Vortex Formation on Horizontal Axis Wind Turbine with Splitted winglets

Rizal Ibnu Syifa*, Setyo Nugroho
Power Plant Engineering, Electronic Engineering Polytechnic Institute of Surabaya, Surabaya, Indonesia, 60111
*rizalibnu95@gmail.com

Abstract
Horizontal axis wind turbine has developed to many combination. One of the combination is by adding variation in their blades such as variation of device. Many kind of devices that added to the blade such as addition splitted winglet to the blade. This additional device is the application device that added to the wings of recent aircraft that gives better performance than wings without this device. The purpose of using this device is to minimize the backflow air in their tip of blade. This research expectation is turbine will get better performance. However, the addition of the device do not increase the turbine performance. Comparing the turbine with the blade without splitted winglet, the turbine without splitted winglet gives better performance than the turbine that has blade with splitted winglet. Vortex formation in the splitted winglet device is one of phenomenon that cause the decrease performance of wind turbine. Vortex is air that flowing back to the device and makes the other flow velocity slower. In turbine without splitted winglet, the performance of Cl is greater than the other. When the air that hit the turbine is 4.5 m/s, the torque of the turbine with splitted winglet is 2.66 E-3 Nm. The turbine without splitted winglet gives better torque about 2.8 E-3 Nm. From variation that compared, additional of device not always give the better performance.

1. Introduction
Wind turbine is a device that is converting energy from wind energy to kinetics and then it conversed to make an electricity. The turbine performance affected by external and internal factor. The external factor come from the ambiance condition including weather. The internal factor that affected in turbine performance come from turbine component such as blades, tail, generator, and its transmissions. Blades make the rotation of turbine to make an electricity. This device can make the turbine’s performance become better by adding some variation. One of the variation that can added into the blades is splitted winglet.

Splitted winglet is the innovation from blended winglet that added by another winglet in the against direction. This additional variation is adapted from the recent aircraft. The purpose of adding this variation is to enhance the performance so that there are minimalize the back flow on the blade. From the research [1] the wings or aircraft with splitted winglet has better performance than wings of aircraft using blended winglet or single winglets.

Computational fluid dynamics simulation is part of process when designing product by some industry. Almost industries using fluid dynamics in their process to make a product [2]. Industries want to their product looks and performance before they process it into the factory. So that, Computational Fluid Dynamics simulation makes the construction of tools become efficient.
In this computation is limited by velocity of the air, sum of the rotor of turbine, pitch angle and angle of attack. This scope of problem is described in each sub in this paper.

2. Geometry
Geometry that used in this paper is rotor of wind turbine with three blades. Wind turbine rotor with three blades give better performance[3]. This three blades rotor show better performance with coefficient of performance Cp about 5.2 when the TSR in 4 until 5. TSR is tip speed ratio which show the ratio of speed on tip of blade to the velocity.

There are two kinds of blade in this research they are blade without winglet and blade with splitted winglets. In the previous research, The blade with single winglets give better performance [4]. Single winglets are used in the recent aircraft. The winglets advantage is to minimize the backflow on the tip of blade. So that, the lift increase and the drag decrease. This single winglets has dimension is the span angle is 70° and the length of the winglets is 2 cm.

The blade with splitted winglets dimension adapted form blade with single winglets a with additional other winglets in the against direction. The splitted winglets ratio is h2 is 0.4 times h1 [5]. Where h1 is the main winglets and h2 is the side winglets. The position of h1 or main winglets is as same as winglets in blade with single winglets with span angle 70°. The h2 or side winglets position is against the main winglets with span angle 52°.

Figure 1. Blade with different additional winglets (a) blade without winglets (b) blade with splitted winglets

Other additional method is using twist. Twist is the additional method in the segment of airfoil that construct blade with different angle of twist, dimension or chord length, and distribution of the segment. Twist give variable angle of attack through the length of the blade. Twist angle is an angle that given to the each segment of airfoil. Twist angle distribution is based on aerodynamic behavior of each individual airfoil at a certain angle of attack [6].

$$\beta (r) = \frac{2}{3} arctan \frac{R}{r \lambda} - \alpha_p$$

(1)

The airfoil variation is defined by the chord length. The chord length is the size of chprd in every segment of chord. The airfoil with different chord and twist angle is distributed based on the length of the blade [7]. The size of each blade is 12 cm and the hub radians is 3 cm.

