Performance Enhancement Effort for Vertical-Axis Tidal-Current Turbine in Low Water Velocity

Dendy Satrio[#*1], I Ketut Aria Pria Utama[#2], Mukhtasor[#3]

[#] *Faculty of Marine Technology, Institut Teknologi Sepuluh Nopember*
Kampus ITS Sukolilo, Surabaya, Indonesia

[1]dendy.satrio15@mhs.na.its.ac.id

[2]kutama@na.its.ac.id

[3]mukhtasor@oe.its.ac.id

Abstract— **Tidal-current turbine which is vertical-axis type has been proposed by many researchers as a power generation with renewable energy resources. The main disadvantage of vertical-axis type is low performance compared with horizontal-type. This study focuses on how to enhance the performance by maintained lift capability with modified an original foil profile of NACA 63(4)021 to be a tubercle profile. The foil is simulated with Computational Fluid Dynamics software with some variations on angle of attack. Tubercle in leading-edge foil section can delay static stall angle and increase lift performance in high angle of attack. Since the vertical-axis turbine is rotated in 360^o that mean its angle of attack change continuously in each position of rotation then it will experience a dynamic stall. Therefore, vertical-axis turbine which the foil profile modified with tubercle become a good chance to increase the range operation of extracting energy in dynamic stall condition by maintain the lift performance in high angle of attack.**

***Keywords*— Tidal-current energy, vertical-axis turbine, foil simulation, performance enhancement effort, low water velocity.**

I. Introduction

Many parties are interested in utilizing tidal-current energy resource as electrical power generation. That is because the availability of this energy resource is very abundant. Especially Indonesia is as one of the largest archipelagic countries in the world. Indonesia has a practical tidal-current energy resources of 17,898 MW [1].

Some vertical-axis turbines that have reached a commercial stage require a high water velocity [2, 3]. However, several potential locations in Indonesia, such as Riau Strait, Boleng Strait and Mansuar Strait, can only achieve a maximum tidal-current velocity of 1.39, 1.5, and 1.79 m/s [4]. Therefore, research on vertical-axis turbine that applicable for low water velocity is interesting to do.

Based on a critical review in previous study, the main disadvantage of vertical-axis turbine is low coefficient of performance [5]. Three constituent components that construct a vertical-axis turbine are foil, blade and rotor. Foil is a cross section base profile of a blade. This study try to solve that problem by modified the foil profile. An original foil profile (NACA 63(4)021) is modified to be a tubercle profile on the leading-edge foil section. Tubercle on leading-edge section can delay stall angle and increase lift performance in high angle of attack [6, 7, 8]. If the foil coefficient of lift can be increased, the turbine coefficient of moment will be increased then finally the energy extraction increasing.

II. Numerical Simulation

A. Geometry

Most of vertical-axis tidal-current turbines have three blades (Fig. 1). The cross sectional profile of the blade is constructed by original shape of a foil without any modification. Usually, the foil sections that used in vertical-axis turbine are NACA 0012, NACA 0018 and NACA 63(4)021. The symmetrical foil section is chosen to maintain the foil lift performance in all position of turbine rotation. In a vertical-axis type, the turbine is rotating in 360^o centered to its axis therefore the blades angle of attack always change dynamically. In this study, NACA 63(4)021 foil section is chosen because the profile is similar with morphology that found on humpback whales flippers [6, 7].

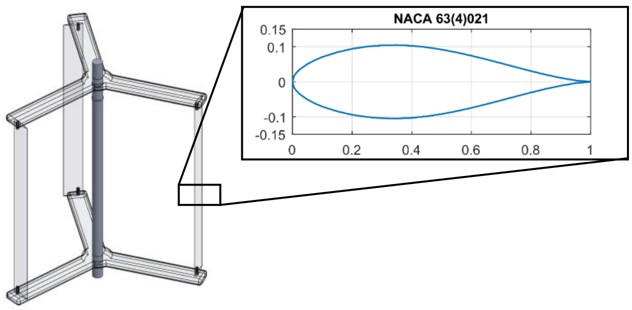


Fig. 1 Cross section of a foil on vertical-axis turbine

In Table. 1, some parameters have been determined and studied to find the best tubercle design. The tubercle profile is controlled with three design parameters, they are chord, width and height as illustrated in Fig. 2. In this study, chord and width is defined as fixed length with value of 1 m and 0.4 m. These values require to be keep constant for resulting amount mesh with same total mesh elements in equal distribution. This is because the meshing distribution will influence on the lift and drag calculation. After that, height parameter is varied to find the best combination parameter of a tubercle design. The original foil profile has height parameter of 0 m. The tubercle foil profile has height parameter of 0.05; 0.075; or 0.1 m.

