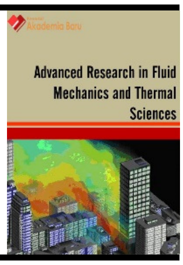
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Numerical investigation of vortex formation effect on horizontal axis wind turbine performance in low wind speed condition

Open Access

Nurul Izyan Majunit[1], Fazila M. Zawawi[1,*], Nur Safwati Mohd Nor[2], Haslinda Mohamed Kamar[1], Nazri Kamsah[1]

[1] Department of Thermofluids, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[2] Department of Applied Mechanics, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia

ARTICLE INFO

Article history:
Received 25 September 2016
Received in revised form 28 October 2016
Accepted 30 October 2016
Available online 19 November 2016

Keywords:
Vortex formation, Small Wind Turbine, Power Losses, Rotor Life, Velocity Deficit, Turbulent Intensity

ABSTRACT

Wind energy is one of the renewable energy with no pollution and low-cost. Many countries especially those with high wind speed have been widely applying wind turbine in generating electricity and managed to reduce much cost of power generation. As the wind turbine relies on the wind, the use of large wind turbine in countries with low wind speed is impractical for the fact that wind is insufficient and not afford to rotate the wind turbine. Thus, small size wind turbines are more appropriate under low speed wind condition. However, small wind turbine suffers from high vortex downstream to the wind turbine that leads to high power losses. Reliable and practical wind turbine in countries whose wind speed is low is the idea behind the work done in this research. The effect of vortex formation on the performance of the downstream small wind turbine is investigated using numerical approach. A suitable modelling and simulation method are identified for the purpose of wake simulations behind a single wind turbine and the important data with respect to the effect of different spacing between wind turbines are also studied.

1. Introduction

As the time passes by, the demand on the electricity keeps increasing while the source from the earth become decreasing. This factor causing human to find another alternative source that were renewable such as solar, biomass, wind, geothermal, waves and tides. Most of this renewable energy had been applied in many countries such as Japan, Australia and China. In Malaysia, the main sources of electricity were oil, gas, coal and hydro. In order to achieve one of the Vision 2020, the Third Outline Perspective Plan (OPP3), to build a resilient and competitive nation, and, to provide adequate and sustainable energy, and reduce the emissions of greenhouse gases, in 2001, the government of

* *Corresponding author.*
E-mail address: fazila@mail.fkm.utm.my (Fazila Mohd Zawawi)

Malaysia widened the country's four fuel energy policy by adding renewable energy as the fifth fuel [1]. Thus, renewable energy had expected to contribute 5% of the electricity in Malaysia by the year 2005 [1]. Thus, wind energy was one of the possible energy that could be extracted in this country besides solar and biomass energy. Compared to other sources of energy, wind is not producing CO_2 emission while generating electricity, thus it does not contribute to the greenhouse effect. Wind energy had been popular worldwide since it only needed a simple and inexpensive automation which was wind turbine. The countries that had applied this wind power were Germany, China, Denmark, Spain and many more. Malaysia had not applied this technology yet as this country has a low wind velocity, which was about 3 m/s.

The wind turbine is an automation used to extract kinetic energy from wind and convert it into electrical energy while the wind turbine farm is a group of wind turbines that placed together. The installation of wind turbines in large wind turbine farms has been increasing day by day in many developed countries in order to gain more electric power. However, there were two main problems due to placing a grouping of wind turbines together; the power production of a wind turbine that operating in the wake of another turbine will reduce due to lower incident of wind speed, and the lifetime of the rotor will be shortened because of the increase of turbulence intensity as concluded by [2]. The other study also found that the power output for three turbines separated by seven rotor diameters face a reduction of 10% [2]. It is also explained that the power losses of downstream turbines for full wake condition can be 30-40%, but when averaged over different wind directions, the losses become 5-8%. Thus, this problem seems to be very serious as it affects the power generated and the lifetime of the blade [3].

