RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING VOL. 2 NO. 1
(2021) 183-192
© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Aerodynamic Characteristic Of Vortex Bladeless Wind Turbine: A Short Review

Muhammad Wafi Sabab[1], Sofian Mohd[1*]

[1]Department of Aeronautical Engineering, Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.01.021
Received 02 March 2021; Accepted 28 March 2021; Available online 15 April 2021

Abstract: The production of electricity from wind generators have been widely used. It attracts a great share in the electrical power production market worldwide as wind turbine industry. There are few types of wind turbine which is Horizontal Axis Wind Turbine (HAWT), Vertical Axis Wind Turbine (VAWT), and vortex bladeless wind turbine. The vortex bladeless wind turbine is a vortex-induced vibration resonant wind turbine which considered as most environmentally friendly wind turbine. As for a start-up in Spain, Vortex Bladeless Ltd have been experimenting on how to bring the maximum efficiency of power conversion of the bladeless wind turbine as it is still not fully done with zero commercialization. Hence, the purpose of the study is to analyze few shape and configurations of the vortex bladeless wind turbine using computational fluid dynamics (CFD) and next identifying the significance of aerodynamic characteristic on the bladeless wind turbine as to determine the optimum shape of the vortex bladeless wind turbine. The method used in this study is run the 2D and 3D model of vortex bladeless wind turbine that designed from SolidWorks in CFD simulation using Ansys-Fluent software. The two main design of 3D model vortex bladeless wind turbine are varied into five different configuration each. The data from simulation is interpreted into several ways which one of it from vertex average of velocity and graph extracted from Fast Fourier Transform (FFT) process. The most optimum configuration of the vortex bladeless wind turbine was obtained which is Design A4 by having the highest value f_v/f_n equal to 0.62, as well as the aerodynamic characteristic and parameters that affect the selection of the configuration. Suggestion for future work is to detail the mass portion of the shape of the bladeless wind turbine and use enhanced way in obtaining data from FFT.

Keywords: Vortex Bladeless Wind Turbine, FFT, Aerodynamic Characteristic

1. Introduction

This study involves designing several Computer Aided Design (CAD) models for the vortex bladeless wind turbine configuration and further performing Computational Fluid Dynamics (CFD) simulation testing for the wind turbine system by using Ansys-Fluent software. In this study, a

Corresponding author: sofian@uthm.edu.my
2021 UTHM Publisher. All right reserved.

theoretical approach was introduced to design a small-scale of vortex bladeless wind turbine and the performance of the designed bladeless wind turbine was evaluated using the CFD simulation method. Other than that, the study also elaborate on how Horizontal Axis Wind Turbine (HAWT) and Vertical Axis Wind Turbine (VAWT) operate so that the difference in working principal between vortex bladeless wind turbine and conventional wind turbine can be obtained. The operation of the vortex bladeless wind turbine involving vortex shedding and vortex induced vibration. Vortex is a rotating region of fluid such as tornado or whirlpool. These vortices are generally created at a moving boundary due to the shear resulting from the no slip condition. In general, vortices move with the fluid and are dispersed by the action of viscosity of the fluid. As for vortex induced vibration, it is a vibration that can occurs anytime when a sufficiently bluff body or the cylinder is exposed to a fluid flow that produces vortex shedding at or near a structural natural frequency, f_n of the body or the cylinder [1].

2. Materials and Methods

The methodology of the study will consist of the development of proposed design of vortex bladeless wind turbine for 2D and 3D model using SolidWork and geometry sketching in Ansys Fluent. The CFD simulation is then run for both type of models. During this method, the 2D model analysis will be only a solid circle (plan view). This to give a better understanding in performing the CFD simulation and relate it with real life event through its validation.

2.1 2D Model

Figure 1 shows the 2D model of the wind turbine with its solution domain's dimension. The diameter, d of the model equal to 1m resulting to value of Reynold number, Re equal to 100 as the value of velocity, U, density of air, and viscosity is equal to 1m/s, 1kg/m^3, and 0.01kg/m-s, respectively.

