Proceedings International Conference on Science and Technology (ICST)

e-ISSN: 2722-7375 pp: 271-276, Vol. 1, Juni 2020

Analysis Set-Up of Blower Utilized for Wind Turbine Performance Experiment

I Kade Wiratama[1)*], I Gusti Ngurah Yudhiyadi[1)], Nur Kaliwantoro[1)]

[1)*] University of Mataram, Jl. Majapahit No. 62, Mataram, Indonesia 83125,I.K.Wiratama@unram.ac.id

Abstract

Wind energy can be converted to kinetic energy by rotor wind turbine and the energy captured is depending on the variation of air velocity that occurring during the day. This paper presents some results of blower setting to get the best distribution of air velocity as well as aerodynamic performance wind turbine in laboratory. The Computational Fluid Dynamic (CFD) software Autodesk and turbulence model (*k-epsilon*) were applied to simulate distribution air velocity in different distance. The simulation was done by the three scenarios of blower distance variation blower in the difference air velocity, i.e. by 3 m/s. 5 m/s. 8 m/s and 10 m/s. The simulation test shows that the best distribution of air velocity was found in the blower set up at 1.5 m distance.

Keywords: Air velocity, blower, wind energy

1. Introduction

Power generated by wind turbine is the most important factor in determining the economic effectiveness of design blade wind turbine, and therefore designing a blade of wind turbine has to consider aerodynamic performance during operation. In wind turbine, the air flow is slowed down and widened. This effect causes a loss in the efficiency of the turbine. Hence, optimization of blade design by considering maximize power can reduce that effect and the corresponding loss in efficiency can be avoided [1].

Rotor wind turbine is a device used to convert kinetic energy of wind into mechanical power, whose performance is depending of variation of wind speed. This is becoming critical issue when the designed blades need to produce more power in different wind speed. Various methods can be used in order to identify the performance wind turbine and flow characteristics. The methods can be classified mainly as BEMT-based methods and computational fluid dynamics-based methods [2-5]. In BEMT calculation and CFD, the average power of mechanical power wind turbine is based on the variation wind speed in the range of 5-25 m/s as a basic calculation to measure the average power of aerodynamic performance design blade [6].

In testing wind turbine blades, it needs a wind-tunnel to test the aerodynamic performance of wind turbine blade design so that blade design can be investigated more thoroughly. However, testing with this method is costly and time consuming. To reduce the costs of using this method, wind turbine blade testing can use computer simulations. The purpose of the blower placement on the wind turbine blades tested in this test is to obtain an even distribution of wind speeds without a large reduction in wind speed and this is a very critical issue to get the best distance for the blower placement to the wind turbine blades to be tested.

In this paper it is analyzed set-up blower by using CFD to put in the right position of blower for examine aerodynamic performance design blade wind turbine. CFD simulations were performed to find the best an even distribution of wind speeds for different distance.

2. Material and Method

Simulation is done by using AutoDest CFD based on the *Navier-Stokes* equations. The solution of the *Navier-Stokes* equations is a flow velocity. It is a field, since it is defined at every point in a region of space and an interval of time. The *Navier-Stokes* equations play a key role in computational fluid dynamics (CFD). By creating model in the 5 blower the simulation run in the variation wind speed of 3 m/s, 5 m/s, 8 m/s and 10 m/s respectively

In the solution method for CFD simulation, the turbulence model used is mostly using the k-ε model and the Shear Stress Transport Model (SST) k-ω model. In the turbulence model with the k-ε model, this model has good stability and precision for turbulent fluid flow with high Reynolds number values, but this model is not good for use in simulations with rotational effects. The standard version of the k-ε model more accurately predicts the distribution of flat flow rates and also gives better predictions about the characteristics of boundary layers in large pressure gradients [7,8].

Then in analyzing a fluid flow stages are carried out, namely determining the external volume of the flow pattern in detail (x, y, z) in each direction, determining the material external volume and determining the boundary conditions, namely the conditions of the limits of an external volume. In general, boundary conditions consist of two types of inlet and outlet. Inlet is usually defined as the place where fluid enters the specified external (volume) domain. Various conditions are defined at this inlet ranging from speed, composition, temperature and pressure. Whereas at the outlet is usually defined as a condition where the fluid is out of the domain or in a CFD application is the value obtained from all variables that are defined and extrapolated from the previous point or cell.

3. Results and Discussion

The simulation was done by making 3 – D model pattern blowers for five blowers and input boundary condition to get result for the best distribution wind speed in the z-distance position of gradient air velocity. The results of simulation are presented in the some figures to show the distribution of air velocity in the three different positions of rotor wind turbine by 1 m, 1.5 m and 2 m z-distance position rotor wind turbine respectively. Some figures of distribution variation air velocty againt distance position of rotor wind turbine can be seen in the through figure below.