The major factor of these issues was the formation of vortex at the downstream of the turbine blades. As the air passing through the blades, lift force generated due to the three dimension force acted, thus causing the wind turbine to rotate. The lift force generated by the blades can be credited to a distributed bound vortex which responsible for the jump in the tangential velocity over the blade and as such the pressure difference and the lift. At the tip of the blade, the difference in pressure between the lower and upper side leads to the formation of a tip vortex. The different tip vortices formed the root vortex at the hub. The root vortex caused the tangential velocity at the blades, which the rotation was opposite to that of the rotor. As the tip vortices had a tangential component, they induced an axial velocity at the blades, which cause the deceleration of the flow in front of the turbine. The velocities of air inside and outside the wake were different which results in shear layer, which thickens when moving downstream. Turbulent eddies are formed in the shear layer. The turbulence in the shear layer is non-uniform due to the ambient shear flow. It mixed the low velocity fluid in the wake with high velocity fluid outside it. This is caused momentum was transferred into the wake, next the wake will expands, but the velocity deficit is reduced. Figure 1 illustrates the velocity profile in the wake of the wind turbine.

To make sure the efficiency and lifetime of the turbine that located at the back of the other turbine does not affect, the minimum and the most optimum distance between those two turbines should be investigated. The minimum distance was the point which the wake velocity deficit is minimal. Ammara et al. (2002) stated that the experimental and theoretical studies have suggested that a wake velocity deficit is minimal after 10D [4] while Hojstrup (1999) mentioned that the mechanical turbulence in the wake still visible after 15D [5]. Furthermore, the calculation of rotor performance and blade loading also need to be performed to get the optimal result of reducing power losses and to improve the lifetime of the blades. Here, getting the value nearest to Betz limit, $C_{Pmax} \approx 0.59259$ would the best outcome. Many studies had appeared that address this problem using numerical study, CDF as this method was cheaper compared to experimental and can provide detailed information both upstream and downstream of the turbine.

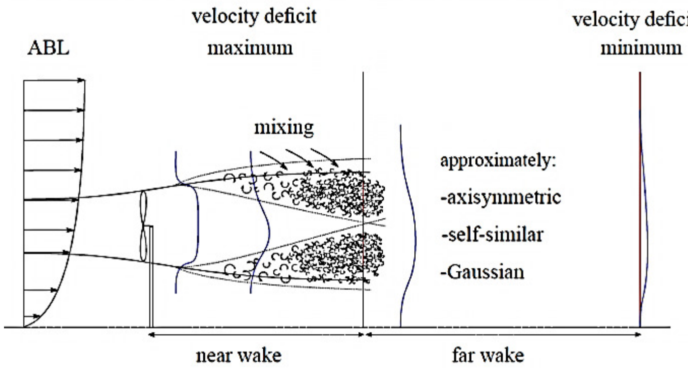


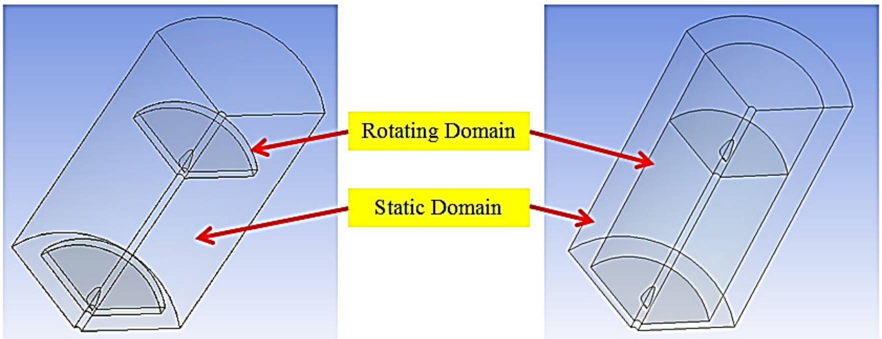
Fig. 1. Velocity profile in the wake of the wind turbine

The wind turbine installation in Malaysia was not become popular yet as the wind speed in this country was very low. Thus, there were not many numerical studies about these issues for this small velocity range. This study will focus on the small scaled wind turbine in lower wind speed area. Those were the aim of this study in order to address the issues: i) to study the effect of vortex formation on performance and aerodynamic loadings of small wind turbine, ii) to establish the numerical model for effectively analysing the effect of vortex on downstream small wind turbine performance. This study was expected to provide methods as a reference to the other researchers that doing numerical study, which analysing the effect of vortex formation at the downstream of the small scaled wind turbine in a low wind speed condition in Malaysia.