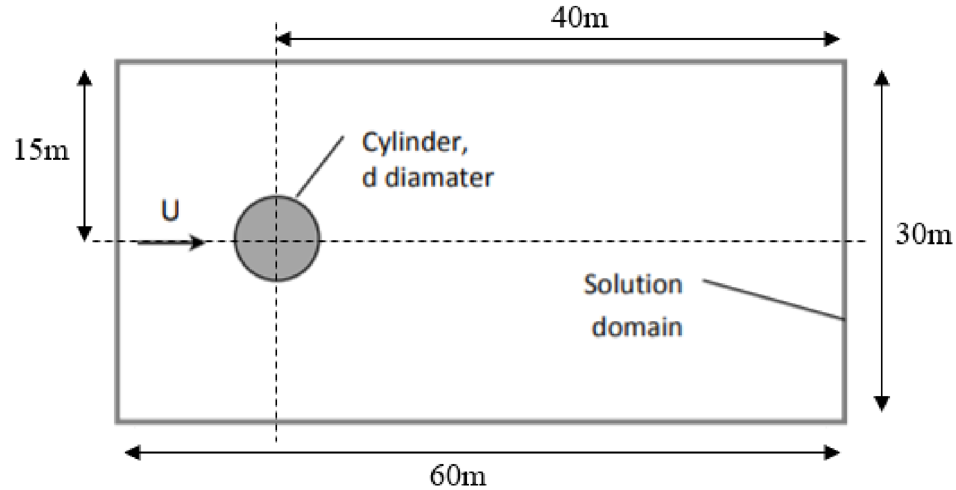

Figure 1: Plan view of 2D model of vortex bladeless wind turbine

As for the CFD simulation there are 4 main steps that have to be followed which is solution domain defining, mesh generation, setup or solving, and lastly is result (chapter 3). Firstly, solution domain. The solution domain defines the abstract region where the solution is calculated. The choice of solution domain shape and size can affect the solution of the problem where the smaller size of domains need less iterations to solve the problem and bigger size of domains need more time to find the solution. The solution domain of the 2D model is defined as in Figure 1 with rectangular domain shape and Figure 2 shows the wireframe arrangement of the solution domain.

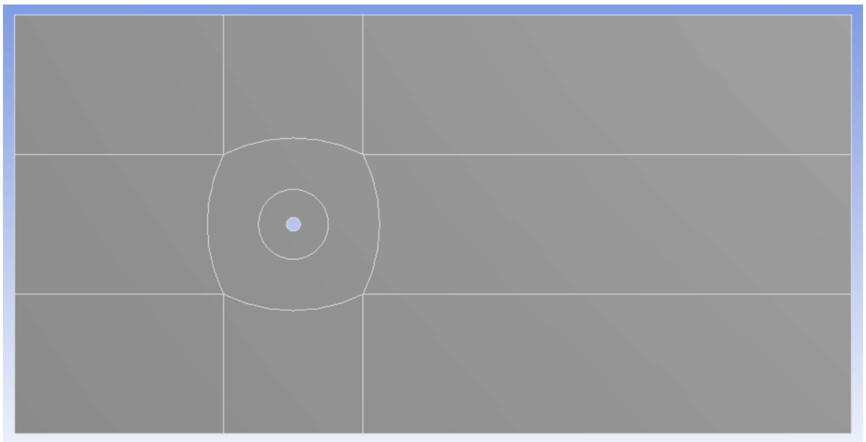

Figure 2: Wireframe arrangement of solution domain

Next, mesh generation. According to [2], best domain and grid quality is the rectangular domain with smooth quadrilateral grid. Figure 3 shows the rectangular domain with smooth quadrilateral grids and name selection of the solution domain.

