JEMMME

JEMMME (Journal of Energy, Mechanical, Material, and Manufacturing Engineering)
Vol.4, No. 1, May 2019

ISSN 2541-6332 | e-ISSN 2548-4281
Journal homepage: http://ejournal.umm.ac.id/index.php/JEMMME

Numerical Simulation of the Effect of Wind Velocity on the Counter-Rotating Wind Turbines Performance

Y. Heru Irawan[a], M. Agung Bramantya[b]
[a]Department of Mechanical Engineering, Institut Teknologi Nasional Yogyakarta
Babarsari Street, Caturtunggal, Depok, Sleman, Yogyakarta, Indonesia
Phone: +6282255939844

[b]Department of Mechanical & Industrial Engineering, Universitas Gadjah Mada
Grafika Street No. 2, Yogyakarta, Indonesia
Phone: +6274631179

e-mail: bramantya@ugm.ac.id, yhirawan@sttnas.ac.id

Abstract

The counter-rotating wind turbines (CRWT) is a wind turbine model developed from a single rotating wind turbine (SRWT) model with a horizontal axis. CRWTs have two rotors rotating in opposite directions on the same axis. The purpose of this research is to investigate the effect of wind velocity on CRWTs performance with different axial distance ratio. The flow around CRWTs is simulated using computational fluid dynamic (CFD) with ANSYS Fluent. The simulation consists of two steps: obtaining the optimum rotation and rotor torque, respectively. These two values are used to calculate the mechanical power of the rotors. In this simulation, the wind velocities are 2 m/s; 3 m/s; and 4.2 m/s. The variations of axial distance ratio are 0.3; 0.5; 0.7; 0.8; and 1. The result of the simulation shows that the optimum ratio of the axial distance will change with the change of wind velocity. Regarding the wind velocity of 2 m/s, the optimal ratio of the axial distance is 0.5. Regarding the wind velocity of 3 m/s and 4.2 m/s, the optimal ratios of the axial distance are 1 and 0.8, respectively.

***Keywords**: axial distance; counter-rotating wind turbines; dual rotor; mechanical power; performance increase; wind turbines*

1. INTRODUCTION

Indonesia is one of the largest archipelago country in the world that has more than 17,000 islands. Energy consumption in Indonesia is still dependent on the fossil fuels. Based on these conditions, the energy consumption will continue to increase along with the in-crease of population. Fossil fuel energy demand continues to rise, while the resources of fossil fuels continues to decrease, causing energy deficiency in many islands (1). Wind energy is one type of renewable energy that grows rapidly. The utilization of wind energy is expected to reduce reliance on the fossil fuels. The use of wind energy in Indonesia is still not optimal compared to other countries.

Wind turbines is a tool to extract wind energy. Wind turbines model most widely used model is horizontal axis wind turbines (HAWT). Based on classical momentum theory, the maximum power coefficient or efficiency of HAWT, without any losses, is about 59% which is known as the Betz limit (2). However, the actual maximum power coefficient of HAWT is about 20-50% due to the losses of viscous and transmission loss (3). In order to improve the efficiency, the conventional model of HAWT which has only one rotor or single rotating wind turbines (SRWT) is developed into dual rotor wind turbines (DRWT) which drives two separate shafts. DRWT consists of two rotor movements: two rotor rotates in one direction (pro rotating wind turbines) and two rotor rotates in the opposite direction (Counter-rotating wind turbines or CRWT). CRWT will be the subject of this research.

CRWT is a wind turbine consisting of two rotors rotating in opposite directions on the same axis. CRWT has a more complex flow field than the conventional wind turbines. The wake of front rotor has the opposite direction to the front rotor rotation. Therefore, the rear rotor should rotate in the direction of the front rotor wake so that the wind energy can be converted optimally (4). Using classical momentum theory, Newman found that the maximum power coefficient of wind turbine having two rotors without any losses was around 64% (5). Recently, based on this result, the CRWT model has been studied extensively to obtain greater power than the conventional model (6-12).

One of the wind turbines research method is the numerical study of fluid flow. Numerical study of fluid flow is known as computation-al fluid dynamic (CFD). In this research, air flow through CRWT is simulated using computational fluid dynamic (CFD). The simulation process is done using ANSYS Fluent software. This simulation investigates the performance increase of CRWT by changing the wind velocity and axial distance between front and rear rotors.