2. Methodology

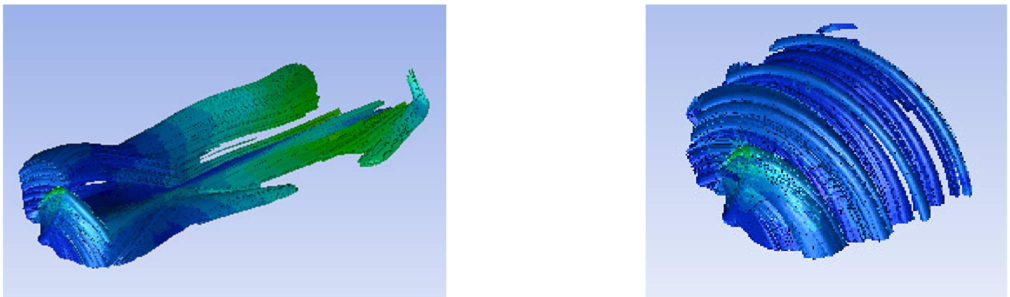
Numerical study were conducted by using the CFD code which is ANSYS – FLUENT software. There were two cases in order to achieve the set objectives. Case 1 was about studying the flow behaviour at the downstream of a single wind turbine and the Case 2 was about studying the interaction of wake between of upstream wind turbine to the downstream wind turbine. A wind turbine blade with a model of cambered thin plate (8312) was used. The diameter of the design wind turbine was 0.4 m and radius of the hub of 3 cm. The simulation was set up using a single blade of the wind turbine in a single reference frame with 120˚ periodic boundary conditions. This periodic boundary condition had been used in order to reduce the computational time [7],[8]. Two types of domains were employed; rotating domain (rotating control volume) and the stationary domain (stationary control volume). Based on the literature review, there were two ways to compute the downstream flow of the wind turbineas per shown in Figure 2. Figure 2 shows the differences between both computational domain styles.

Both types of computational domain were simulated using ANSYS FLUENT and the flow downstream result can be seen in the Figure 3. It could be seen that the downstream flow of model are straight following the streamline while the downstream flow of model B is rotating. Hansen in 2008 stated that since a horizontal-axis wind turbine consists of rotating blades, a vortex system similar to the linear translating wing must exist [3]. The vortex sheet of the free vortices is oriented in a helical path behind the rotor. The strong tip vortices are located at the edge of the rotor wake and the root vortices lie mainly in a linear path along the axis of the rotor, as shown in Figure 1. It can be concluded that the Model B is able to illustrate the downstream flow of wind turbine more accurately to the theoretical compared to computational domain using Model A. Thus, this study will use the computational domain of Model B.



(a) Model A (b) Model B

Fig. 2. Two types of computational domains for study the downstream flow of wind turbine. (a) Short rotating domain [7] (b) Extended rotating domain [6].



(a) Model A (b) Model B

Fig. 3. Flow downstream of the wind turbine (Velocity magnitude flow computed using ANSYS FLUENT) – (a) Short rotating domain [7] (b) Extended rotating domain [6]

The structured meshed was applied to a static domain while unstructured meshed was applied to the other part which are rotating domain and the blade surface. As this study was focused on the downstream flow of wind turbine, thus the GIT had been measured using the streamline velocity at 0.6 meters behind the wind turbine. The GIT had been done by using a wind velocity of 6 m/s with an RPM of rotating domain of 1000. Based on the result shown in Figure 4, the velocity graph along the axial line when using mesh 3 and mesh 4 were almost the same, thus mesh 3 will be selected from GIT result. Mesh 3 has a fine mesh size and maximum face size of 0.04m.