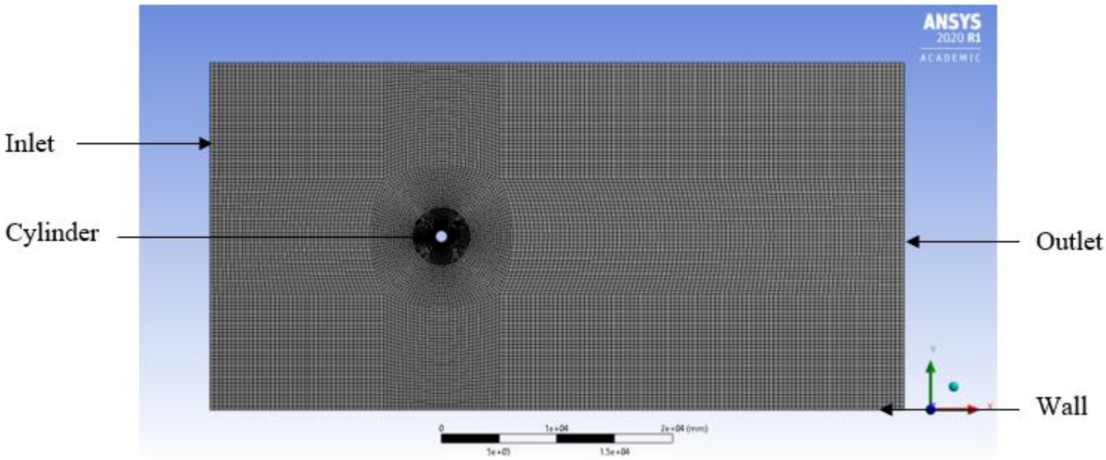

Figure 3: Rectangular domain with smooth quadrilateral grids for the meshing

As for setup, general setup of the models is defined. The setup covering the general condition of the fluid flow (model), definition of the material, condition of the boundary, solution method, report definition, initialization and run calculation. The value of velocity, U, density of air, and viscosity is defined equal to 1m/s, 1kg/m^3, and 0.01kg/m-s, respectively resulting to Re equal to 100. For solution method, SIMPLE is chosen for Pressure-Velocity Coupling. SIMPLE is an acronym for Semi-Implicit Method for Pressure Linked Equations [2]. It is widely used to solve the Navier-Stokes equations and extensively used by many researchers to solve different kinds of fluid flow and heat transfer problems. In report definition, lift coefficient,C_l and drag coefficient,C_d graphs are chosen to be plotted and before run calculation the number of time step is set as 800 with time step size of 0.2.

2.2 3D Model

The proposed shape of the vortex bladeless wind turbine is break in two configuration known as Design A and Design B. For each design or shape proposed, the parameter for each shape which is diameter and length were varied in order to obtain the most desired configuration the bladeless wind turbine according to the aerodynamic characteristic. Figure 4 a) and b) show the shape of Design A and Design B respectively. Design B involving taper ratio (TR).

a) b)

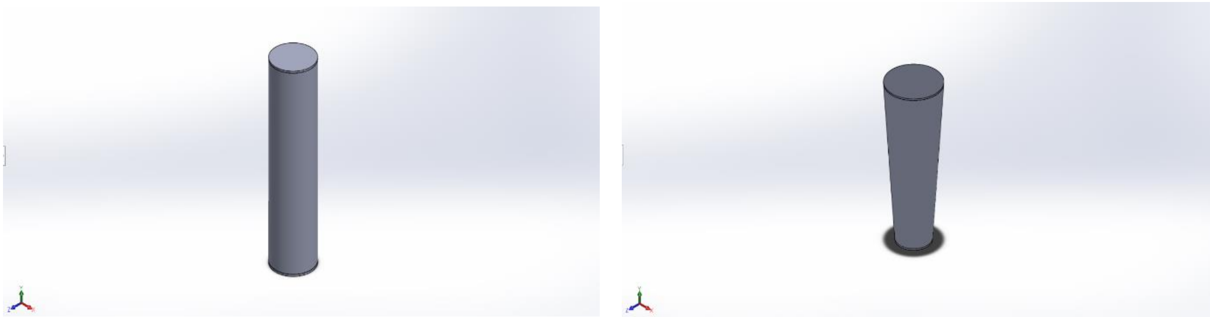

Figure 4: Main design for 3D model

Table 1 shows the detail of configuration for each proposed shape of vortex bladeless wind turbine design.