Table. 1 Tubercle design parameters

Foil	Chord C (m)	Width W (m)	Height H (m)
NACA 63(4)021	1	0.4	0.05; 0.075; 0.1

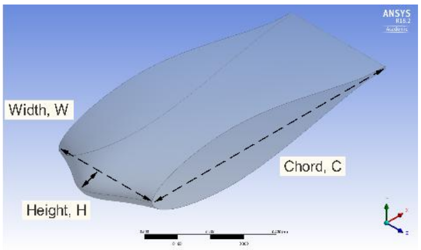


Fig. 2 Foil profile modified with tubercle on leading-edge foil section

B. Boundary Conditions

In a numerical simulation, a computational domain requires to determine as depicted in Fig. 3. Diameter of computational domain is determined as 20C with circle shape. The foil is located in the centre of the circle. This is because for ensuring that vortex generated behind the foil formed far away to make sure a fully developed phenomenon. Boundary conditions that used in this simulation are described in Fig. 3. Inlet boundary condition is set as velocity inlet with a constant value of 0.1836 m/s. This value is signed as same Reynold number value of 183,000 in the reference experimental paper [7]. The experimental paper is used for validation in the next section. Side A and B are defined as periodic boundary conditions to accomodir vortex interaction between side to side tubercle pofile. The foil specify as wall with no slip conditions.

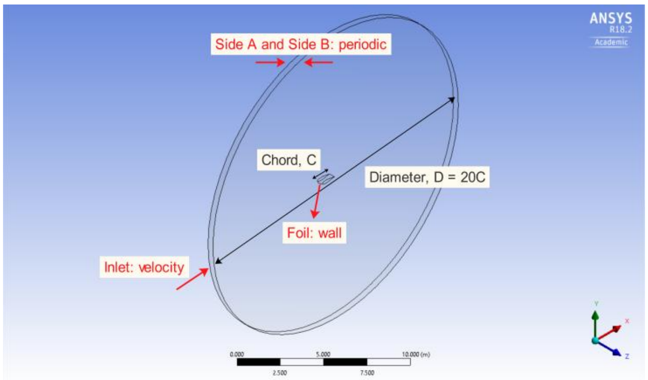


Fig. 3 Computational domain

C. Meshing

The meshing strategy uses O-type structured mesh. Some sizing element controls in foil edge, width edge and slice domain edge are implemented to keep the mesh element in amount of needed. The total mesh elements is 1,400,000. These mesh elements is enough to capture the flow around the foil regarding in grid independence study. The elements is distributed from foil in the centre to edge of the circle as depicted in Figure 4.

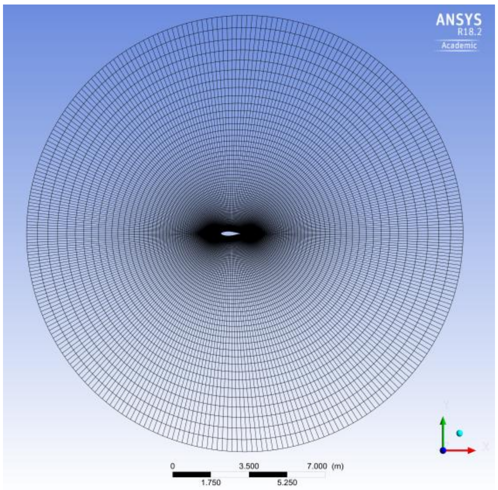


Fig. 4 Structured mesh overview

Lift and drag calculation in a foil simulation are strongly influenced by the mesh strategy especially on location near the foil wall. It is because lift and drag base on surface force calculation. Figure 5 shows the zoom in view in leading-edge foil to explain that in this location required to be treat for resolving the boundary layer flows. The non-dimensional wall distance (y+) value for the first mesh is 0.99 regarding the Reynolds number value from the experiment and according experience in previous study [7, 9, 10]. The first mesh height is 1.13 10^{-4} m as calculated with y+ equation.