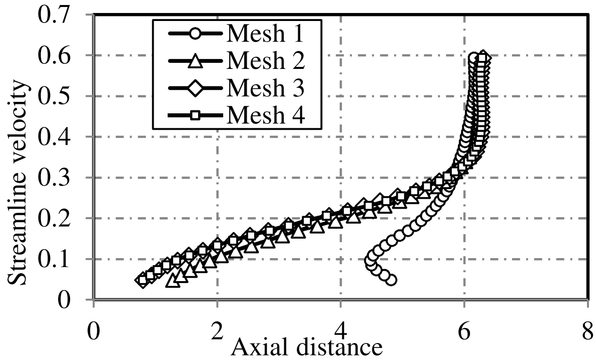


Fig. 4. The result of Grid Independent Test (GIT)

The resolution of the near-wall grid can be determined by the description of y+ parameter. The size of mesh around the blade had been reduced by specifying the face sizing around the blade with element size of 1e-3 meter, growth rate of 1.2 and local minimum size of 1e-5 meters. Inflation had been set for the first aspect ratio around the blade with maximum layers of five and growth rate of 1.2. Based on the y+ plotted in Fig. 6, it indicates that the number of cells within the 'viscosity affected region' are sufficient to perform a good estimation of the mean velocity and turbulent quantities near the wall as the y+ value is less than 10.

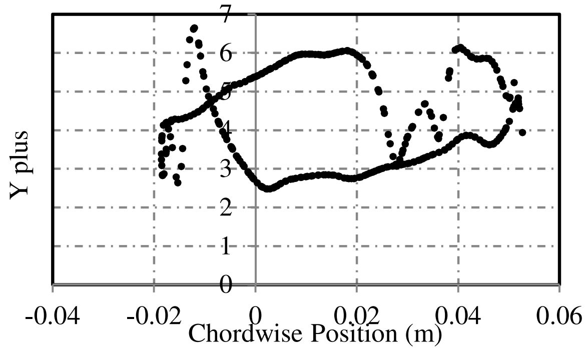


Fig. 5. Wall function (Dimensionless wall distance in chord wise position at mid-span)

The turbulence model used in the simulation of the both propeller models was k-ω shear-stress transport (SST). This turbulence model was chosen due to the fact that the k-ω SST model was conceived to combine the best characteristics of the ε and ω equations, making it a perfect model to calculate the boundary layer transition and separation effects experienced by the blades and to calculate the highly turbulent turbine slipstream flow [7]. The boundary conditions used by this model are presented in Table 1. Relative to, the solution methods in the pressure-velocity coupling SIMPLE scheme was used. Spatial discretization methods used are presented in following Table 2.

Table 1
Boundary condition for every zone

Zone	Boundary Condition
Blade and Rotating hub	Moving wall (Non-Slip)
Inlet	Velocity Inlet
Outlet	Pressure Outlet
Rotating Zone	Interface
Rotating Volume	Fluid
Static Volume	Fluid

Table 2
Methods applied in spatial discretization solutions

Spatial discretization	
Gradient	Least squares cell based
Pressure	Standard
Momentum	Second order upwind
Turbulent kinetic energy	Second order upwind
Specific dissipation rate	Second order upwind

Case 1: This case was carried out to investigate the performance and aerodynamic loading behind the wind turbine. There are some conditions that had been fixed, including the velocity of wind, 4 m/s, number of rotations of blade per meter, 600 RPM and the model of blade with diameter 0.4 m.

The first dimension of the domain had been made based on literature review where Maia (2014) stated that the minimum axial velocity deficit was between 1.5 to 3D of wind turbines and minimum wake velocity deficit is at 10D [8].

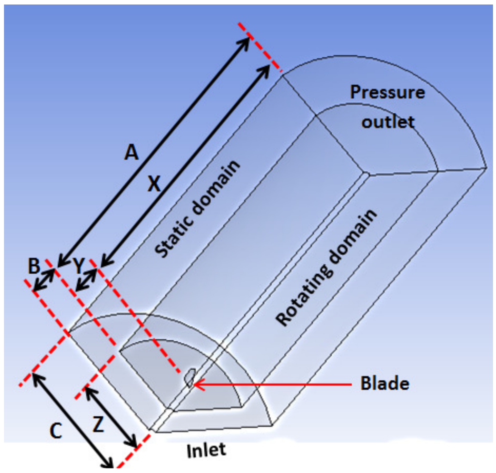


Fig. 6. The computational domain of Case 1 domain geometry

Table 3
Dimension of domain based on literature review

Part	X	Y	Z	A	B	C
Size	11D	1D	1.5D	10D	1D	2.5D

Three conditions had been tested by varying the wind velocity and RPM that leads to different values of Tip Speed Ratio as shown in table 4. Tip Speed Ratio (TSR) is the local velocity ratio for the tip of the blade:

$$TSR = \frac{\Omega R}{u_\infty} \quad (1)$$

where

$$\Omega = \frac{2\pi(RPM)}{60} \quad (2)$$

where R is radius of the rotor, Ω is the angular velocity and u_∞ is Vwind or wind velocity.

