CFD analysis of vertical axis wind turbine using ansys fluent paper

by Mark Finne

Journal of Physics: Conference Series

PAPER • OPEN ACCESS

CFD analysis of vertical axis wind turbine using ansys fluent

To cite this article: A A Afif *et al* 2020 *J. Phys.: Conf. Ser.* **1517** 012062

CFD analysis of vertical axis wind turbine using ansys fluent

A A Afif, P Wulandari*, and A Syahriar

Electrical Engineering, University of Al Azhar Indonesia, Jakarta, Indonesia.

*Email: putri.wulandari@uai.ac.id

Abstract. Renewable Energy Resources are increasing in a few years, this is due to the increasing increase in environmental pollution and fossil fuels which are increasingly depleted. Both bridges and toll roads actually have wind speeds that can be used and used as electricity. But there is a problem to put the power plant on the bridge or the toll road, which is quite limited land, so to install the Horizontal Axis Wind Turbine will be very difficult. Therefore, Vertical Axis Wind Turbine is used as an alternative. VAWT is used to create power plants that can use wind from vehicles on toll roads and bridges to make electricity. To be able to take advantage of the wind around the place, Computational Fluid Dynamics (CFD) is needed to optimize the design of the turbine.

1. Introduction

Renewable Energy Resources are increasing in a few years, this is due to the increasing increase in environmental pollution and fossil fuels which are increasingly depleted. many types of renewable energy such as Bioenergy, Geothermal, Solar, Hydropower, and Wind. Among other resources, wind resources are inexpensive alternative energy sources and this has led to much research being carried out so that the use of wind power generation technology can increase [1]. The world has enormous wind power potential that can be used as a power plant. There are two types of wind turbines are horizontal wind turbines and vertical wind turbines [2].

Wind is renewable energy that is very easy for everyone to use. Both bridges and toll roads actually have wind speeds that can be used and used as electricity. But there is a problem to put the power plant on the bridge or the toll road, which is quite limited land, so to install the Horizontal Axis Wind Turbine will be very difficult. Therefore, Vertical Axis Wind Turbine (VAWT) is used as an alternative. VAWT is used to create power plants that can use wind from vehicles on toll roads and bridges to make electricity. To be able to take advantage of the wind around the place, Computational Fluid Dynamics (CFD) is needed to optimize the design of the turbine.

CFD, drawn from various scientific disciplines, fluid mechanics and heat transfer, also find their way into other uncharted fields in processes, chemical engineering, civil, and the environment. Construction of new and improved system designs and optimization is carried out on existing equipment through applications to improve efficiency and lower operating costs. With fears of global warming and increasing world population, engineers in the power generation industry rely heavily on CFDs to reduce development and strengthening costs. This computational study is currently being carried out to address issues related to technology for clean and renewable energy and meet the challenges of strict regulation of emissions control and a substantial reduction in environmental pollutants. CFD simulations can be done on software such as Ansys Fluent, Ansys CFX, Openfoam, and Hypermesh [3].