Table 1: Detail of configuration for each proposed shape of vortex bladeless wind turbine

Design	Diameter (m)	Height (m)
Design A1	1.0	5.0
Design A2	0.5	5.0
Design A3	3.0	5.0
Design A4	1.0	7.5
Design A5	1.0	2.5
Design B1	Upper: 1.5 Lower: 1.0	5.0
Design B2	Upper: 1.0 Lower: 1.5	5.0
Design B3	Upper: 1.0 Lower: 3.0	10.0
Design B4	Upper: 1.0 Lower: 1.5	7.5
Design B5	Upper: 1.0 Lower: 1.5	4.0

As for the CFD simulation there are 4 main steps as well as in 2D's which is solution domain defining, mesh generation, setup or solving, and lastly is result (chapter 3). For 3D model CFD simulation, Boolean operation and forming new part from several parts is highly considered. Figure 5 shows the solution domain of the 3D model analysis which is same as for 2D's except for the depth (height), the depth will be equal to two times of the length (height) of the bladeless wind turbine.

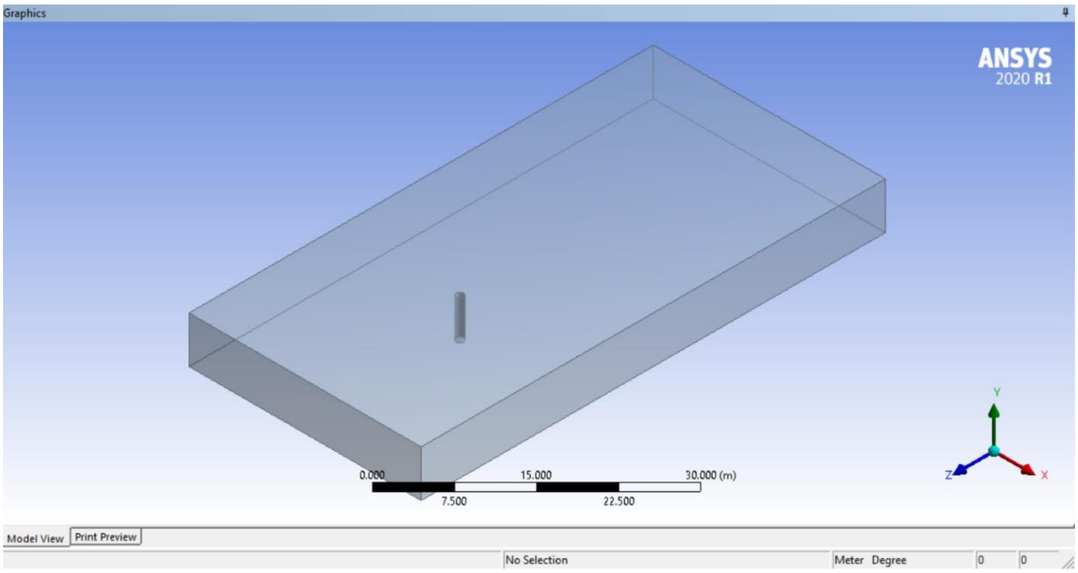

Figure 5: Solution domain for the 3D model

Next, mesh generation. The method used in meshing for 3D model is tetrahedrons as shown in Figure 6 involving edge sizing and refinement of several faces of the solution domain in order to achieve favourable condition for simulation.