2. Research Method

Computational fluid dynamic (CFD) is a method used to model fluid flow behavior using numerical approach to calculate and analyze a problem. In this case, a computer is used as a tool to perform calculations in the form of iteration. CFD consists of three governing equations that become the basis of the calculation process per-formed computer. Governing equations in CFD consist of conservation of mass, conservation of momentum, and conservation of energy. In addition to the three governing equations, we can also include additional models according to the case to be simulated. There are a variety of CFD software, in which ANSYS fluent is one of the software that is often used because of its comprehensive and easy-to-operate features.

2.1 Simulation Steps

The simulation consists of two steps. The first step is flow driven rotor simulation, which is used to calculate rotation speed. The second step is steady state simulation which is used to obtain torque. Rotation speed is multiplied by torque to calculate the mechanical power. Flow driven rotor simulation and steady state simulation use the same simulation parameters. The simulation steps are illustrated in Figure 1.

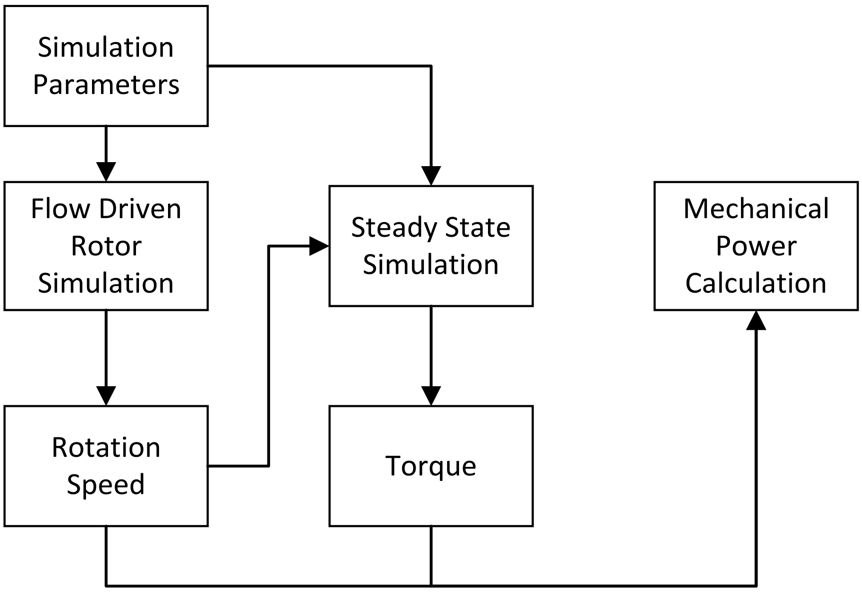

Figure 1. The Simulation Steps

Flow driven rotor simulations is a numerical process used to calculate the optimal angular velocity of rotor with a given wind velocity. This simulation calculates the rotor rotation in each timestep until the rotor rotation becomes stable. In order to use this simulation, we need to activate the function of 6dof solver in ANSYS fluent. The function of 6dof solver in ANSYS fluent uses force and moment of inertia of the rotor to calculate the rotational motion of the rotor with a predetermined center of gravity (13). The rotational motion that has been obtained is then used to determine the rotor position. Rotor position in each time step can be monitored by the motion history, through the motion history, rotor rotation can be calculated using the following equation 1:

$$\omega = \frac{\Delta\theta}{\Delta t} \quad (1)$$

The angular velocity (ω) is the magnitude of the angle change (Δθ) within a specified period (Δt). Motion history on ANSYS fluent records every movement of the rotor in each time step simulation, so by using motion history we can calculate angular velocity and rotation speed of the rotor in each time step simulation. The output of this simulation, the rotation of the rotor, is used as an input parameter to calculate the torque in the steady state simulation.