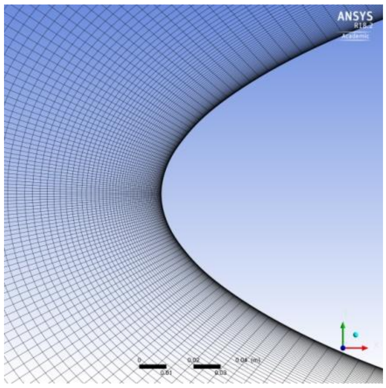


Fig. 5 Leading-edge mesh overview

D. Solver setup

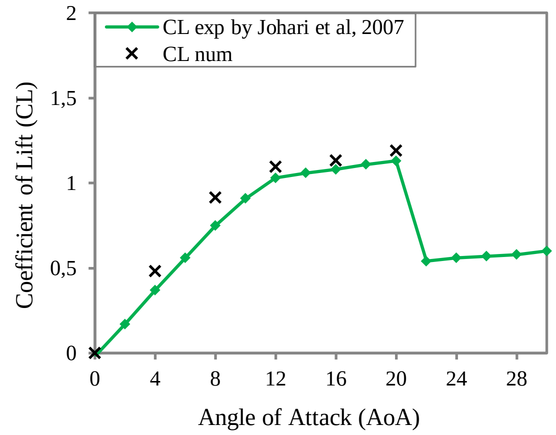
The foil simulation performs with Computational Fluid Dynamics software by ANSYS-Fluent code. The simulation uses 6 cores internal parallel processing. The foil is located in the centre of computational domain with static condition (Fig. 3). After that, the incoming flow velocity from inlet is changed in accordance with variation on angle of attacks. Therfore the code is used to solve incompressible steady state of Reynolds-Averaged Navier-Stokes (RANS) equations.

Turbulence model that used to compute Reynolds stress in the RANS equations is K-ω SST two equations. K-ω SST turbulence model is selected because it has good accuracy at the bulk regions and at the boundary layer regions [9, 10, 11]. The pressure velocity coupling uses SIMPLE algorithm scheme. The spatial discretization gradient uses finite volume method with least squares cell based. All transport equations are set as second-order scheme.

III. RESULTS AND DISCUSSION

A. Numerical validation of original foil

Foil experiment results that conducted by Johari et al (2007) are used to validate the simulation of original foil profile [7]. Firgure 6 shows coefficient of lift and drag of original foil. Green line with diagonal point represents the experimental data by Johari et al (2007) [7]. And cross points represent numerical results of foil before modified. Both of them are original foil profile of NACA 63(4)021. There is very good agreement between the numerical result and experimental published data. The numerical model can follow the experiment data in the pre-stall regime precisely. Pre-stall regime for NACA 63(4)021 is starting from 0 degree until 20 degree. After that angle, the lift performance experiences sudden stall while increasing drag performance.



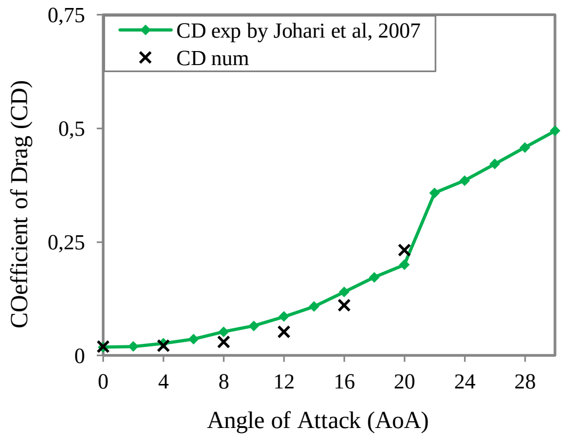


Fig. 6 Validation of coefficient of lift and drag with experimental data

In this paper, the simulation results are kept in just before stall. After stall occurred, the lift performance is very difficult to predict. Stall is a phenomenon when the coefficient of lift reach peak performance then drop significantly while changing in angle of attack. However in this simulation, the coefficient of lift is become higher and higher as increasing angle of attack starting from 24 degrees and the rest. This result is not correct eventhough the drag coeffient constantly follows the experimental data. Therefore, these errors are not shown on the graph.

Simulation in post-stall regime produce large scale flow structures being shed off behind the foil which the vortice size is very big. There are two turbulence phenomena which are at the microscopic scale and flow structures scale. Steady state simulation with RANS models can not capture that flow phenomena precisely. K-ω SST turbulence model is based around a single length scale being representative of the full turbulence spectrum.