Figure 1 show the influnce of distance on 1 meter rotor wind turbine position againt distribution gradient air velocity in the variation of air velocity by 3 m/s, 5 m/s, 8 m/s and 10 m/s respectively.

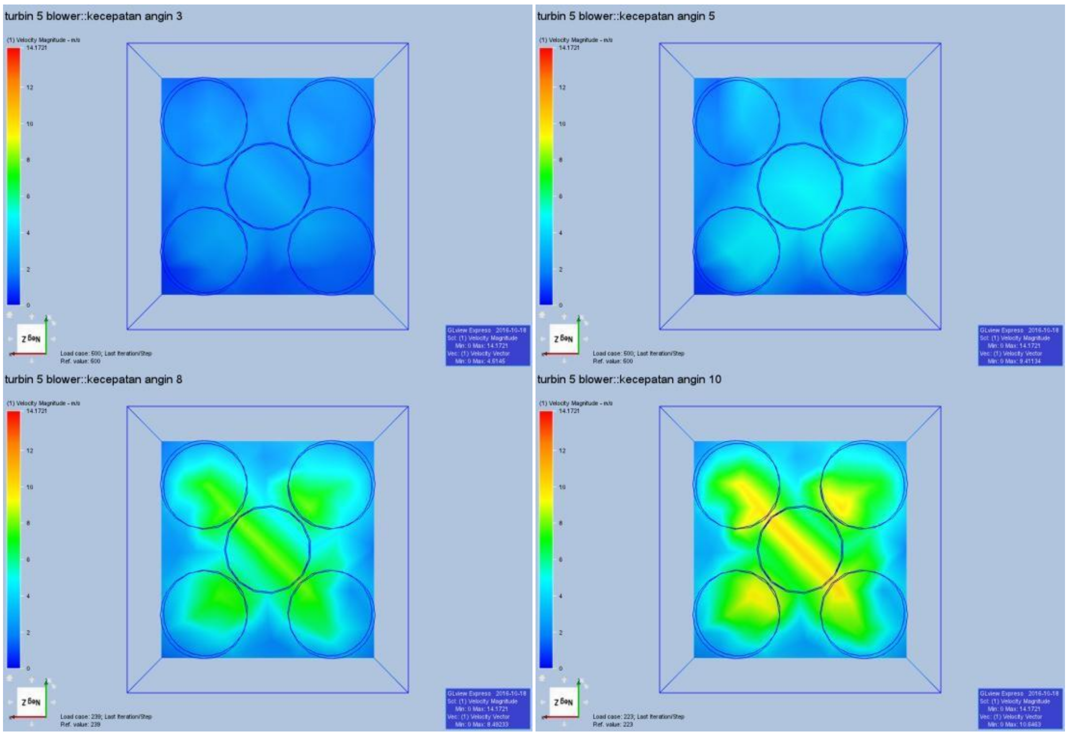

Figure 1. Distance on 1 meter rotor wind turbine position against blower

Figure 2 show the influnce of distance on 1.5 meter rotor wind turbine position againt distribution gradient air velocity in the variation of air velocity by 3 m/s, 5 m/s, 8 m/s and 10 m/s respectively.