Steady state simulation uses multiple rotating reference frame method for modelling fluid flow in the area around the wind turbines rotor. The basic principle of multiple rotating reference frame meth-od is to move the fluid to the area around a moving object according to speed and direction of rotation of the object. Multiple rotating reference frame method is used to model certain CRWT on steady rotation to get the torque of each rotor. This simulation uses identical three-bladed rotors. Therefore, most of turbine researchers con-duct their simulations with only a single blade domain (periodicity of 120^o) 3. The torque of one blade with respect to the rotational axis is obtained by solving the equation in moving reference frame. The torque is multiplied by three (the number of blade). The mechanical power of each rotor in CRWT is obtained from equation 2, in which P is the mechanical power of each rotor, T is the torque of each rotor, ω is the angular velocity, and n is the rotation of each rotor. The mechanical power of CRWT is obtained from equation 4, in which PCRWT is the mechanical power of CRWT, P_{front} is the mechanical power of front rotor, and P_{rear} is the mechanical power of rear rotor.

$$P = T \times \omega \quad (2)$$

$$\omega = \frac{2\pi n}{60} \quad (3)$$

$$P_{CRWT} = P_{front} + P_{rear} \quad (4)$$

The configuration of CRWT rotors can be seen in Table 1. The front and rear rotors are made from balsa wood and consist of three blades each. The diameter of the front rotor and rear rotors are 230 mm and 400 mm, respectively. The airfoil type of each rotor is NACA 0012, with an upwind position. The rotation of front rotor is clockwise, whereas the rotation of the rear rotor is counter clockwise.

Table 1. Rotor configuration

Specification	Front rotor	Rear rotor
Blade number	3	3
Rotor diameter	230 mm	400 mm
Airfoil type	NACA 0012	NACA 0012
Blade material	Balsa wood	Balsa wood
Rotor position	upwind	upwind
rotation	clockwise	

2.2 Flow Driven Rotor Simulation Domain

The computational domain of flow driven rotor simulation is ex-tended in the axial direction (z) for a distance of 600 mm from the rear rotor in the upstream direction and for a distance of 1200 mm from the rear rotor in downstream direction. In the vertical direction (y), the cylindrical domain is extended for a distance of 400 mm from the center of each rotor to accommodate the rotor wake. Boundary condition consists of inlets (specific wind velocity and turbulence intensity), outlets (specific pressure), symmetry (the limit of the computational domain), wall (the boundary condition of rotor surfaces), and rotating region (interface boundary conditions to accommodate rotational motion). This computational domain con-sists of 766,283 tetrahedral meshes. The domain and its boundary condition are illustrated in Figure 2.

