A Study of Modified Vertical Axis Tidal Turbine to Improve Lift Performance

Nu Rhahida Arini, Stephen R. Turnock, and Mingyi Tan
FSI Group, Boldrewood Campus, University of Southampton, Southampton, United Kingdom
Email: nu.rhahida@gmail.com

***Abstract*—A design of Darrieus vertical axis tidal turbine using modified airfoil was studied numerically in this work. The turbine design was evaluated in 2D CFD model using k-ω turbulence model, upwind interpolation scheme and simulated using OpenFOAM. The turbine had three blades which were arranged symmetrically. The blades were of NACA 0012 airfoil which had been modified in the trailing edge region to increase its lift performance. The modification was made by truncating the trailing edge at the 15% of chord length from the trailing edge. Single normal and blunt airfoil were modelled and investigated prior to the turbine design evaluation. For generating the mesh, C-structured grid was employed to the single airfoil model and hybrid mesh to the vertical axis tidal turbine model. From the single airfoil simulations, it was found that blunt NACA0012 had 12% higher lift coefficient than normal airfoil and the pressure coefficient magnitude of blunt vertical axis tidal turbine significantly rise two times from vertical axis tidal turbine using normal NACA0012 airfoil.**

***Index Terms*—vertical axis tidal turbine, Darrieus turbine, truncated Naca0012, blunt airfoil**

I. Introduction

Renewable energy has become a high demand over the past few years because of the fuel price increase as the effect of energy crisis. Natural resources ideally become alternative energy which are more environmental friendly and lower cost regarding its recovery and renewability. For some countries which are surrounded by oceans, waves and tides can become promising resources to gain renewable energy. Lundin [1] mentioned several designs of device which were commonly used for capturing marine and tidal current.

One renewable energy device which is considered to have progressive development is vertical axis tidal turbine. The vertical axis tidal turbine is a recommended device for a country which intends to gain renewable energy from abundant tidal energy resources. Vertical Axis Tidal Turbine (VATT) uses force from the tide which acts on each blade, to rotate its rotor and generate mechanical energy. Amongst many vertical turbines, Darrieus type is the widely used for harnessing energy. Khalid [2] overviewed the use of VATT for power generation and found that it was more environmental friendly than other tidal device such as tidal barrack. Environmental effect of tidal turbine operation was also investigated by Lloyd [3] regarding the underwater noise and potential impact on the marine environment.

In the turbine operation, fluid flow is perceived to be turbulent. The turbulence condition which is characterized by the irregularity and diffusivity leads to the fluid structure interactions complexity thus increases the difficulties of solving the problem as precise as the real condition. Computational Fluid Dynamic (CFD) method likely becomes an approach to overcome with an accurate result. In this work, CFD was implemented in the study of a single blunt NACA0012 airfoil and compare the result with normal airfoil prior to the evaluation of VATT. The result then was applied to the design of symmetric three bladed VATT using the blunt NACA0012 airfoil. Some parameters are of importance in turbine design as studied by Gosselin [4]. He found that solidity, number of blades, Reynolds number (Re), blade pitch angle (fixed and variable) and blade thickness strongly affected the dynamic configuration of Darrieus vertical axis turbine. One of his postulates was to find the optimal radius-based solidity for vertical axis turbine which was around 0.2.

II. CFD Design

A. Airfoil Design

Fitzeger [5] explained rigorously the CFD method including the stage of preprocessing, processing and postprocessing. The success of this method depends on the preprocessing stage in which the mesh is generated. In this stage, basically a domain which represents a region over where the fluid passes through is discretised into number of cells. The discretization exhibits grids and form a CFD mesh. The grids should be arranged in a high quality manner which is stated by terms like aspect ratio or skewness, in order to obtain high quality mesh on all over the domain and yield an accurate simulation result. The mesh should be sufficiently fine to capture the turbulence phenomena around an airfoil but in other hand it also requires to be feasibly coarse to ensure the calculation reliably doable. Date [6] showed a topology to generate mesh in the modified NACA0012 airfoil vicinity. He refined the trailing edge region around the flap which had similar profile to the blunt trailing edge airfoil. His result showed the refine grid (number of grid was 49) had good agreement with his experiment.

Manuscript received June 30, 2015; revised December 28, 2015.

doi: 10.18178/ijoee.4.1.37-41

In the processing stage, Navier stokes equation is imposed to all cells and solved by the chosen turbulence model. There is no definite rule or specific criteria to consider in choosing turbulence model for solving CFD cases, however the model impact the simulation stability and solution accuracy. In this work, k-ω SST RANS turbulence model was chosen for both cases, single airfoil and VATT simulation. Any other parameters which were taken into account such as airfoil vicinity, mesh topology and boundary conditions for VATT design were also the same as in single airfoil model.