2. Methods

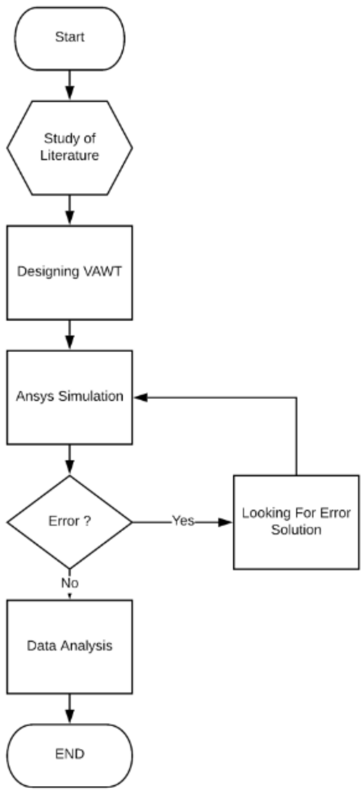

Figure 1. Research Method CFD Analysis of VAWT

Figure 1 shows the research method for CFD analysis from VAWT. Literature study is the preparation stage, such as searching for journals and papers related to Computational Fluid Design (CFD) on Vertical Axis Wind Turbines (VAWT). The next step is to design VAWT, the design used is a design created through the Design Modeller that has been provided by ANSYS. After designing vertical axis wind turbines, VAWT will be simulated to determine the speed, pressure and airflow of this turbine. But not always this simulation will run smoothly, sometimes there will be an error that occurs, if an error occurs then the location of the error must be found and find a solution, if the solution has been found it will be simulated again. If there is no error, then the data analysis phase will continue, the simulation results will be analysed by VAWT's speed, pressure and airflow.

Figure 2 shows the 2D design of VAWT, This turbine has a length of 1.5 meters and a radius of 30 cm for each blade. The blade has an angle difference of 120 degrees. To analyse this design we need to design it in AutoCAD or the Design Modeller that has been provided by ANSYS. To see the results of CFD calculations we can use 3D or 2D designs.

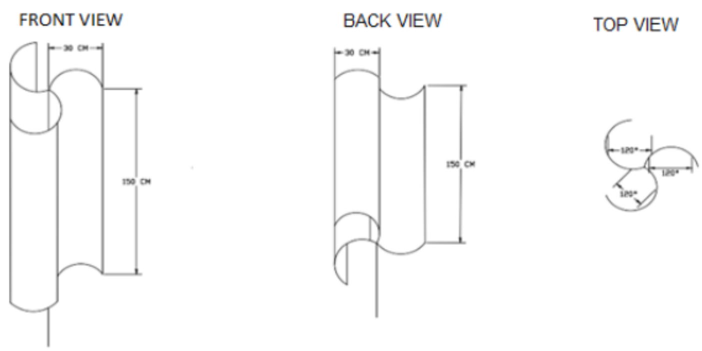

Figure 2. Designed 2D of VAWT

Figure 3. Designed 3D of VAWT

Figure 3 shows the 3D Design of VAWT which is designed using a Design Modeller ANSYS. This design will also be simulated ANSYS to determine turbine pressure. CFD analysis is performed using Fluid Flow (Fluent) provided by ANSYS. The first step to do is import Geometry from software such as SolidWorks or AutoCAD. In addition to importing from design software, we can also design it through the Designer Design provided by ANSYS.

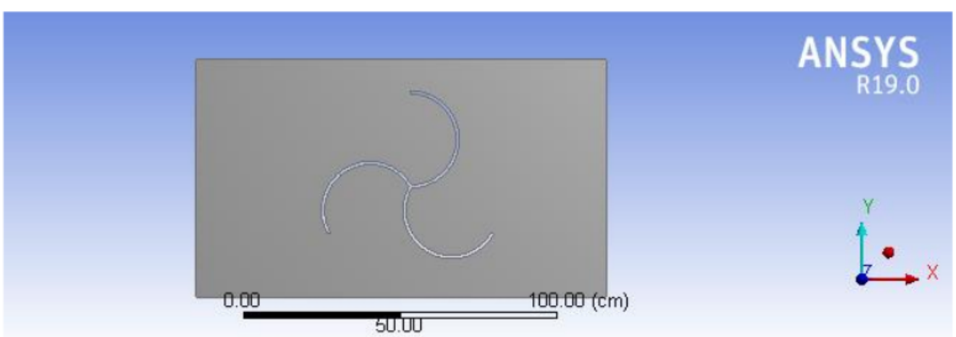

Figure 4. Designer 2D Top View VAWT Using Design Modeler

Figure 4 shows 2D top view of VAWT design which is designed using Design Modeler. After making geometry or importing it we must do the Meshing process to analyse fluent. To do the meshing process we can directly choose Mesh on the fluent flow. After that the meshing process will run according to the existing geometry.