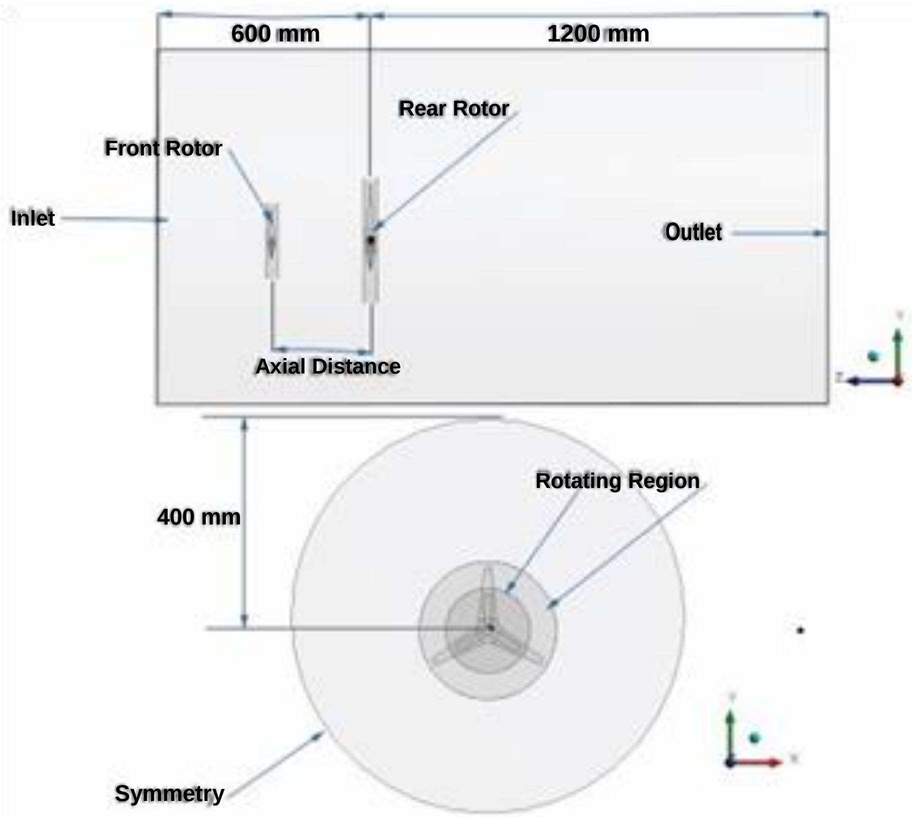

Figure 2. Computational domain of flow driven rotor simulation

Table 2. Set Up and Solution method of Flow Driven Rotor Simulation

Set Up	Solution Method
Solver	Transient Pressure-Based
Turbulence Model	SST k - ω
Spatial Discretization Gradient	Least Squares Cell Based
Spatial Discretization Pressure	Second Order
Spatial Discretization momentum	Second Order Upwind
Spatial Discretization k	Second Order Upwind
Spatial Discretization ω	Second Order Upwind
Transient Formulation	Second Order Implicit

The solution methods of flow driven rotor simulation are a second-order accurate model and a second-order upwind model for pressure discretization, as well as a momentum and turbulence model. A second-order implicit transient formulation is used as well. This simulation is done with 0.01 s time step size, with the total of 600 time steps. The set up and solution methods of flow driven rotor simulation can be seen in Table 2.

2.3 Steady State Simulation Domain

Computational domain of steady state simulation is 1/3 part of computational domain of flow driven rotor simulation. The boundary condition of this simulation consists of inlet, outlet, wall, and symmetry. Inlet is a boundary condition specifying the wind velocity and turbulence intensity. Outlet is a boundary condition determining the pressure. Wall is a boundary condition which represents the rotor surface. Symmetry is a boundary condition representing the limit of the computational domain. The computational domain consists of 311,282 tetrahedral meshes. The domain and boundary condition are illustrated in Figure 3.