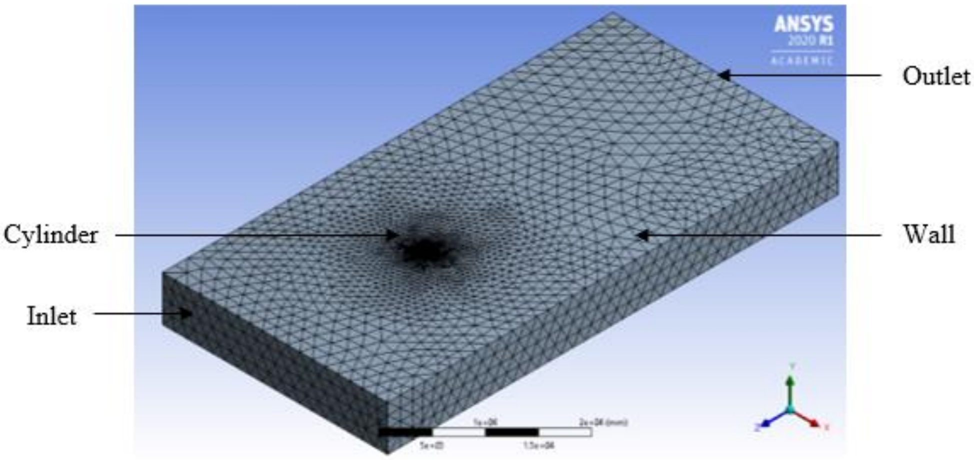

Figure 6: Meshing of 3D model with boundary name of the solution domain

As for setup, same as in 2D's which it covers the general condition of the fluid flow (model), definition of the material, condition of the boundary, solution method, report definition, initialization and run calculation. Largely part of the setup is quite similar to 2D's except certain parts. First is the value of velocity, U, density of air, and viscosity is defined equal to 2.5m/s, 1kg/m^3, and 0.01kg/m-s. Second, solution method of 3D model simulation is PISO. This is because PISO gives a stable solution for transient application of a fixed cylinder and third, for momentum spatial discretization section the Second Order Upwind is chosen rather than QUICK as it will typically be more accurate on structured meshes aligned with the flow direction as it produces an error due to the negative cell volume. Lastly, report definition. Rather than only plotting lift coefficient, Cl and drag coefficient, Cd graphs the vertex average for x-component velocity is calculated. This by pointing several points behind the cylinder and make the point capture the data (x-component velocity) of the air flowing through it. Based on the data captured from the points, Fast Fourier Transform (FFT) is applied to interpret the data.

3. Results and Discussion

Based on the scope of the study the 2D and 3D model of vortex bladeless wind turbine that have been build have undergoes CFD simulation and the result shall be presented. The determination of significance of aerodynamic characteristics on vortex bladeless wind turbine from Ansys-Fluent simulation and the determination of parameters that will affect the performance of vortex bladeless wind turbine will be further discussed.

3.1 2D Simulation

As for the 2D model analysis the value of lift coefficient, Cl and drag coefficient, Cd were obtained from the simulation. Figure 7 shows the graph of Cd against flow-time which the graph shows after few flow-time the value of Cd starts to converge at approximately 1.20 which agree with [3] where they stated that Cd was found to be at value equal to 1.21, 1.21, and 1.18 when the respective velocities equal to 0.9, 1.4, 1.75 m/s.

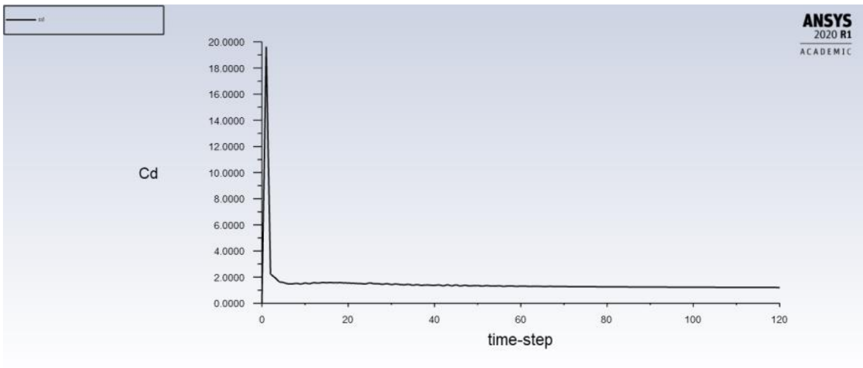

Figure 7: Cd against flow-time graph

Figure 8 show the velocity streamline where the value Re is equal to 100. Hence, the formation of laminar vortex street should occur. As shown in the Figure 8, the laminar vortex street do occur and this formation is known as Von Kármán vortex street where vortices depart from alternate sides of the cylinder.