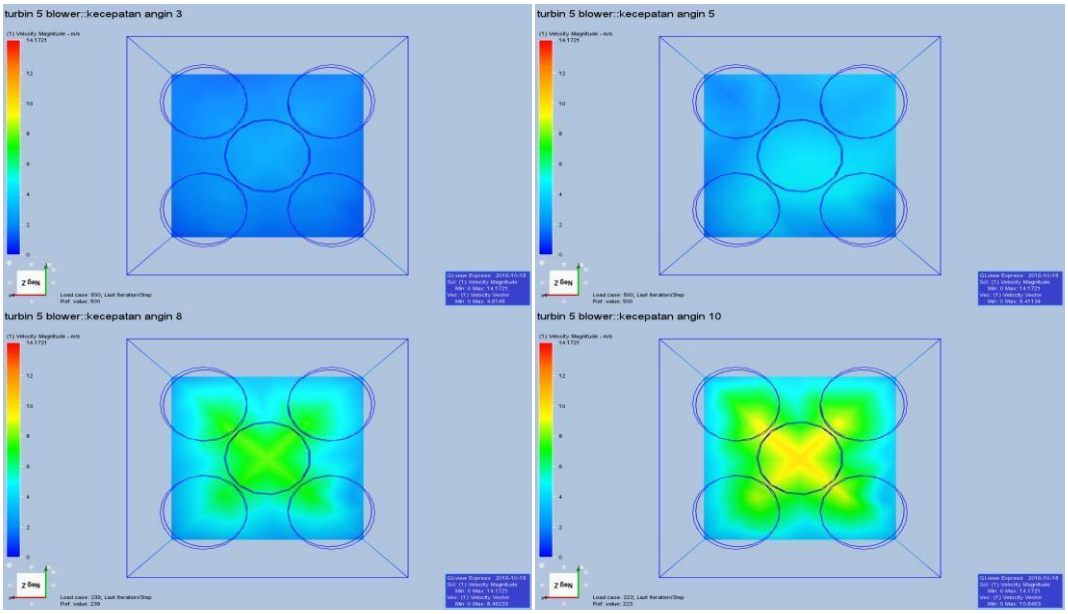

Figure 2. Distance on 1.5 meter rotor wind turbine position againt blower

Figure 3 shows the influnce of distance on 2 meter rotor wind turbine position against distribution gradient air velocity in the variation of air velocity by 3 m/s, 5 m/s, 8 m/s and 10 m/s respectively.

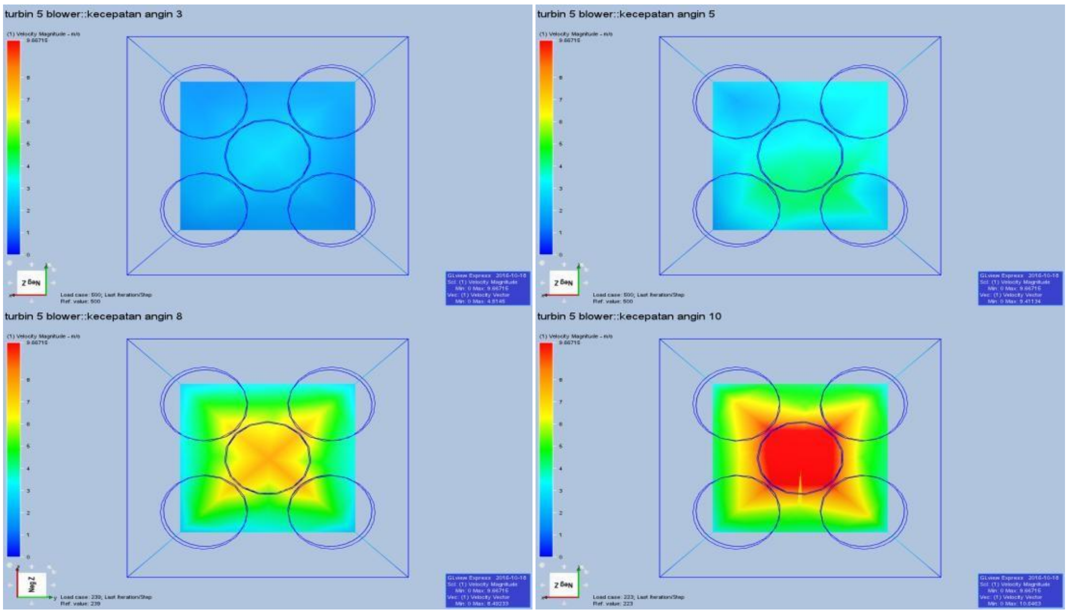

Figure 3. Distance on 2 meter rotor wind turbine position againt blower

From the results simulation it was found that five blowers to provide variations in wind speed was made 1.5 m distance from the tested wind turbine. This can be seen from the figure 2 which is in distance 1.5 m of set up blower is found that distribution air velocity is the best.

Figure 4 through figure 6 show the average air velocity at different distance of 1 m, 1.5 m and 2 m respectively at variation air velocity. The average air velocity at 1.5 and 2 m for all variations air velocity are nearly similar for every single scenario air velocity as can be seen in the through of Figure 4 to Figure 6