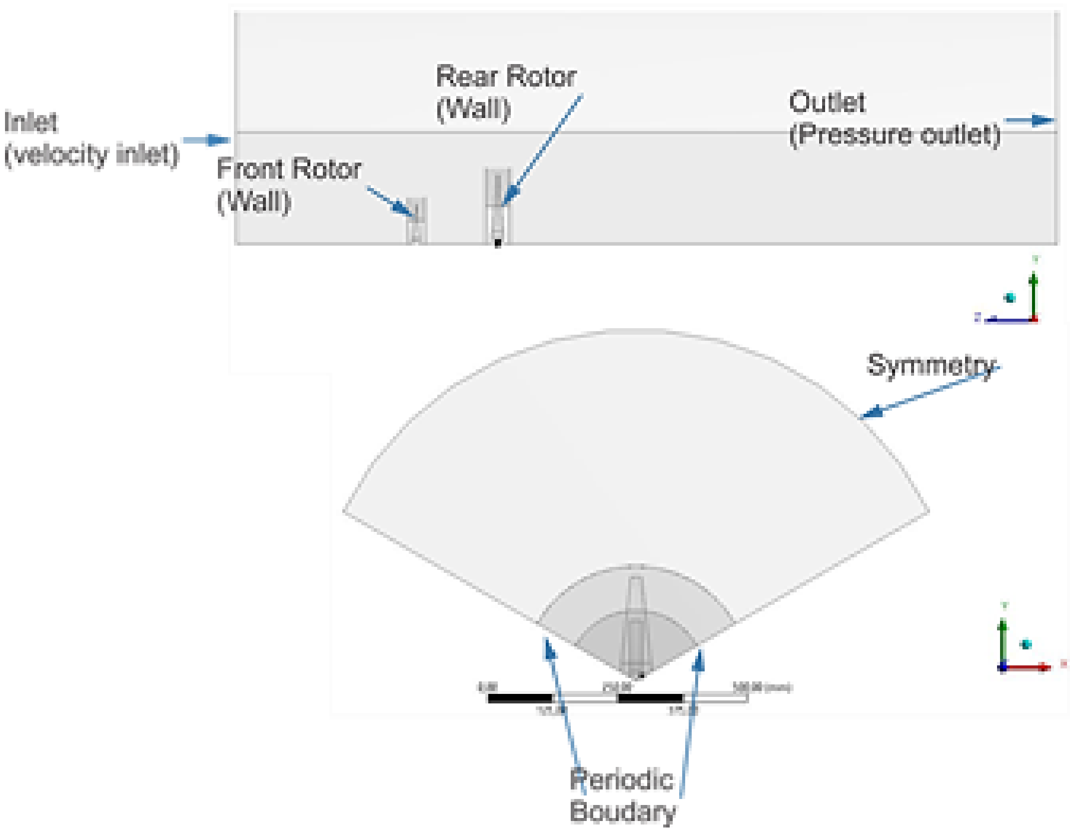

Figure 3. Computational domain of steady state simulation

Table 3. Set Up and Solution Steady State Simulation

Set Up	Solution Method
Solver	Steady Pressure-Based
Turbulence Model	SST k - ω
Spatial Discretization Gradient	Least Squares Cell Based
Spatial Discretization Pressure	Second Order
Spatial Discretization momentum	Second Order Upwind
Spatial Discretization k	Second Order Upwind
Spatial Discretization ω	Second Order Upwind

The solution methods of steady state simulation include a second-order accurate model and a second-order upwind model for pressure discretization in the momentum and turbulence model. This simulation is done in steady state through 1,000 iterations or the less than 1 x 10^{-5} residuals of continuity, momentum, and turbulence models to ensure convergence. The set up and solution methods for steady state simulation can be seen in Table 3.

3. Result and Discussion

3.1 Flow Driven Rotor Simulation Result

Simulations have been done using wind velocity of 2 m/s; 3 m/s; and 4.2 m/s. Comparison of axial distance (x) and front rotor diameter (d1) used are 0.3; 0.5; 0.7; 0.8; and 1. Figure 4 shows the optimum rotation of front rotor. The flow driven rotor simulation results for the wind velocity 2 m/s shows the front rotors with axial distance ratio of 0.3 produces the lowest rotation 642.95 rpm. The highest rotation 713.33 rpm is generated by the front rotor with the axial distance ratio 1. The front rotor rotation with the wind velocity of 2 m/s increased when the axial distance ratio is enlarged. Due to the enlargement of the axial distance ratio, the effect of the rear rotor which rotates in the opposite direction becomes smaller.

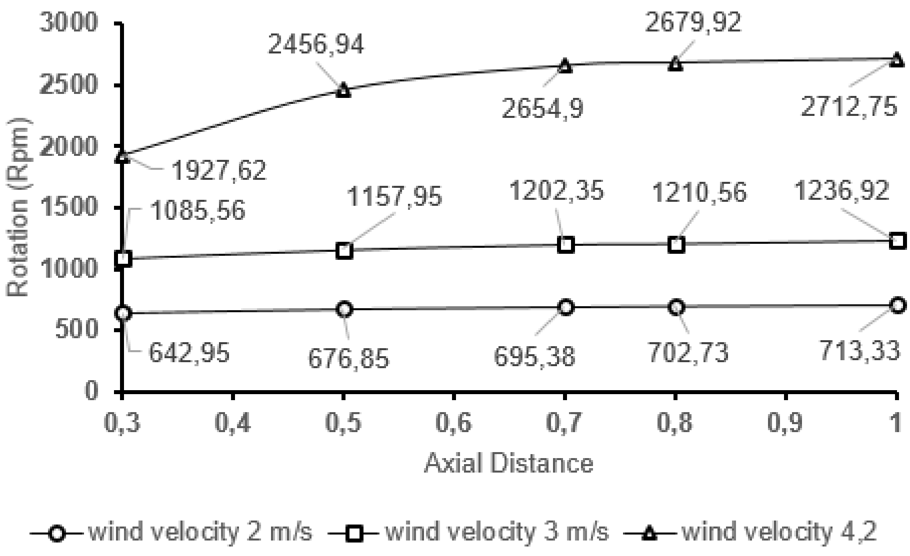

Figure 4. Optimum rotation of front rotor

Flow driven rotor simulation results on the wind velocity of 3 m/s and 4.2 m/s have a similarity with the simulation results on wind velocity of 2 m/s. The front rotors with axial distance ratio of 0.3 produces the lowest rotation, the simulation with wind velocity of 3 m/s produces 1085.56 rpm and the simulation with wind velocity of 4.2 m/s produces 1927.62 rpm. The front rotor with the axial distance ratio of 1 produces the highest rotation of 1236.92 rpm and 2712.75 rpm for the simulation with wind velocity of 3 m/s and 4.2 m/s. Based on the simulation results, it can be concluded that when the wind velocity increases, the rotation of front rotors will also increase because of the increasing kinetic energy when the wind velocity increases.