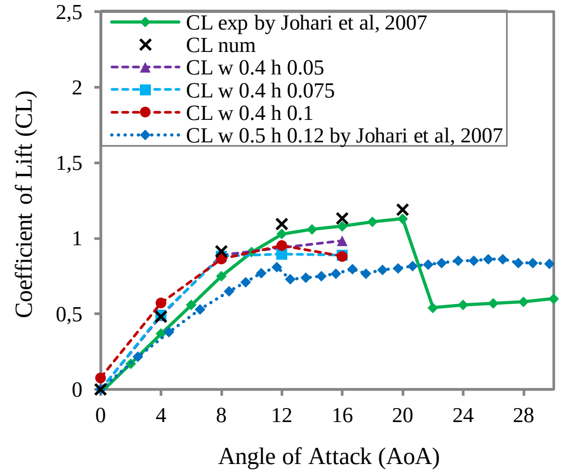
Since in post-stall regime there are two very distinct length scales of turbulence phenomena. Whereas normal turbulence models only have a size spectrum which is represented by a single length scales. Therefore, another approach of turbulence models such as LES or one of its derivatives, they are DES and SAS, is required to be apply to capture the phenomena. So the simulation should be run in transient solution since in post-stall simulation is required to run with a LES family of turbulence models. A steady state simulation is not appropriate to apply.

The problem of foil simulation in post-stall regime is complicated because the fluid flow behind the foil highly unsteady. Therefore the simulation should be run in transient solution. However if the simulation in the post-stall regime want to keep with steady state solution, advanced efforts should to do in order to get good statistics value such as mean lift and drag force. So that will probably have to do quite long simulations. Furthermore it also will require a very fine mesh especially in area near the foil.

B. Simulation results of modified foil

Foil simulation for original foil profile of NACA 63(4)021 have been succesfully done in the pre-stall regime. After that, the simulation can be continued with modify the original profile in leading-edg sectione with tubercle profile. The simulation results of the modified foil with three given scenarios are shown in Fig. 7. As the same as the original profile, the modified foil just keep in 0 to 16 degrees angle of attack. The previous results of both the experiment and numerical for the original foil of NACA 63(4)021 are kept in the graphic. Then the following results of modified foil are added into the graphic. So the different can be seen making easy to evaluate and analyse.

Furthermore, another experimental data of modified foil by Johari et al (2007) is also added to the graphic to give an inslustration of full effect of tubercle foil starting from pre-stall until post-stall regime [7]. The tubercle experimental data shows with dash line and diagonal blue points. This tubercle foil has different tubercle design with the tubercle foils that studied here. Hopefully in the future, the numerical model can predict well in post-stall regime as the experimental result has been done. Then the best tubercle design could be obtained.



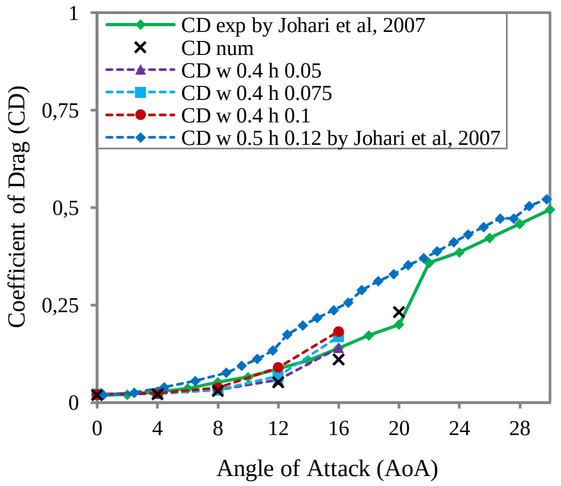


Fig. 7 Numerical simulation results of modified foil

The upper graph at Fig. 7 shows coefficient of lift for both original and modified foils. The lift performance for all foils are increasing as increasing angle of attack from 0 to 8 degrees. Then the coefficient of lift of original foil continue to increase until 20 degrees. In angle of attack 12 to 20 degrees, the original profile of NACA 63(4)021 has the best lift performance. However for tubercle foil, the coefficient of lift starting to perform constantly flat. From the simulation results, the coefficient of lift is flat from 8 to 16 degrees. Therefore, tubercle foil in this regime is not to profitable. The simulation can not continue to predict the lift because already experience stall. However, adding the experimental data is for explaining how the performance of modified foil after stall [7]. In angle of attack 20 to 30 degrees, tubercle foil do not experience stall compare with the original foil. The coefficient of lift continue to perform well until 30 degrees. Here is the benefit of modified foil with tubercle.

