "Numerical and Analytical Investigation of Vertical Axis Wind Turbine," *FME Transactions*, vol. 41, pp. 49-58, 2013.
- [10] 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, pp. 1195-1203, 2013.
- [11] A.C. Aranake, V.K. Lakshminarayan, K. Duraisamy, "Computational Analysis of Shrouded Wind Turbine Configurations," 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine (Dallas/Ft Worth Region), Texas, 1-17, 2013
- [12] M. Nini, V. Matta, G. Bindolino, A. Guardone, "Three-dimensional simulation of a complete vertical axis wind turbine using overlapping grids", *Computational and Applied Mechanics*, In Press, 2014.
- [13] P. R. Spalart, S. R. Allmaras, "A one-equation turbulence model for aerodynamic flows", in: *AIAA Paper*. 92-0439, 1992.
- [14] A. Y. Qasim, "Investigation and Design of a New Impeller Wind Turbine", PhD. Thesis, School of Manufacturing Engineering, University Malaysia Perlis, 2013.
- [15] N. Fuller, D. M. Wienes, A. Johnston, J. French, " Design and analysis of the aerodynamic components for a kilowatt scale VAWT optimized for low altitude implementation in remote rural villages,". *ASME 2011 5th International Conference on Energy Sustainability ES2011*. August 7-10, Washington, DC, USA, 2011,
- [16] S. Roy, U. K. Saha, "Review on the numerical investigations into the design and development of savonius wind rotors," *Renewable and Sustainable Energy Reviews*, vol. 24, pp. 73 – 83, 2013.
- [17] W.S. Widodo, A. C. Chin, H. Sihombing and M. Y. YUHAZRI, "Design and Analysis of 5 kW Savonius Rotor Blade," *Global Engineers and Technologists review*, vol. 2, PP. 1-8, 2012.