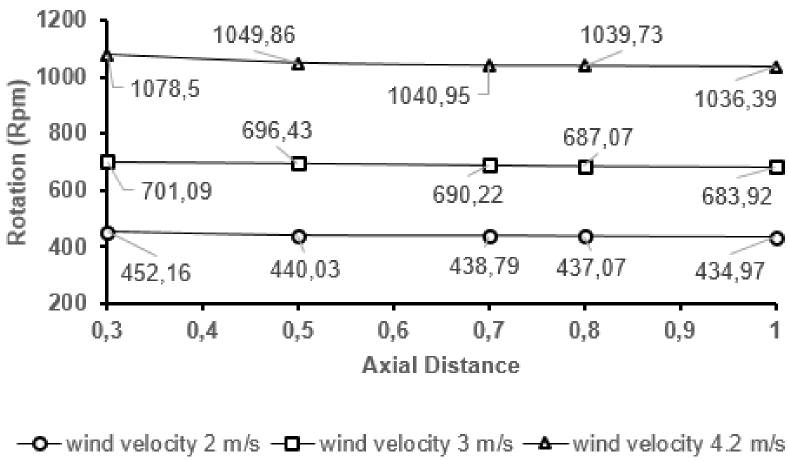

Figure 5. Optimum rotation of rear rotor

Figure 5 shows the optimum rotation of rear rotor. The lowest opti-mum rotation of 434.97 rpm; 683.92 rpm; 1036.39 rpm is produced by the rear rotor with axial distance ratio of 1 for the simulation with wind velocity of 2 m/s; 3 m/s; and 4.2 m/s, respectively. The rear rotor with axial distance ratio of 0.3 produces the highest rota-tion 452.16 rpm; 701.09 rpm; and 1078.5 rpm on the simulations with wind velocity of 2 m/s; 3 m/s; and 4.2 m/s, respectively. These results indicate that the rotation of the rear rotor in CRWT is in-versely proportional to the axial distance between the rotors. The greater axial distance increases the influence of the rear rotor's wake, thus making the rotation rate of the rear rotor decreases. The same as the front rotor, rear rotor always shows the addition of the opti-mum rotation when the wind velocity enlarges.

3.2 Steady State Simulation Result

The same simulation set up parameters is applied for the steady state simulation. The torque of each rotor is obtained by simulating the rotor from the optimum rotation until it stops (0 rpm). The torque of steady state simulation results are obtained from the process using equation 2 to obtain the power characteristic of each rotor. Mechanical power used is the maximum mechanical power that can be achieved by each rotor. Figure 6 shows the maximum mechanical power of front rotor at different wind velocity. Steady state simulation with wind velocity of 2 m/s produces the highest mechanical power of 0.0336 watt at the ratio of axial distance 1. At the same axial distance ratio, the front rotor produces the highest mechanical power 0.1175 watt and 0.3362 watt for steady state simulation with wind velocity of 3 m/s and 4.2 m/s. The front rotor with the axial distance ratio of 0.3 produces the lowest mechanical power at all of the simulated wind velocities. The same as the rotation characteristic, the characteristic of the mechanical power of front rotor will in-crease when the axial distance ratio enlarges.

Figure 7 shows the mechanical power of rear rotor on different wind velocity. Rear rotor with axial distance ratio of 0.5 produces the highest mechanical power 0.0943 watt at steady state simulation with wind velocity of 2 m/s. Steady state simulation with wind velocity of 3 m/s produces the highest mechanical power 0.3485 watt for axial distance ratio of 0.5 and 0.8. Simulation on wind velocity of 4.2 m/s produces the highest mechanical power 1.4129 watt on the axial distance ratio 0.8. Rear rotor with axial distance ratio of 0.3 produces the lowest mechanical power 0.0877 watt; 0.327 watt; and 1.0621 for steady state simulation with wind velocities of 2 m/s; 3 m/s; and 4.2 m/s, respectively. Based on the steady state simulation results, it can be concluded that the small axial distance ratio also produces a relatively small mechanical power on the rear rotor of CRWT.

