

Simulation of Horizontal Side Rotor Performance Using Ansys Program Version 18.0 For Dual Rotor Wind Turbine Model

Mulyadi, Hasanuddin, Waskito and Syahrul

Jurusan Teknik Mesin, Fakultas Teknik, Universitas Negeri Padang, Indonesia

Article Info

Article history:

Received June 18th, 2018

Revised Aug 28th, 2018

Accepted November 16th, 2019

Keywords:

Wind Turbine

Wind Speed

Maximal Pressure

Blade

ABSTRACT

The dual rotor wind turbine is a wind turbine which has horizontal and vertical sides. The data obtained in the form of average wind speed, the speed of rotation of the horizontal and vertical vanes. Researchers want to create simulations of multi-rotor blade performance on the horizontal side using Ansys application version 18.0. The purpose of this research is looking at changes in wind speed in the outlet area with wind speed sign at the inlet area is 4 m/s. Researchers want to see maximum wind pressure at the inlet area, pressure on the area of the blade, the pressure at the outlet area. The results obtained show that the wind speed in the area of the inlet is of 4 m/s changed to 3.99237 m/s at area outlets. Maximum pressure in the inlet area is of 0.133612 Pa. turn into 457,528 Pa. area blade windmills, to the outlet pressure area turn into 0 Pa.

Corresponding Author:

Mulyadi,

Jurusan Teknik Mesin, Fakultas Teknik, Universitas Negeri Padang

Jln. Prof. Dr. Hamka Air Tawar, Padang (25131), Sumatera Barat, Indonesia

Email: mulyadiset06@gmail.com

1. INTRODUCTION

The use of plants using fuel oil and coal in remote areas is usually not economical, because of the scale of generation and the high cost of fuel [1]. The energy crisis in Indonesia can be overcome by using alternative energy. The use of wind energy is the most effective use of renewable energy throughout the world because wind resources are always available wherever they are. One of the renewable energy sources that can be used on a small scale is wind energy [2]. Regions in the world based on WWEA data (World Wind Energy Association) find wind potential at high speed, which is 10 -13 m / sec [3]. Based on LIPI data, the wind potential in the west coast of Sumatra island (including West Sumatra) is in the range of 2.5 to 4.0 m / s with a power capacity of up to 10 kW [4].

Basically, the wind generated energy cannot be directly used, therefore a machine that can convert wind kinetic energy into mechanical energy is needed. This tool is called a wind turbine or windmill [3]. Wind power plants convert wind power into electrical energy using windmills or wind turbines [5]. Wind turbines have different models and shapes, these differences can be seen from the axis which consists of vertical and horizontal axes. From the results of testing in the lab that the pump will be fully lifted optimally in variations in the number of blades 4 and 6 with speeds of 2.65 and 3.49 m / s [6].

To simulate the performance of wind turbines researchers used Ansys 18.0 CFD application. Computational Fluid Dynamic (CFD) is the study of how to predict fluid flow in heat transfer of chemical reactions and other phenomena by solving mathematical equations [7]. In connection with this problem, the researcher wants to conduct a research entitled "Performance Simulation of Multi-Blade Rotor on Horizontal Side using Ansys Version 18.0 Program for Planning Dual Rotor Wind Turbine Model". The aim that the researcher wants to achieve is to know the changes in wind speed and the pressure on the inlet and outlet area. And the power produced by wind turbines is based on theoretical calculations.

2. METHOD

The type of research used in this study is numerical method, in this study the author examines the location of the field of wind exposure, changes in wind speed and pressure changes on the blade of the windmill, this research is a kind of semi-experimental research (numerical fluid dynamics calculation) where the results want

to know is the power that is generated theoretically, the flutter of the wind after the collision with the Blade windmill with the initial wind speed is 4m / s. researchers also want to know the pressure on the windmill blade. This study uses the Ansys version 18.0 application. The material used is a windmill blade 3D image drawn using the Autodesk Inventor Professional 2018 application. Then it was analyzed using Ansys Fluent version 18.0.

3. RESULT AND DISCUSS

The research was conducted on June 8, 2018 - January 2019. The results of the study experienced changes in wind speed in the outlet area, the initial wind speed of 4 m / s experienced a change in wind speed to 3.99237 m / s. See Figure 1. and Figure 2. This change in speed is caused by the resistance that the wind experiences to the blade causing the wind speed to wane.