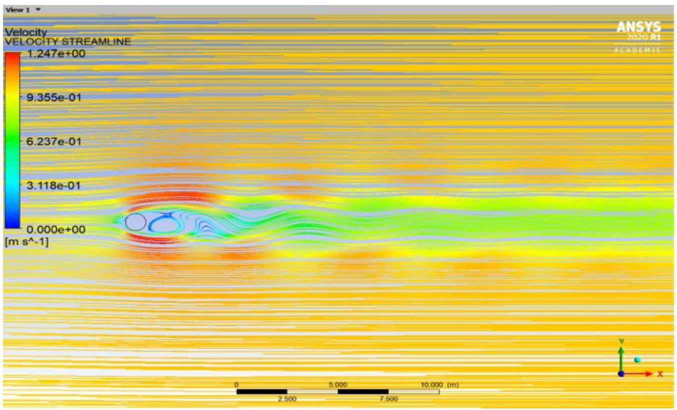

Figure 8: Velocity streamline around the cylinder

3.2 3D Simulation

The main proposed designs of the vortex bladeless wind turbine are Design A and Design B. After the variation of diameter and length for each design, Design A break into five configurations as well as Design B which it also break into five configurations. Taper ratio of a cylinder is defined as the ratio of length of the cylinder to the difference between maximum and minimum diameters of the cylinder [4].

In obtaining which design is the most favourable or most optimum for a vortex bladeless wind turbine, FFT is applied. Other than that, vortex shedding frequency (f_v), Strouhal number (S_t), and natural frequency (f_n) for each design body are calculated. Using FFT, a single output file with a time dependant data is interpreted into a summation of cosine and sine waves or in another word into a graph as for this study an output file of vertex average time history is used.

Firstly, Design A1. This design is a circular cylinder with a height of 5m and a diameter of 1m with value of Re equal to 250. After the simulation, Figure 9 shows the behaviour of vertex average velocity for x component at certain point behind the cylinder. Figure 10 shows the contour velocity of the simulation which the maximum value of velocity yield from the flow around the cylinder is 3.12 m/s where sides and edge of the cylinder face the highest value of velocity.