Normal NACA0012 with 0.75 m chord length and had Reynolds number of 3.07×10^6, was primarily simulated and validated to assure that mesh topology was valid to model the blunt airfoil in the following step. The generated single airfoil mesh employed C-strucutred grid as illustrated in Fig. 1. The mesh was divided into nine subdomains including boundary layers around the airfoil, nose part, the upper and lower surface of airfoil, the streamwise region and five far field region which was away from airfoil surface. The airfoil had the same number of points on the upper and lower surface which was 200. The nose part had 65^o of angle and contained of 100 points along the nose surface. The streamwise mesh region was drawn from the trailing edge to the outlet of the domain which was 200 and also set to have the same nodes for upper and lower part.

Eleni *et al.* [7] simulated a model of NACA0012 with C-structured grid which showed that independent mesh simulation was performed when number of cell reached 80,000. Total number of cells in this domain reached 380,221. The airfoil was located in the centre of the domain. The inlet was half circle with 17c in diameter to avoid blockage effect as seen in the Fig. 1. The vertical direction mesh was configured with the same nodes for upper and lower part.

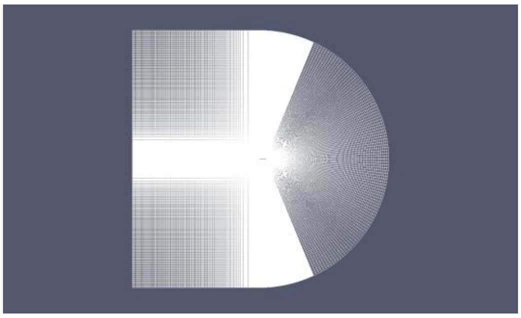

Figure 1. Domain of single airfoil

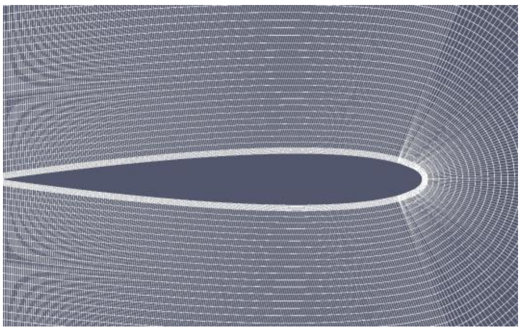

Figure 2. Mesh topology of single NACA 0012 at 0^o angle of attack

Both, normal and blunt, single airfoil were simulated at -8^o, -6^o, -4^o, -2^o, 0^o, 2^o, 4^o, 6^o, 8^o angle of attack. There were seven mesh boundaries used in the simulation of single airfoil including inlet, outlet, top, bottom, front, back and wall of the airfoil. The mesh topology of single airfoil at 0 °angle of attack is depicted in Fig. 2.

For blunt airfoil, Ramjee [8] studied experimentally four different truncations and found that the highest lift coefficient accured at airfoil with 15% of truncation. In this blunt airfoil model, NACA0012 was truncated at 15% of chord length from trailing edge as drawn in the Fig. 3.

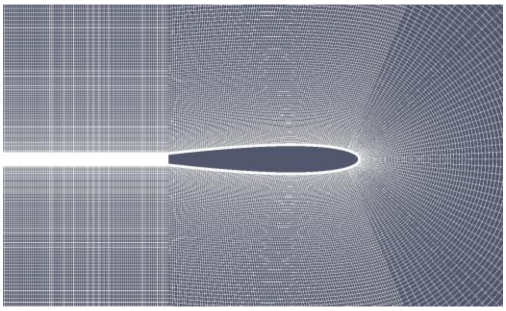

Figure 3. Mesh topology of single blunt NACA 0012 at 0 °angle of attack

The mesh topology and boundary conditions of the blunt airfoil was the same as the normal one except for the trailing edge region. Behind the trailing edge, boundary layers were added and refined to resolve the turbulence phenomena. The boundary had 30 layers in total. Total cells in the blunt airfoil domain was 391,879 and simulated using SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm in OpenFoam.

B. Turbine Design

VATT was designed using hybrid grid which combined structured and unstructured mesh. From his work, Gretton [9] suggested a VATT topology using O-structured grid to model the turbine and unstructured grid for far field region. Gosselin [4] was also meshing using similar topology as Gretton's. Both simulation came with the same agreement as their experiment. Similar mesh topology was also conducted by Lanzafame [10] who generated a Darrieus turbine model with hybrid mesh. The VATT domain was decomposed from a rotating and fixed part named rotor and stator respectively as illustrated in Fig. 4.