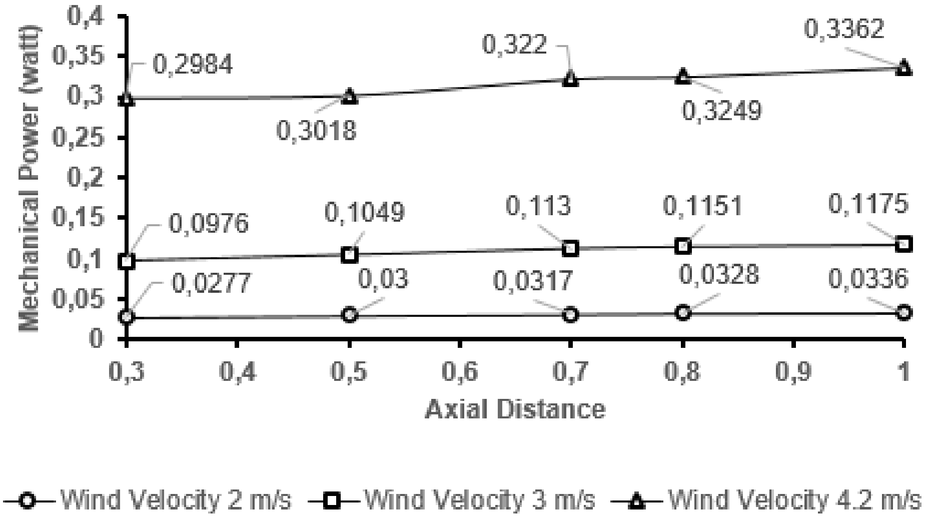

Figure 6. Maximum Mechanical Power of front rotor

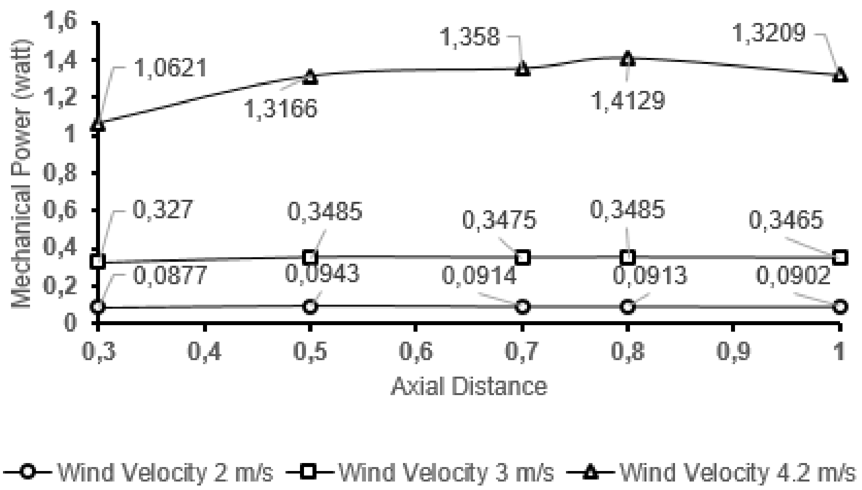

Figure 7. Mechanical power of rotor

The mechanical power of CRWT is calculated from the sum of the front and rear rotor's mechanical power. Figure 8 shows the mechanical power of CRWT. CRWT with axial distance ratio of 0.3 produces the lowest mechanical power at all of the simulated wind velocities. CRWT with axial distance ratio of 0.5 produces the highest mechanical power 0.1243 watt on the simulation with wind velocity of 2 m/s. CRWT with axial distance ratio 1 produces highest mechanical power 0.464 watt on simulation with wind velocity of 3 m/s. For simulation with wind velocity of 4.2 m/s, the highest mechanical power 1.7378 watt is generated by CRWT with axial distance ratio of 0.8. CRWTs consist of front rotors and rear rotors which have different characteristics regarding the axial distance between rotors, and thus it is necessary to find the proper axial distance to obtain the optimal performance of CRWT. Based on the simulation results, CRWT with small axial distance ratio pro-duces the total mechanical power which is also small. The optimal axial distance ratio of CRWT will change when the wind velocity changes. In other words, when the wind velocity changes, the optimal ratio of the axial distance of CRWT will also change.