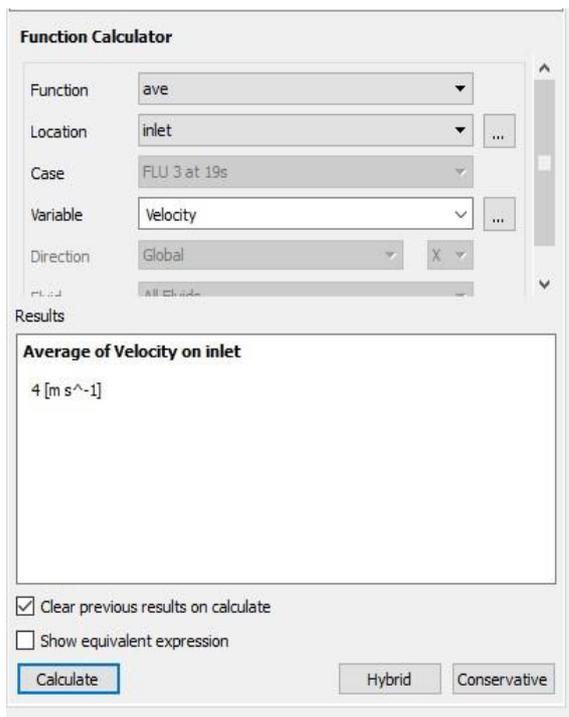


Fig. 1. Value of Inlet Speed

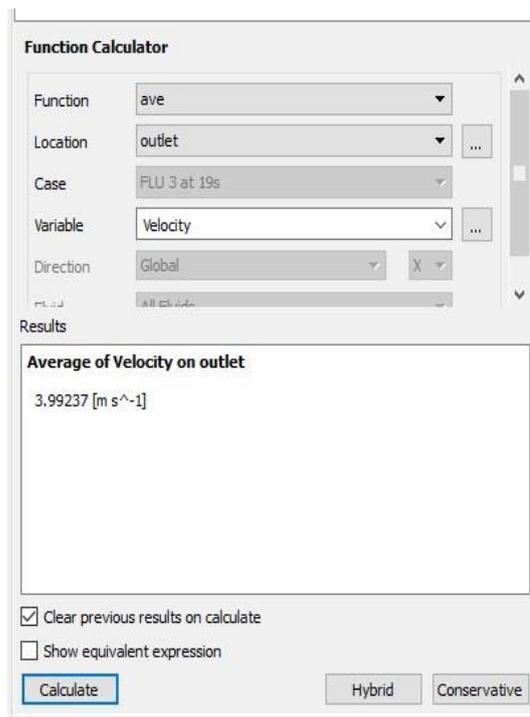


Fig. 2. Value of Outlet Speed

Changes in wind speed in the blade area based on Testing using the Ansys 18.0 application can be seen in Figure 3. This data is a change in wind speed that occurs in the blade area with wind speed according to the default legend view. The highest wind speed is shown in letter A in figure 3. The lowest wind speed is shown in letter B in figure 3.

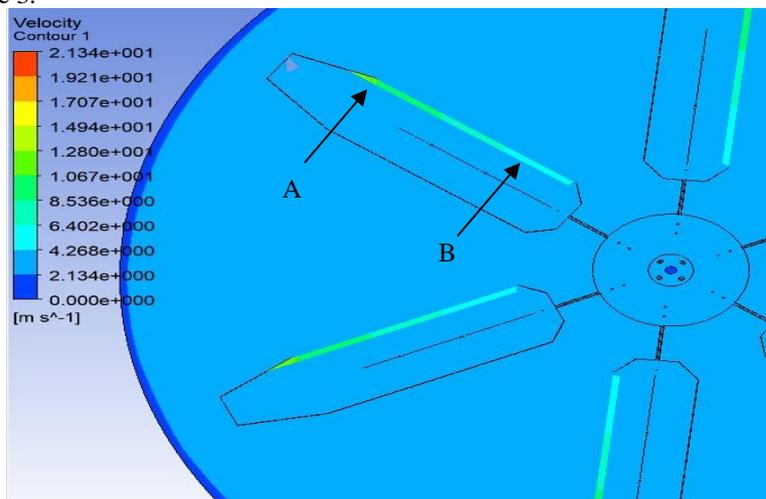


Fig. 3. Changes in Wind Speed on Blade

The result of the pressure analysis that the researchers analyzed in the Ansys 18.0 application was the pressure that occurred in the inlet area. The pressure that the researcher gets is equal to 0.133612 Pa. see picture 3. This is the maximum pressure in the inlet area.