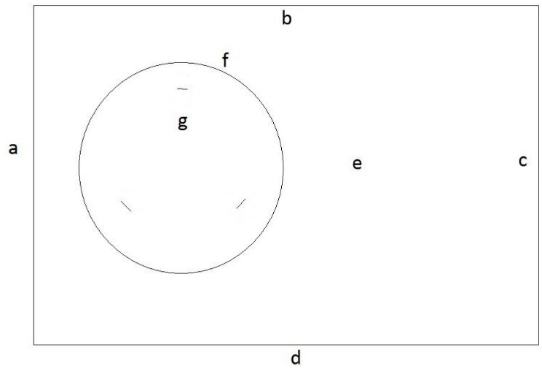

Figure 4. Illustration of VATT domain: a. inlet, b. top, c. outlet, d. bottom, e. front, f. interface, g. rotor front

The rotating domain represented the vertical axis turbine which rotated with respect to its vertical axis, normal to the flow direction. The stator model the environment of the far field region and remained fixed during the simulation. There were nine boundaries utilized in the VATT model which were inlet, outlet, top, bottom, front, back, airfoil wall, stator and rotor which shared a surface as an interface. Incoming flow entered the domain from the stator inlet, went across and swept the rotor part prior to the final discharge at the outlet of stator. The rotor radius was 3c of which the centre was located in the middle of vertical direction as can be seen in Fig. 5. The domain size was 17c height and 13c from inlet of 62c length domain.

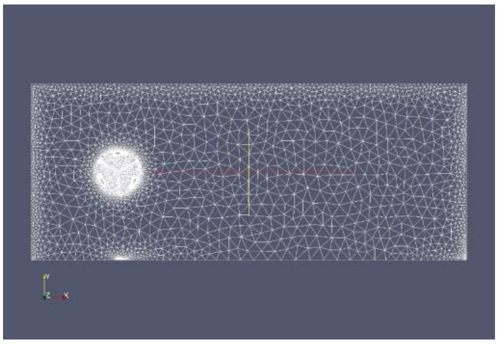

Figure 5. Rotor and stator mesh

The domain of the stator was set to have equal distribution for all sides. For the interface between rotor and stator, there were 258 points in total with equal distribution at circumference and all connector lines of the three blade domains. The total number of cells was 188,753 which exceeded the minimum cells required for independence mesh. The examination of the mesh gave the skewness and aspect ratio mesh value of 0.6 and 2.5.

The streamwise behind the rotor was intentionally modelled with longer space to permit the flow fully developed. Twenty boundary layers which generated around each airfoil using structured grid with the growth ratio of 1.1 and equal for and lower. The boundary layers were constructed with the same topology surface for all the three blades and the far field region mesh was built up with unstructured grid as shown in Fig. 6.

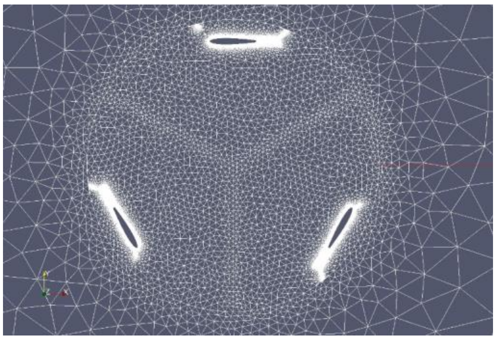

Figure 6. Rotor (turbine) mesh

For the airfoil surface, total number of nodes on each airfoil surface was 240 and the distribution of the nodes was one hundred times denser at the trailing edge than at leading edge curve as the vorticity was likely to appear behind the trailing edge. The boundary layers at trailing edge were extended until half of the chord length with coarser nodes at the further part away from trailing edge and had 200 nodes in total for each surface. In that extension region, the connector had 50 times finer distribution in the closer part to blunt trailing edge. Near trailing edge is shown in Fig. 7.