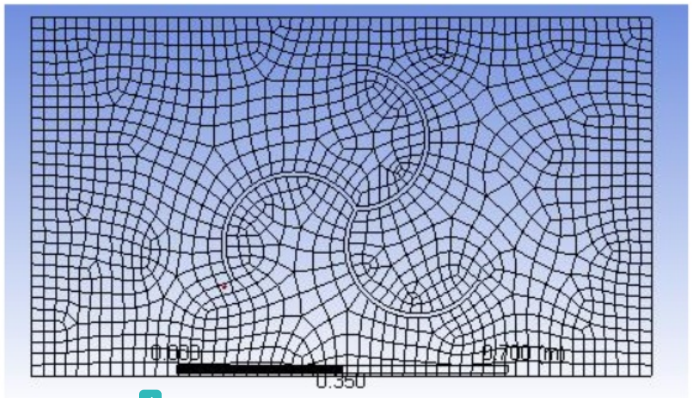

Figure 5. Meshing Geometry on ANSYS

Figure 5 shows Meshing Geometry on ANSYS Fluent. After all the processes are completed, then setup to get the solution that want. In this paper the results are shown in Figure 6, Figure 7, and Figure 8.

3. Result and Analysis

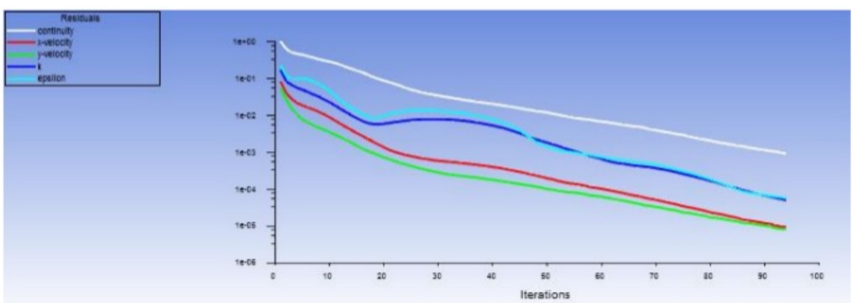

Figure 6. Iterations

Figure 6 shows the number of iterations performed by the solvent. The number of iterations specified in the solvent is 200 to achieve converged, but it turns out that the solution has converged in 95 iterations. After converged we can see Velocity & Pressure Contours.