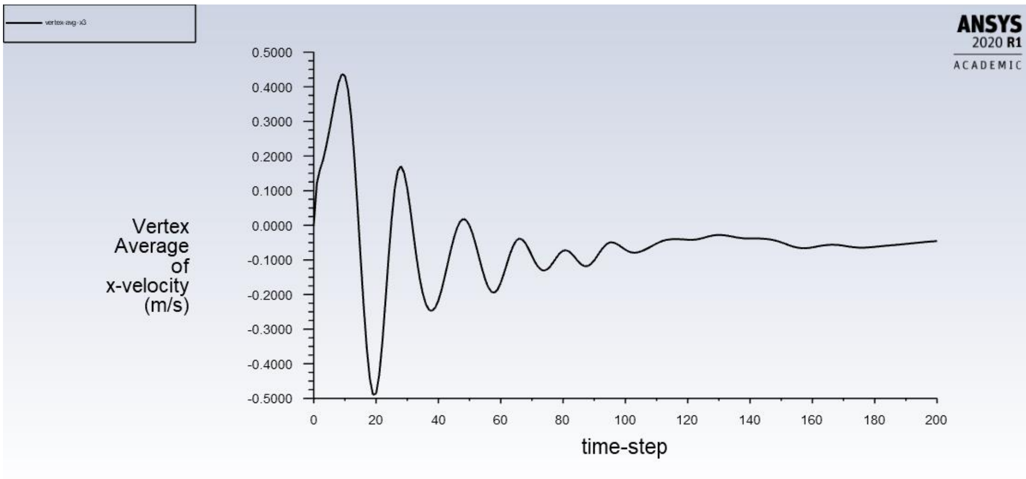

Figure 9: Vertex average of x-velocity for Design A1

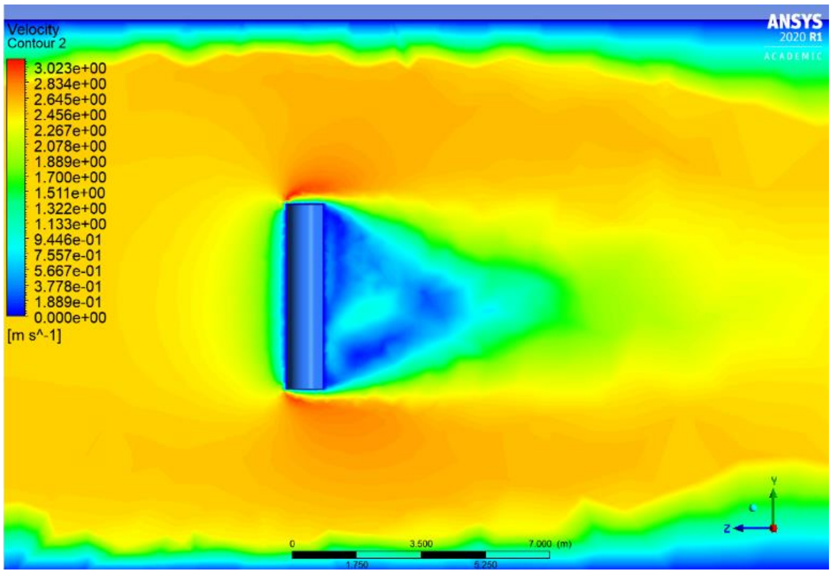

Figure 10: Side view of Design A1 velocity contour

As stated before, FFT is implemented to calculate the value of f_v and S_t. Figure 11 shows the value of vortex shedding frequency (f_v) is 0.249 Hz (at peak magnitude) while the value of Strouhal number (S_t) is 0.0996.

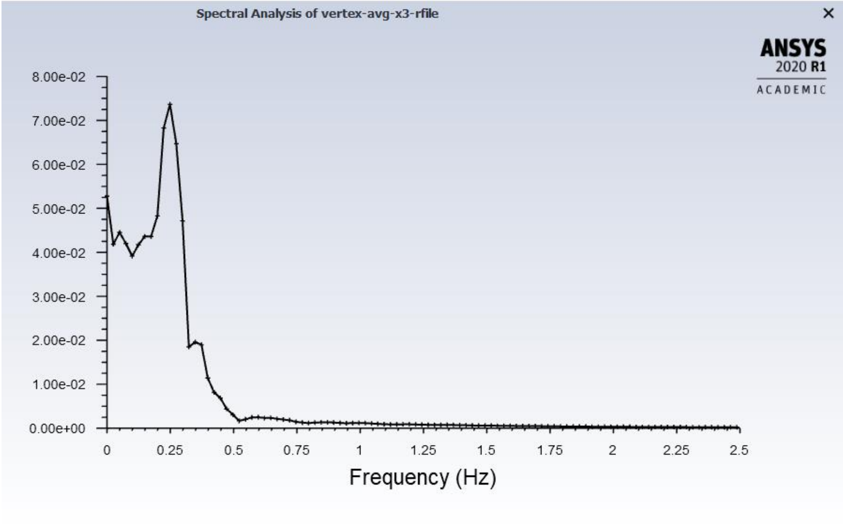

Figure 11: Spectrum of fv for Design A1

The interpretation of the data for Design A1 is repeated for each design proposed. Table 2 shows the summary of the result for each design. Purpose in obtaining value of f_n and f_v is to see at which configuration of bladeless wind turbine is near the lock-in phenomenon. This means that in this particular range the lift force which is the shedding oscillates in sympathy with the cylinder's motion resulting in the largest amplitude. Simply said, when the value of f_v is same with the value of f_n the situation is called lock-in. The optimum design of the vortex bladeless wind turbine will be chosen at which have the nearest value of f_v with its value of f_n. This show the optimum design of the vortex bladeless wind turbine will achieve lock-in phenomenon the quickest given the same environment (air speed) with the other proposed designs.

Table 2: Summary of the result for each design

Design	Re	f_v (Hz)	St	f_n (Hz)	f_v/ f_n (Hz)
A1	250	0.249	0.0996	0.8	0.31
A2	125	0.124	0.025	0.4	0.31
A3	750	0.0995	0.119	2.3	0.04
A4	250	0.249	0.0995	0.4	0.62
A5	250	0.35	0.139	3.4	0.10
B1 (TR: 10)	Max: 375 Min: 250	0.224	0.0896	0.7	0.32
B2 (TR: -10)	Max: 375 Min: 250	0.249	0.0995	1.54	0.16
B3 (TR: -5)	Max: 750 Min: 250	0.149	0.0597	0.98	0.15
B4 (TR: -15)	Max: 375 Min: 250	0.174	0.0697	0.69	0.25
B5 (TR: -8)	Max: 375 Min: 250	0.249	0.0995	2.35	0.11

3.3 Validation

As for 2D simulation, the value obtained for coefficient of drag, Cd is equal 1.20 which agree with [3] as it stated the Cd value of a cylinder when the velocity is 0.9m/s is equal to 1.21. The velocity of the fluid for 2D simulation is equal to 1m/s.