Table 4
Conditions tested

Condition	1	2	3
V_{wind}	2m/s	4m/s	7m/s
RPM	250	600	1500
TSR	2.618	3.142	4.488

After the computation, it could be seen that in the area of static domain, the graph had been linear and do not have any changes along the static domain line. Thus, the radius of the static domain could be reduced. It could be seen that the front distance of static domain does not give any significant of changing, so this distance also could be reduced. The reduction of dimension will reduce the computational time. It also could be seen that the velocity deficit was still high at the distance of 10D after the wind turbine, so the length behind the wind turbine should be increased. Notice that there was no significant difference of power get when small reduction of static domain size was applied. Fig. 6 and Table 3 shows the new dimension of the domain that will be used in this case.

Table 4
Dimension of domain for case 1

Part	X	Y	Z	A	B	C
Size	20D	0.5D	1.25D	20.5D	0.25D	1.75D

Case 2: This case will study on how the distance between two wind turbines can affect the performance of the downstream wind turbine. Three computational domains with the same size as Case 1 but added with another rotating domain at the back with different distance from two blades had been constructed and shown in Figure 7. As the downstream flow of the second wind turbine are not under the consideration of the study, only short in length of the second rotating domain was constructed. Performance of a turbine can be calculated using the Blade Element Theory (BET) through power coefficient and also by observing the mechanical power produced by the wind turbine.

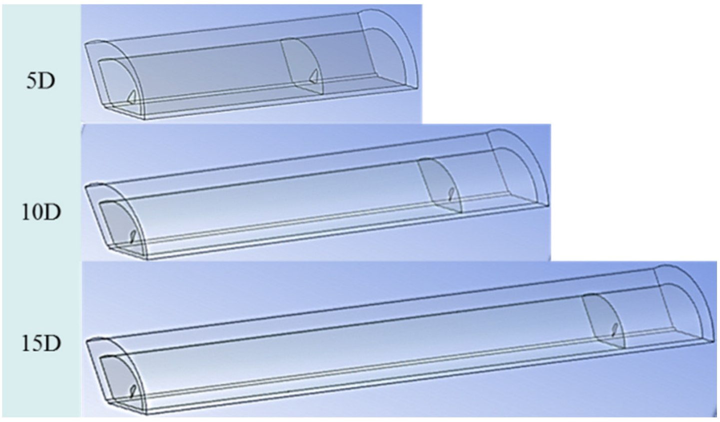


Fig. 7. distance between two wind turbines

4. Result and discussion

Case 1: the velocity deficit cause of the vortex formation would be analysed by extracting the data from the horizontal line from different distance downstream as shown in figure 8. As the simulation used the periodic boundary condition, only one side data of the wind turbine will be taken as the other side was assumed to be symmetrical.

The time taken for solving this case was approximately 39 hours with a number of cells of 1.42 million. The simulations were done by using parallel 4 processors. The residual was set to be $1x10^{-5}$ and had been converged to get the accurate result. As expected, the velocity deficit behind the wind turbine and is recovered at a certain distance. Figure 9 shows the velocity deficit variation with distances behind the wind turbine. Velocity deficit are seen to be highest at the distance of 1D from the wind turbine. This phenomenon might happen due to maximum turbulent intensity present due to vortex formation. This shows that velocity deficit reduces with distance. The trend of the graph can be compared to the experimental result reported in Vermeer *et al*. [9]. Besides, the velocity could still not recovered at the distance of 20D due to turbulent intensity [8]. However, the decay of the velocity deficit is more rapid than the decay of turbulence intensity. This result proved that the finding from the numerical simulations done are in good agreement with the experiments. Vermeer et al found that turbulence effects are more persistent, and that the decay of the velocity deficit is more rapid than the decay of turbulence intensity; this is also observed in large field experiments [9]. This result is also confirmed by field experiments: Højstrup and Högström et al. found that turbulence effects are noticeable even at 12D and 10D downstream, respectively, whereas velocity defects are almost negligible at those distances [9]. This situation also could be seen in this study. For the better view of this, figure 10 and figure 11 shows a good illustration on the rate of decayed for velocity deficit and turbulent intensity. This condition affects to the estimation of velocity deficits and farm efficiency in terms of energy production. Thus, Case 2 gives a better view on how serious was this problem.