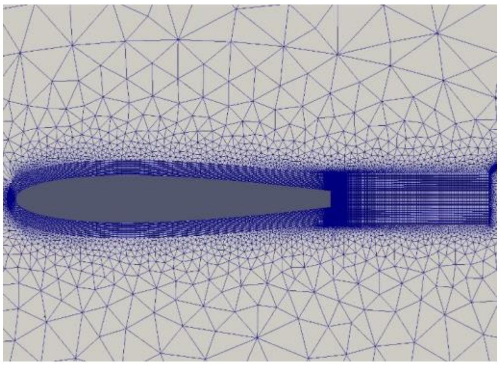

Figure 7. Airfoil mesh

III. Result and Discussion

In the simulations of single airfoil, the flow was assumed incompressible and turbulent. Lift coefficient was calculated in the simulation for each angle of attack. The coefficient then was validated using Abbott's experiment [11], Eleni [7] and Mittal's simulation [12] (Fig. 8).

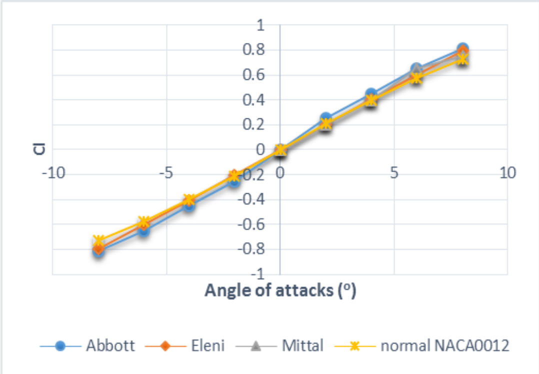

Figure 8. Validation of normal NACA 0012 using references data

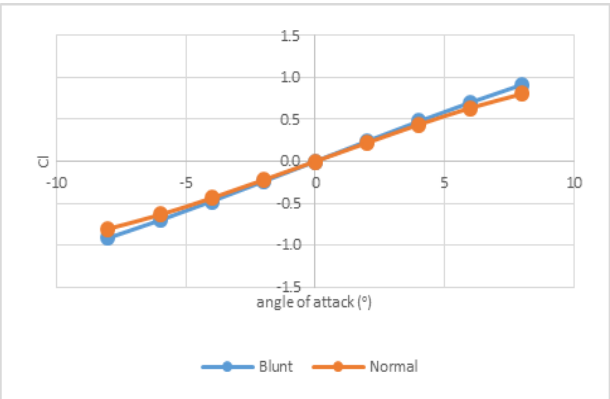

Figure 9. Comparison of blunt and normal NACA 0012 lift coefficient

In general at 2° angle of attack, the error was found to be maximum amongst all angle of attacks simulated. The maximum error was 6% from the Abbott's experiment and 10% from Eleni's and Mittal's simulation. The error was contributed mainly by the fixed mesh which cannot be adapted automatically by the change of angle and likely the mesh of 2° angle was the most distorted. From that error it can be deduced, the model was accurate enough for evaluating blunt airfoil and the comparison of normal and blunt result are shown in Fig. 9.

The maximum Cl increase was found at 8° angle of attack and reached 12% from normal airfoil. At the blunt airfoil, the fluid was accelerated more abruptly because of sudden change of the truncation profile at the trailing edge. The truncation profile was considered as a gap between upper and lower surface which created a sudden change in fluid velocity. The drastically velocity alteration drove the fluid to move faster across the blunt airfoil surface than in the normal airfoil. The effect appeared stronger at the higher angle of attack.

In the vertical axis tidal turbine model, the turbine had three blades of blunt NACA0012 which was arrayed symmetrically as illustrate in Fig. 6. The model was simulated using unsteady solver of OpenFoam named pimpleDyMFoam with 0.9 maximum courant number. The solidity and tip speed ratio of the vertical axis tidal turbine was 0.1829 and 5 respectively as suggested by Gosselin [4]. The angular velocity was 4.2 rad/s and evaluated at different azimuth angle along one rotation. The pressure coefficient at lower surface trailing edge at zero azimuth angle with respect to vertical axis during one rotation is shown in Fig. 10. The azimuth rotation could be assumed as an oscillating airfoil which starts from 0° moving upward. It can be seen from Fig. 10 that the pressure was negative since the point laid on the surface which acted as upper surface of upward motion of oscillating airfoil thus having suction pressure. From the starting point, the pressure magnitude was higher and behaved similar to higher angle of attack of oscillating airfoil. At approximately 20°, the pressure magnitude exceeded peak value at which maximum lift coefficient likely occurred. The pressure magnitude went down progressively until approximately reached 300° azimuth angle at which airfoil experienced a stall phenomena.