$$c (r) = \frac{1}{\beta c_1} \sin^2 \left( \frac{1}{3} \tan^{-1} \left( \frac{R}{\lambda^3} \right) \right)$$

(2)

So, the rotor diameter is 30 cm. From the equation above [7], The segment distribution is counted on excel and described in table below. In the table 1 from research [4] we know that longer segment position from the hub, smaller chord we get. The variation of twist angle is different in each segment.
Table 1. Detail Of Geometry Data Input

| Segment | \( r \) (cm) | \( c \) (cm) | Twist \( \beta \) (°) |
|---------|--------------|--------------|----------------------|
| 1       | 3,20         | 3,80         | 28,13                |
| 2       | 4,14         | 3,59         | 22,06                |
| 3       | 5,09         | 3,32         | 17,53                |
| 4       | 6,04         | 3,04         | 14,11                |
| 5       | 7,00         | 2,79         | 11,46                |
| 6       | 7,95         | 2,56         | 9,35                 |
| 7       | 8,90         | 2,36         | 7,63                 |
| 8       | 9,85         | 2,18         | 6,21                 |
| 9       | 10,81        | 2,03         | 5,02                 |
| 10      | 11,76        | 1,89         | 4,00                 |
| 11      | 12,71        | 1,77         | 3,12                 |
| 12      | 13,66        | 1,67         | 2,36                 |
| 13      | 14,62        | 1,57         | 1,70                 |
| 14      | 15,62        | 1,48         | 1,08                 |
| 15      | 15,95        | 1,45         | 0,88                 |

3. Simulation

The simulation is using ANSYS Computational Fluid Dynamics package. ANSYS Computational Fluid Dynamics package is using divided into three step they are geometry making, mesh distribution and fluent for the calculation. Geometry making is use to make the domain of the simulating and the fluids flow. The mesh is used to distribute the calculation point. The Fluent is used to calculate the fluids flow in every point that made by Mesh. This simulation is using MRF method.

3.1. Geometry Preparation

The domain is using rotor of wind turbine that covered by the tube cylinder. The tube cylinder defined as the ambient environment for the air flow. The air flow trough the input front cylinder to the output in back cylinder. The cover of cylinder tube is defined as wall. The rotor of turbine defined as the object that hit by the air. The rotor is subtracted by boolean method. The use of boolean method is to trim the blades area and makes the cylinder volume less because of the rotor and makes a hole rotor shaped in the middle of tube cylinder.

3.2. Mesh Distribution

The mesh function is to makes some nodes point to calculate. The calculation is used to solve the problem in each nodes. In this paper, the mesh method is using global mesh with tetrahedron. In the figure 3 the mesh is more close when the nodes approach the blades. The different colour is caused by
rotation duplicate. The generate process gives average nodes 7.0 E+5 for every body geometry of simulation.

![Figure 3. Mesh Geometry](image)

3.3. Fluent Calculation

The calculation process is solving by ANSYS Fluent. The fluent using Computational Fluid Dynamics fluid dynamics analysis. This simulation process using Moving Reference Frame method or MRF method. This method is The fluids flow through the cylinder is defined by air with standard density. The air flow through the cylinder in rotational flow. The fluids flow with turbulence flow in k-omega. In the research [2] k-omega result is closer to the test result.

4. Result and Discussion

From the simulation of wind turbine using ANSYS, data that we can obtain are torque, lift force and drag force. Torque is the force in length of force that happen on rotor. Lift force is the force that makes the rotor rotate. Drag force is the force that it is the disturbance of rotor rotation. From the data that obtained, we must calculate it into lift coefficient and drag coefficient.

\[
C_L = \frac{F_L}{\frac{1}{2} \rho V_o^2 c} \quad (3)
\]

\[
C_D = \frac{F_D}{\frac{1}{2} \rho V_o^2 c} \quad (4)
\]

Lift coefficient is is coefficient that show the capability of airfoil to give lift force in the rotor. Drag coefficient is is coefficient that show capability of airfoil to give drag force in the rotor. The result of the simulation is given by data in the table below.