That matter open a benefit of applying this concept in vertical-axis turbine. Blade in vertical-axis turbine have a good chance to put tubercle on the leading-edge of blade section. Even though there is a negative report from another researcher that this idea is not profitable [12]. Contrary, there is an optimistic hypothesis that the opportunities are very promising regarding the lift behaviour is increased significantly in high angle of attack. The blades of vertical-axis turbine always change on angle of attack in every position rotation. Application this tubercle can delay the static stall that affected on delaying the dynamic stall performance. Therefore, the opportunity for increasing the vertical-axis turbine performance is big.

The bottom graph in Fig. 7 shows coefficient of drag for both original and modified foil. The drag performance for all foils are also increasing as increasing angle of attack from 0 to 8 degrees. The original foil of NACA 63(4)021 continue to increase slightly until 20 degrees. However for tubercle foil, the drag is higher than the original foil in 12 to 22 degrees. The tubercle foil from numerical simulation also higher than original foil at 12 to 16 degrees. This is also proof that the simulation results are as same as tubercle experiment behaviour. Starting from 22 to 30 degrees, the coefficient of drag is remain the same. The tubercle foil just slightly higher than the original when comparing both experimental data. Here it can be concluded that tubercle foil can produce better lift in high angle of attack but also having higher drag.

C. Fluid flow visualizations

Fig. 8 and 9 show pressure contour in foil surface and streamline contour around the foil. The pressure contour is used to evaluate the pressure different between bottom and upper surface of an foil. This is connecting with lift performance that generated by the foil. If the pressure of bottom surface higher than the upper surface, the foil is producing lift. The streamline contour is used to explain the vorticity than happen around the foil. Angle of attack 0 and 16 degrees are chosen to represent foil condition in pre-stall and post stall-regime in both original and modified foil.

Fig. 8 shows pressure contour in foil surface and streamline contour around the foil at 0 degree angle of attack. Both original and modified foil have a same pressure contour. The highest pressure happen in leading-edge section that marking with red colour. Then the pressure is decreased in centre of both bottom and upper surface location. After that, the pressure gradually increase to trailing-edge direction. In angle of attack 0 degree, both original and modified foil also have a same streamline contour. The tubercle profile at leading-edge section has no effect at this angle of attack.

Fig. 9 shows pressure contour in foil surface and streamline contour around the foil at 16 degrees angle of attack. The original foil has a different pressure contour compared with modified foil. The highest pressure happen in front of bottom surface than indicated with red colour. Then the pressure is lower at upper surface. However, in the Fig. 9 can be seen that in upper surface of original foil has pressure lower than modified foil that marking by dominant colour in blue to green. This is explained that the coefficient of lift for original foil higher than modified foil at 16 degrees angle of attack.