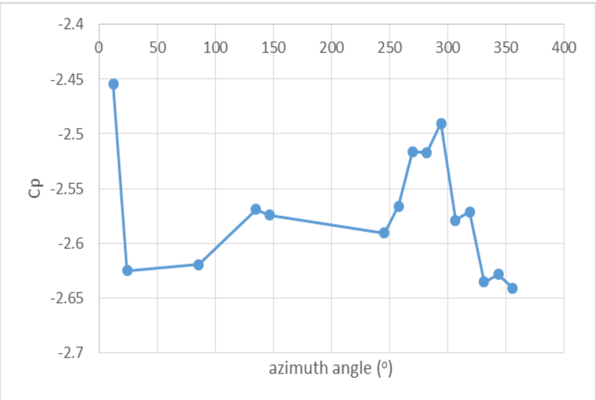

Figure 10. Pressure at lower surface trailing edge during one rotation

The maximum Cp magnitude of blunt VATT was two times higher than using normal NACA 0012 [4]. Although lift coefficient was not precisely calculated, it still can be predicted that Cl will also increase as it is proportional to the pressure distribution.

IV. Conclusion

The single normal and blunt NACA0012 was observed numerically in this paper prior to the numerical evaluation of three bladed blunt NACA0012 vertical axis tidal turbine. 2D CFD model was generated using C-structured grid for single airfoil and hybrid mesh for turbine simulation. k-ω SST was applied for both simulation, single airfoil and the turbine model. The evaluation was performed at nine different angles of attack for single airfoil simulation and fifteen azimuth angles of one rotation for VATT simulation.

From the simulation it can be found that C-structured mesh with the topology suggested was capable to model the turbulent phenomena in the airfoil and VATT. The lift coefficient of blunt airfoil was 12% higher than normal one and reasonable to be applied in VATT. The pressure coefficient magnitude using blunt airfoil was predicted two times higher than using a normal NACA0012.

References

[1] M. G. U. Lundin and M. Leijon, "Ocean energy. Research report," Department of Electricity and Lightning Research Uppsala University, Sweden, 2010.

[2] S. S. Khalid, Z. Liang, and N. Shah, "Harnessing tidal energy using vertical axis tidal turbine," *Research Journal of Applied Sciences, Engineering and Technology*, vol. 5, no. 1, pp. 239-252, 2013.

[3] T. Llyod, "Modelling techniques for underwater noise generated by tidal turbines in shaloow water," in *Proc. 30th International Conference on Ocean, Offshore and Arctic Engineering*, 2011.

[4] R. Gosselin, G. Dumas, and M. Boudreau, "Parametric study of H-Darrieus vertical axis turbines using uRANS simulations," in *Proc. 21st Annual Conference of the CFD Society of Canada*, 2013.

[5] J. H. Ferziger and M. Peric, *Computational Methods for Fluid Dynamics*, 2nd ed., Berlin: Springer-Verlag, 1999.

[6] J. Date and S. R. Turnock, "Computational evaluation of periodic performance of a NACA 0012 fitted with a gurney flap," *Journal of Fluids Engineering*, vol. 124, 2002.

[7] D. C. Eleni, T. I. Athanasios, and M. P. Dionissios, "Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil," *Journal of Mechanical Engineering Research*, vol. 4, no. 3, pp. 100-111, 2012.

[8] V. Ramjee, E. G. Tulapurkara, and V. Balasbaskaran, "Experimental and theoretical study of wings with blunt trailing edge," *Journal of Aircraft*, vol. 23, no. 4, pp. 349-352, April 1986.

[9] G. I. Gretton, T. Bruce, and D. Ingram, "Hydrodynamic modelling of a vertical axis tidal current turbine using CFD," in *Proc. EWTEC Conference*, 2009.

[10] R. Lanzafame, S. Mauro, and M. Messina, "2D CFD modeling of H-Darrieus wind turbine using a transition turbulence model," in *Proc. 68th Conference of the Italian Thermal Machines Engineering Association*, 2014, pp. 131-140.

[11] I. H. Abbot and A. E. V. Doenhoff, *Theory of Wing Sections*, New York: Dover Publishing, 1959.

[12] S. Mittal and P. Saxena, "Hysteresis in flow past a NACA0012 airfoil," *Computer Methods in Applied Mechanics and Engineering*, vol. 191, pp. 2179-2189, 2002.

Nu Rhahida Arini received her bachelor and master degree from mechanical engineering department at Intitut Teknologi Bandung in Bandung, Indonesia. Her interest in fluid mechanics and energy conversion drives her attention to focus her research on renewable energy. Currently she is taking PhD study on tidal current turbine in University of Southampton.

She is a senior lecturer in electronic engineering polytechnic institute of Surabaya, in Surabaya, Indonesia. She published some papers in renewable energy and control system.