| Table 2. Data Result of Simulation |
|-----------------------------------|
| a. Turbine without winglets       |
| Torque                          |
| V (m/s) | Torque (Nm) | Cd            | Cl             |
|---------|-------------|---------------|----------------|
| 4.5     | 0.002806    | 8.97033E-05   | 0.346412       |
| 5       | 0.003663    | 0.000313371   | 0.363378       |
| 5.5     | 0.004454    | 0.000326399   | 0.363325       |
| 6       | 0.005661    | 0.000576013   | 0.383118       |

| b. Turbine with Splitted Winglets |
| Torque                          |
| V (m/s) | Torque (Nm) | Cd            | Cl             |
|---------|-------------|---------------|----------------|
| 4.5     | 0.00266     | 0.000270187   | 0.324787       |
| 5       | 0.003551    | 0.000618428   | 0.348624       |
| 5.5     | 0.004365    | 0.000556515   | 0.351798       |
| 6       | 0.005361    | 0.000692312   | 0.360047       |
From the table 2, the best torque is given by turbine rotor without splitted winglets. The coefficient lift is increasing when the velocity raise. The drag coefficient also increase when the velocity raise. This condition is proofed by figure 4 that show air flow vector on each blades. The figure 4 show blade a for blade without winglets and blade b for blade with splitted winglets. This two blades are simulated in the same condition with velocity of air 5 m/s. That two kinds of blade gives different vector that happened around the tip blades.

In the blade a the vectors show minimum blue vectors than the vectors that happened in the blade b. This blue vectors show backflow air in the tip of blades. The backflow air is the air that move back and hit another air flow. The air flow that hit will decrease the velocity of its flow. So, in the tip of blades makes the lift force decrease and increase the drag force. This condition makes the torque lower. In the wind turbine smaller drag that happen small engine that required [8].

![Figure 4. Vector of flow in blade tip in different additional winglets in velocity of 5 m/s (a) blade without winglets (b) blade with splitted winglets](image)

![Figure 5. Variation coefficient of lift to drag as function of velocity](image)
The graph in figure 5 that show lift to drag coefficient as function of velocity gives that the lift to drag coefficient is decrease as the increase of velocity in each typical of blade. The blue line is for blade without winglets named by blade a. The red line is for blade with splitted winglets named by blade b. The highest value of coefficient lift to drag is given by point in velocity of air 4.5 with 3.86 E+3 by blade a. The coefficient of lift to drag value is almost the same for blade a and b in the velocity of 6 m/s. The blade without splitted winglets give better performance than blade with splitted winglets in the lower air velocity.

5. Conclusion
This research is successfully by doing simulation about horizontal axis wind turbine using ANSYS Computational Fluid Dynamics package by using MRF method with result:
1. The additional of splitted winglets do not give better performance than blade without winglets.
2. The torque of the turbine with splitted winglet is 2.66 E-3 Nm when the velocity of air hit the turbine is 4.5 m/s.
3. The turbine without splitted winglet gives better torque about 2.8 E-3 Nm.
4. The backflow on the tip of blade the torque decrease.

Acknowledgement
Thank you to fluid laboratory-EEPIS for helping this research by providing data that needed in this research. Authors wish the data can be used for the next research.

References
[1] Sohail R, Helmut S, Abas A, George S, “Winglets-Multiobjective Optimization of Aerodynamic Shapes,” Department of Mechanical and Materials Engineering, MAIDROC Laboratory, Florida International University, Miami, FL 33174, USA
[2] Mohammad Moshfeghi, Ya Jun S, Yong H, “Effects of near-wall grid spacing on SST-K-ω model using NREL Phase VI horizontal axis wind turbine,” J. Wind Eng. Ind. Aerodyn., vol. 94, pp. 107–108, 2012.
[3] Tony B, David S, Nick J, Ervin B, “Wind Energy Handbook,” John Willey & Sons Ltd., UK, 2001.
[4] Akhmad K, “Experimental Study of Effects of blade pitch angle Against Horizontal Axis Wind Turbine Performance,” EEPIS Release, 2016.
[5] Louis B, “Split Blended Winglet” Patent Application Publication, Seattle, WA (US), 2012.
[6] Rainaldo G, Diego M, Juancarlo R, "Winglets Design and Analysis for Wind Turbine,” Florida International University Press. 2011.
[7] Soren Gundtoft, “Wind Turbines,” University College of Aarhus, 2009.
[8] Martin O. L. Hansen, “Aerodynamics of Wind Turbines,” Earthscan, UK, 2008.