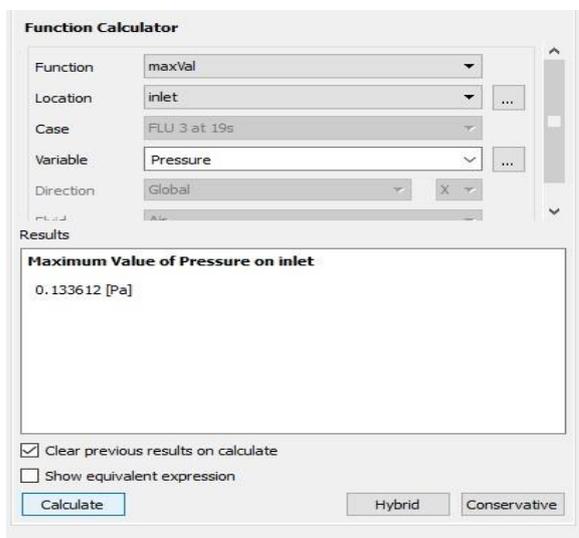


Fig. 4. Maximum Pressure in the Inlet

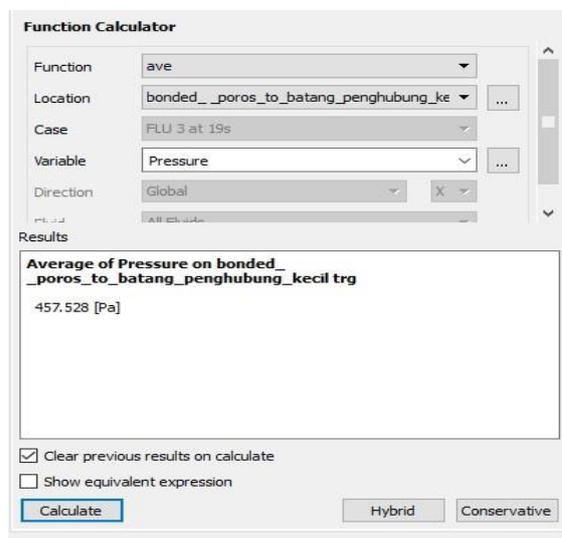


Fig. 5. Pressure on Blade

The pressure that occurs in the windmill blade area based on the application of Ansys 18.0 is 457,528 Pa. The pressure that occurs at the outlet area is equal to 0 Pa. See figure 4. The pressure obtained based on the research using the application of any 18.0 is 457,528 Pa, converted to 0.00457528 bar.

4. CONCLUSION

This research can be concluded some of the initial wind speed is 4 m / s at the inlet changes to 3.99237 m / s. The maximum velocity in the resulting plane area is 15.3185 m / s. The maximum pressure obtained in the inlet area is 0.133612 Pa. While the pressure generated in the inlet area is 0.0523853 Pa. Techniques that occur in the outlet area are 0 Pa.

REFERENCES

- [1] M Nur Syafe'i, "Rancang Bangun Simulasi Turbin Air Cross Flow," Universitas Negeri Padang, 2016.
- [2] F. Aryanto, I. M. Mara, and Made Nuarsa, "Pengaruh Kecepatan Angin dan Variasi Jumlah Sudu Terhadap Unjuk Kerja Turbin Angin Poros Horizontal," vol. 3, no. 1, pp. 50–59, 2013.
- [3] F. H. Napitupulu and F. Mauritz, "Uji Eksperimental Dan Analisis Pengaruh Variasi Kecepatan Dan Jumlah Sudu Terhadap Daya Da Putaran Turbin Angin Vertikal Axis Savoniu Dengan Menggunakan Sudu Pengarah.," *J. Din.*, vol. II, no. 12, p. 11, 2013.
- [4] Hasanuddin and Hendri Nurdin, "Unjuk Kerja Kincir Angin Transmisi Roda Gigi Planiter Differensial untuk Peningkatan Revolutionary Energy dan Konversi Potensi Angin Menjadi Tenaga Listrik," no. January, 2019.
- [5] R. Sumiati and A. Zamri, "Rancang Bangun Miniatur Turbin Angin Pembangkit Listrik Untuk Media Pembelajaran," *J. Tek. Mesin*, vol. 3, no. 2, pp. 1–8, 2013.
- [6] A. Sifa, T. Endramawan, I. Maolana, and A. R. Muhammad, "Pengujian Variasi Jumlah Blade Fiberglass Kincir Angin Tipe Horizontal untuk Pemompa Air Garam," vol. 8, no. 2, pp. 89–97, 2018.
- [7] A. M. Harahap, "Analisis Numerical Aliran Fluida pada Rumah Vortex Turbine dengan Water Dept 200 mm dan Inlet Velocity 0.20 m/s Menggunakan Aplikasi Ansys Fluent Versi 16.2," Padang, 2017.