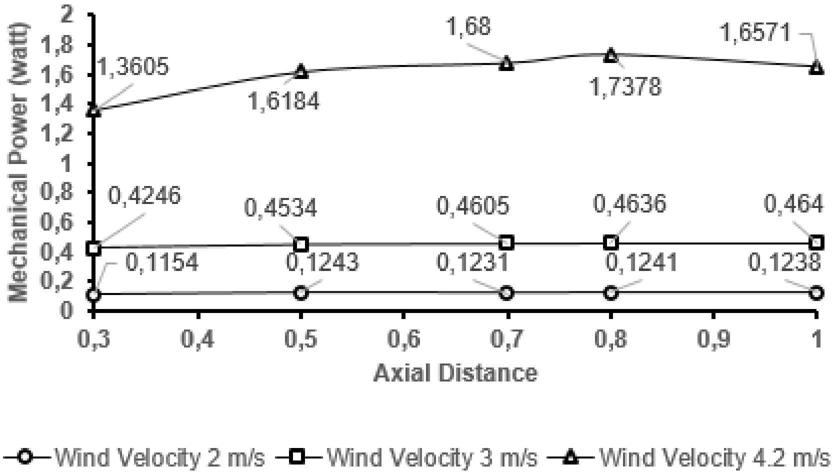

Figure 8. Mechanical Power of CRWT

4. CONCLUSION

Numerical simulation has been carried out on CRWT with the ratio of the axial distance (x) and front rotor diameter (d1) is 0.3; 0.5; 0.7; 0.8; and 1. All simulations have been done using wind velocity of 2 m/s; 3 m/s; and 4.2 m/s. This study has found that the characteristic rotation of the front rotor is proportional to the axial distance between the front and rear rotors. On the other hand, characteristic rotation of rear rotor is inversely proportional to that distance. The optimum ratio of the axial distance which will change with the change of the wind velocity. For the wind velocity of 2 m/s, the optimal ratio of the axial distance is 0.5. For wind velocity of 3 m/s and 4.2 m/s, the optimal ratio of the axial distance are 1 and 0.8.

REFERENCES

1. Martosaputro S, Murti N. Blowing the wind energy in Indonesia. Energy Procedia. 2014; 47: 273–82.
2. Betz, A. Wind-Energie und Ihre Ausnutzung durch Windmuehlen. Goettingen: Vandenhoeck & Ruprecht. 1926.
3. Lee S, Kim H, Son E, Lee S. Effects of design parameters on aerodynamic performance of a counter-rotating wind turbine. Renewable Energy. 2012;42:140–444.
4. Hoang AD, Yang C. Design and Performance Evaluation of a 10kW Scale Counter-Rotating Wind Turbine Rotor. J Korean Soc Mar Environ Safety. 2014;20(1):104–122.
5. Newman BG. Actuator-disc theory for vertical-axis wind turbines. Journal of Wind Engineering and Industrial Aerodynamic. 1983;15(1–3):347–55.
6. Irawan YH, Bramantya MA. Numerical simulation of the effect of axial distance between two rotors in counter-rotating wind turbines. In: Proceedings - 2016 2nd International Conference on Science and Technology-Computer, ICST 2016. 2017. p. 1–5.
7. Kumar PS, Bensingh RJ, Abraham A. Computational Analysis of 30 Kw Contra Rotor Wind Turbine. ISRN Renewable Energy. 2012;2012:1–5.
8. Santhana Kumar P, Abraham A, Joseph Bensingh R, Ilangovan S. Computational and experimental analysis of a counter-rotating wind turbine system. Journal of Scientific and Industrial Research. 2013;72(5):300–306.
9. Jung SN, No TS, Ryu KW. Aerodynamic performance prediction of a 30 kW counter-rotating wind turbine system. Renewable Energy. 2005;30(5) 631–644.
10. K. Appa, Energy innovations small grant (EISG) program (counter rotating wind turbine system), EISG final report, (california, US) 2002.
11. Sutikno P, Saepudin DB. Design and Blade Optimization of Intelligent Wind Turbine. International Journal of Mechanical and Mechatronics Engineering. IJMME-IJENS. 2010;11(01):1–15.

12. Lee S, Kim H, Son E, Lee S. Effects of design parameters on aerodynamic performance of a counter-rotating wind turbine. Renewable Energy. 2012;42:140–144.
13. Le TQ, Lee KS, Park JS, Ko JH. Flow-driven rotor simulation of vertical axis tidal turbines: A comparison of helical and straight blades. International Journal of Naval Architectur and Ocean Engineering. 2014;6(2):257–268.