As for 3D simulation, Design A2 (circular cylinder) shows good agreement with [2] where for vortex induced vibration simulation at Re equal to 100 they obtained f_v equal to 0.1666 Hz at f_n equal to 0.2 Hz. Simulation of Design A2 that operate at Re equal to 125 resulting to f_v equal to 0.124 Hz for f_n equal to 0.4 Hz. The percentage difference of f_n is 1% while for f_v is 2.6% which show a good agreement considering the slightly different value Re. Design B1 and Design B2 show good agreement with [4] which they stated that for the inverted configuration of the tapered cylinder (negative value of TR) at TR=10 , they observed the maximum amplitude of oscillations for the inverted cylinder increases as shown in Table 2 where the value of f_v for Design B2 is slightly higher than Design B1 (f_n can be a factor). Based on the result of present study and the referred study, the results obtained can be regarded as being in a good agreement with previous study.

4. Conclusion

The significance of the aerodynamic characteristic involved was obtained and the optimum shape of the vortex bladeless wind turbine among the proposed design also achieved. The main conclusions of the study can be summarized as following;

1. The drag coefficient for 2D simulation was successfully obtained with value of Cd equal 1.20 at velocity if air equal to 1m/s.
2. This study proposed two main design which each of it have a varied value of diameter and height resulting to total 10 different configurations of vortex bladeless wind turbine.
3. Spanwise length or the height of the circular cylinder (Design A) do affect the performance of the bladeless wind turbine as study shows the cylinder have to have a height equal or above its two times diameter length. This show why Design A3 and Design A5 have quite unusual trend and result of St.
4. Taper ratio (TR) of the linearly tapered cylinder (Design B) do affect the performance of the bladeless wind turbine as study show a wider lock-in ranges can be obtain compared to uniform circular cylinder as the lock-in range is shifted when the velocity of fluid is higher as the TR decreased. Other than that, minor difference in amplitude was obtained from previous study where the negatively tapered cylinder have increase in amplitude compare to the positively tapered cylinder at same TR below and equal to 10. This show why Design B1 and Design B2 have slightly difference value with each other despite having same dimension.
5. The most optimum vortex bladeless wind turbine design among the proposed design is **Design A4**. Having the highest value f_v/ f_n which equal 0.62 making the design could achieve lock-in phenomenon quicker than other proposed design at given velocity of fluid (air). This is correspond to the current development of vortex bladeless wind turbine from Vortex Bladeless Ltd company where the ratio between the height to the diameter of the bladeless wind turbine is high.

Acknowledgement

The author would like to thank the author's supervisor, Dr Sofian bin Mohd, for the guidance and encouragement throughout this research. The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for giving the opportunity to conduct this study.

References

[1] Y. Bai and Q. Bai, Subsea Pipelines and Risers. Burlington: Elsevier Science, 2005.

[2] M. Asyikin, CFD Simulation of Vortex Induced Vibration of a Cylindrical Structure, 2012. Available: https://core.ac.uk/download/pdf/30817511.pdf. [Accessed: 12- Jul- 2020].

[3] G. Bruschi, T. Nishioka, K. Tsang and R. Wang, DRAG COEFFICIENT OF A CYLINDER, 2003. Available: https://sv.20file.org/up1/916_0.pdf. [Accessed 29 December 2020].

[4] B. Seyed-Aghazadeh, D. Carlson and Y. Modarres-Sadeghi, "The influence of taper ratio on vortex-induced vibration of tapered cylinders in the crossflow direction", Journal of Fluids and Structures, vol. 53, pp. 84-95, 2015. Available: 10.1016/j.jfluidstructs.2014.07.014.