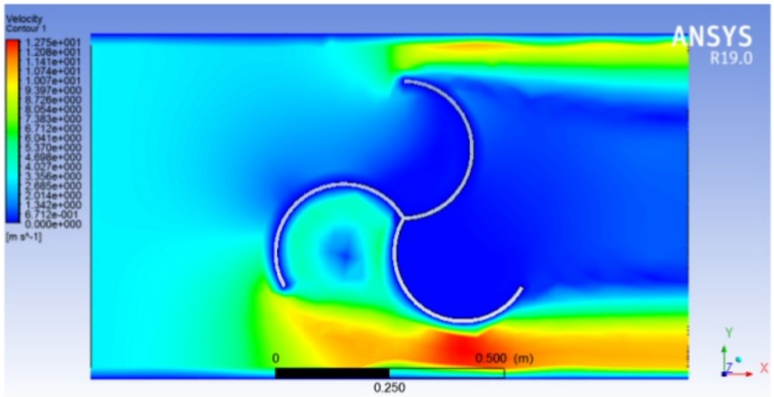

Figure 7. Velocity Contours

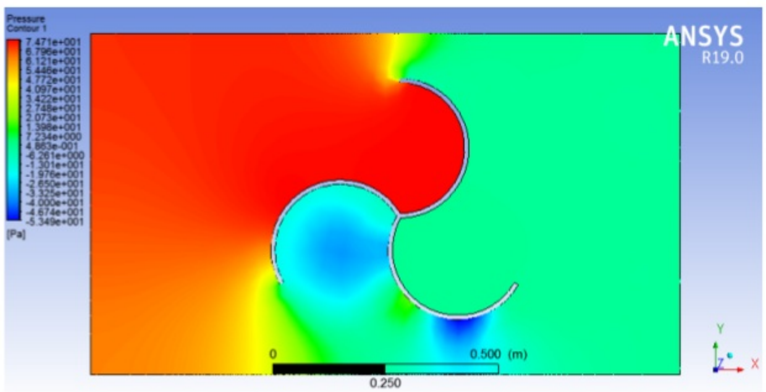

Figure 8. Pressure Contours

In this case, input speeds of 3 m/s are used. Based on Figure 7, it can be seen that the turbine moves at a minimum speed of 0 m/s to 12.75 m/s. and it can be seen also that this turbine has a minimum pressure of -53 Pa and also has a maximum pressure of 74 Pa and also the pressure on each side of the turbine is not the same.

In fluid dynamics, Bernoulli's law says that the increase in fluid velocity will also cause a decrease in pressure or potential fluid energy. From figure 8 it can be seen that the upper blade has a pressure higher than the other blade. This is because the upper blade is the blade that is directly affected by the wind speed so that the pressure on the blade is greater. When comparing Figure 7 and Figure 8 it can be seen that the greater the velocity of the fluid will cause the pressure of the fluid to be smaller.

4. Conclusions

In the simulation that has been done, the results are in accordance with Bernoulli's law which say that the increase in fluid velocity will also reduce fluid pressure. By using CFD simulation, blade manufacturing can be done easily without having to make physical items first. By using CFD the developer can simulate it and analyse it first to get optimal results before it is made in its physical form.

References

[1] M. M. A. Bhutta, "Vertical axis wind turbine – A review of various," Renewable and Sustainable Energy Reviews, vol. 16, p. 1926–1939, 2012.

[2] M. S. H. a. S. K. Afaq, "Design and analysis of a straight bladed vertical axis wind turbine blade using analytical and numerical techniques," Ocean Engineering, vol. 57, p. 248–255, 2013.
[3] J. Tu, G.-H. Yeoh and C. Liu, Computational Fluid Dynamics : A Practical Approach.
[4] M. H. Ali, "Experimental Comparison Study for Savonius Wind Turbine of Two & Three Blades At Low Wind Speed".
[5] K. Suffer, "Modeling and Numerical Simulation of a Vertical Axis Wind Turbine Having Cavity Vanes".
[6] J. Mccosker, "Design and Optimization of a Small Wind Turbine".
[7] M. R. Castelli, A. Englaro and E. Benini, "The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD," 2011.
[8] A. Alaimo, "3D CFD Analysis of a Vertical Axis Wind Turbine," 2015.
[9] A. M. Chowdhury, H. Akimoto and Y. Hara, "Comparative CFD analysis of Vertical Axis Wind Turbine in upright and tilted configuration," 2015.

CFD analysis of vertical axis wind turbine using ansys fluent paper

ORIGINALITY REPORT

7%	3%	5%	2%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

1	Ying Wang, Sheng Shen, Gaohui Li, Diangui Huang, Zhongquan Zheng. "Investigation on Aerodynamic Performance of Vertical Axis Wind Turbine with Different Series Airfoil Shapes", Renewable Energy, 2018 Publication	1%
2	Submitted to University of East London Student Paper	1%
3	Submitted to University of Liverpool Student Paper	1%
4	Asad Muneer, Mohammad Bilal Khan, Umer Bin Sarwar, Zia Ahmad Khan, Muhammad Sohaib Badar. "CFD analysis of a Savonius Vertical axis wind turbine", 2015 Power Generation System and Renewable Energy Technologies (PGSRET), 2015 Publication	1%
5	James Pace. "Spatial Analysis, Policy, Planning, and~Alternative Energy Production", Planning	1%

and Socioeconomic Applications, 2009
Publication

6	link.springer.com Internet Source	1%
7	www.mdpi.com Internet Source	1%
8	Masoud Ghasemian, Z. Najafian Ashrafi, Ahmad Sedaghat. "A review on computational fluid dynamic simulation techniques for Darrieus vertical axis wind turbines", Energy Conversion and Management, 2017 Publication	<1%

Exclude quotes	On	Exclude matches	Off
Exclude bibliography	On		