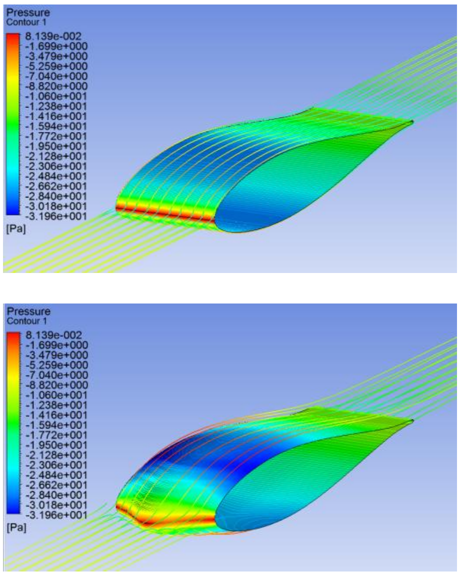


Fig. 8 Pressure and streamline contours at 0 degree AoA

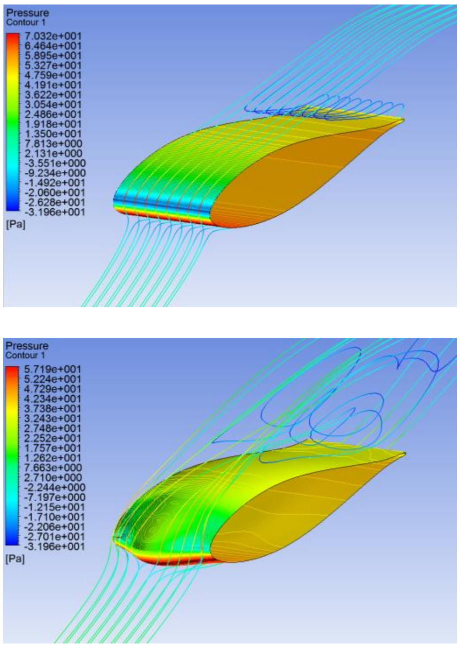


Fig. 9 Pressure and streamline contours at 16 degrees AoA

In angle of attack 16 degrees, both original and modified foil also have a different streamline contour. The original foil has a laminar streamline contour. However in the modified foil has a turbulence streamline contour. There is very big vortice behind the foil at the upper surface position. This phenomena explain that the modified foil has coefficient of lift lower than the original foil. This big vortice contribute to produce drag that reducing the lift performance.

IV. CONCLUSIONS

Foil simulation in pre-stall regime has succesfully done. The simulation results of original profile of NACA 63(4)021 is very consistent with the experimental data. However, further study is needed to make a good numerical model that could predict foil performance in post-stall regime. Tubercle profile on leading-edge foil section can delay static stall angle in high angle of attack. Therefore, vertical-axis turbine which is the foil profile modified with tubercle will become a good chance to increase the verticl-axis turbine performance by increasing the range operation of extracting energy in dynamic stall condition.

ACKNOWLEDGMENTS

The authors, would like to convey their appreciation to the Directorate General of Resources for Science, Technology and Higher Education; Ministry of Research, Technology and Higher Education, the Republic of Indonesia, which funding the current project partly under a scheme called The Education of Master Degree Leading to Doctoral Program for Excellent Graduates (PMDSU) under contract number 803/PKS/ITS/2018 and a scheme called Competitive Researche Program under contract number 845/PKS/ITS/2018.

REFERENCES

[1] Mukhtasor, *Introduction to Ocean Energy* (in Indonesian language), Surabaya: Indonesia, page 1-13, 2014.

[2] Instream Energy System, URL <http://www.instreamenergy.com/>, accessed date: 05.06.2018.

[3] LucidEnergy, URL <http://www.lucidenergy.com/>, accessed date: 05.06.2018.

[4] Mukhtasor, Susilohadi, Erwandi, Pandoe W., Iswadi A., Firdaus A. M., Prabowo H., Sudjono E., Prasetyo E., and Iluhade D., *Indonesia Ocean Energy Potential Resources* (in Indonesian language), Badan Litbang Kementrian Energi dan Sumberdaya Mineral (ESDM) and Asosiasi Energi Laut Indonesia (ASELI), 2014.

[5] D. Satrio, I. K. A. P. Utama, and Mukhtasor, *Vetical Axis Tidal Current Turbine: Advantages and Challenges Review*, Proceeding of Ocean, Mechanical and Aerospace -Science and Engineering- Vol.3: 64-71, 2016.

[6] D. S. Miklosovic, M. M. Murray, and L. E. Howle, *Hydrodynamic Experimental Evaluation of Sinusoidal Leading Edges*, Journal of Aircraft Vol. 44 No. 4, 2007.

[7] H. Johari, C. Henoch, D. Custodio, and A. Levshin, *Effects of Leading-Edge Protuberances on Airfoil Performance*, AIAA Journal Vol. 45, No. 11, 2007.

[8] W. Shi, M. Atlar, R. Norman, B. Aktas, and S. Turkmen, *Numerical optimization and experimental validation for a tidal turbine blade with leading edge tubercles*, Renewable Energy 96 42-55: Elsevier, 2016.

[9] D. Satrio, I. K. A. P. Utama, and Mukhtasor, *The influence of time step setting on the CFD simulation result of vertical axis tidal current turbine*, Journal of Mechanical Engineering and Sciences Volume 12, Issue 1, pp. 3399-3409, 2018.

[10] D. Satrio, I. K. A. P. Utama, and Mukhtasor, *Numerical Investigation of Contra Rotating Vertical-Axis Tidal-Current Turbine*, Journal of Marine Science and Application, 2018.

[11] P. Marsh, D. Ranmuthulaga, I. Penesis and G. Thomas, *The influence of turbulence model and two and three-dimensional domain selection on the simulated performance characteristics of vertical axis tidal turbines*, Renewable Energy 105 106-116: Elsevier, 2017.

[12] Z. Wang, and M. Zhuang, *Leading-edge serrations for performance improvement on vertical-axis wind turbine at low tip-speed-ratios*, Applied Energy 208 1184-1197, 2017.