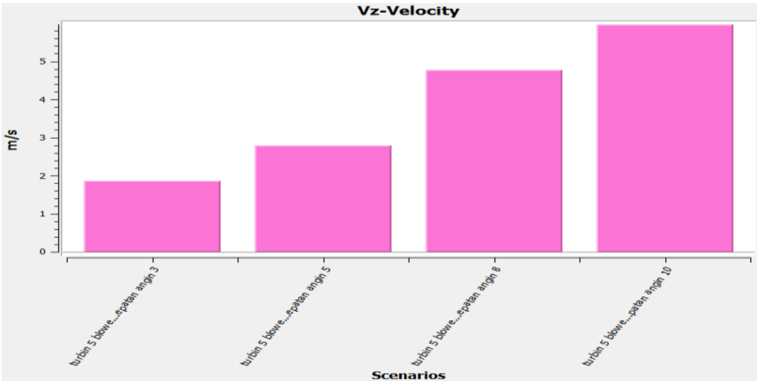

Figure 4. Average air velocity at distance position 1 m

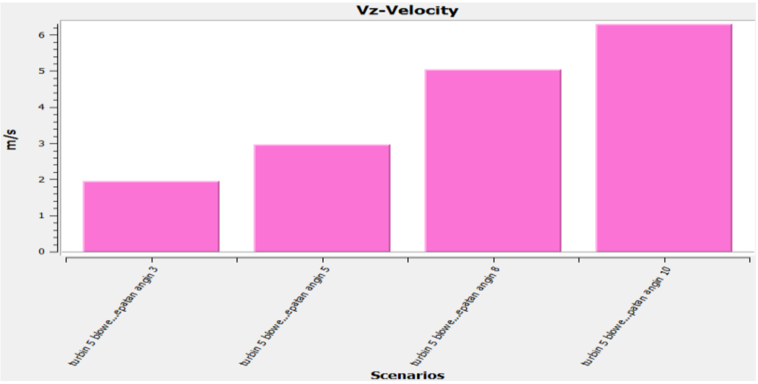

Figure 5. Average air velocity at distance position 1.5 m

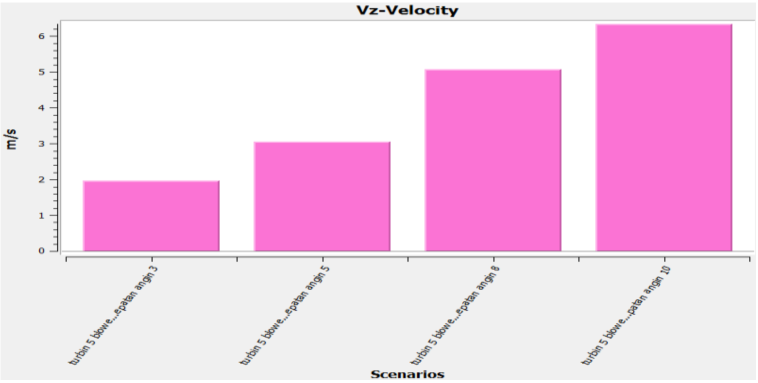

Figure 6. Average air velocity at distance position 2 m

4. Conclusion

Based on the results of the analysis and the description of the discussion, it can be conlusion that the simulation by the three scenarios of distance variation of position five blower in the difference air velocity by 3 m/s. 5 m/s. 8 m/s and 10 m/s respectively show that in distance 1.5 m of set up blower was found that distribution air velocity was the best and the average air velocity at 1.5 and 2 m for all variations air velocity are nearly similar for every single scenario air velocity.

Acknowledgements

The first author thanks to the Ministry of Research, Technology, and Higher Education of the Republic of Indonesia for the financial support of this research grant and paper.

References

[1] Ueno, H.,and Serada, T. 2002. Wind turbine generation system and photovoltaic power generation system. *NTT Power Build Facil J*, 39(231),42-45.
[2] Wiratama,I.K. 2014. Validation of AWTSim as aerodynamic analysis for design wind turbine blade, *Journal Applied Mechanics and Materials* ,. 493,105-110
[3] Maheri, A. and Isikveren, A. T. 2010. Performance prediction of wind turbines utilising passive smart blades: approaches and evaluation. *Wind Energy*, 3, 255-265.

[4] Moriarty, P. J., and Hansen, A. C. 2005. *AeroDyn theory manual*. National Renewable Energy Laboratory, Technical report NREL/TP-500-3688
[5] Snel, H. 2003. Review of aerodynamics for wind turbines, *Wind Energy*, 6, 203–211.
[6] Wiratama, I.K. 2012. Aerodynamic design of wind turbine blades utilising nonconventional control systems, Dissertation Northumbria Univeristy .
[7] Rocha. P. A. C., Rocha. H. H. B., Carneiro. F. O. M., da Silva. M. E. V., Bueno. A. V., 2013. *k-ω SST (Shear Stress Transport) Turbulence Model Calibration : A Case Study on a Small Scale Horizotal Axis Wind Turbine*. Laboratory of Aerodynamics and Fluid Mechanics (LAero), Campus do Pici, Bl. 718, Fortaleza, CE 60440-554, Brazil
[8] Wilcox, D.C., 2006. Turbulence Modeling for CFD, DCW Industries, Inc., 5354 Palm Drive, La Canada, California 91011.