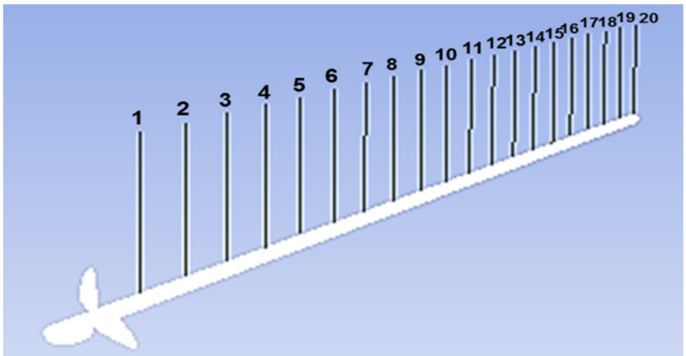


Fig. 8. Layout of line to analyse flow at downstream of wind turbine

Case 2: Three model of domain with different distance between blades was computed. The number of cells for model with distance 5D, 10D and 15D were 2.11, 2.23 and 2.32 million, respectively, while the time taken for cases to be solved were 41, 26 and 29 hours respectively. The computational time for model with distance 5D was very long as the turbulent intensity and turbulent kinetic energy were very high and result in fluctuating residuals of flow model k-w. The simulations were done by using parallel 12 processors. The residual was set to be $1x10^{-5}$ and were converged in order to get the accurate and precise result.

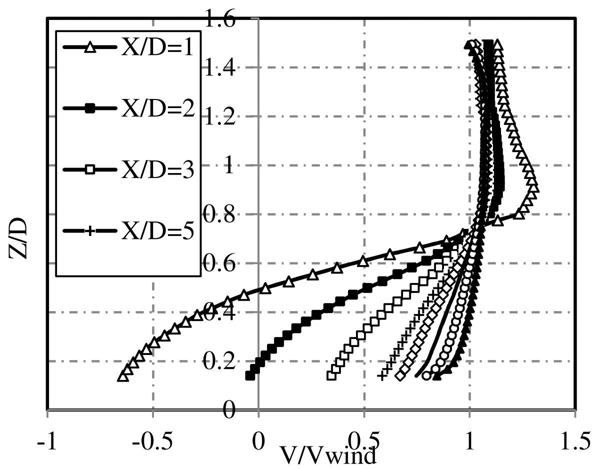


Fig. 9. Graph of dimensionless velocity profiles with comparing some distances at the downstream of wind turbine

Fig. 10. Streamline velocity contour on vertical plane bisecting turbine hub

Fig. 11. Turbulent intensities contour on vertical plane bisecting turbine hub

Performance coefficient of the upstream wind turbine is 0.443. This value shows that the performance of this wind turbine was quite good as the C_P value near to Betz limit, $C_{Pmax} \approx 0.59259$. The mechanical power produce by the first wind turbine or upstream wind turbine were the same for all models which was 2.128Watt. However, the mechanical power produce by the second wind turbine or downstream wind turbine were different depend on the distance between two blades. The mechanical power generated by the downstream wind turbine for model with distance 5D, 10D and 15D were 1.232, 1.270 and 1.316Watt respectively. It could be said that the power of downstream wind turbine increase with the increasing of the distance between two blades. The power losses for the downstream wind turbine were 43.538, 41.797 and 39.688percent for model with distance 5D, 10D and 15D each. It could be concluded that the power losses, decreasing with the increasing of the distance between two blades. This might be due to velocity deficit and turbulent intensity reduced with the increment of distance from the single wind turbine. Also, it could be observed that there are slightly different in velocity deficits at the same distance after the first wind turbine for single turbine and two turbines. It could be said that vortex formation not only affects the downstream flow, but also affects the upstream flow of the turbine itself. This situation has been said earlier where right in front of the second turbine, there was a slight increase in velocity deficit [7]. He also said that the disturbance caused by the second turbine actually acts to slow the rotation, even though the turbines are rotating at the same angular velocity and direction.

The second wind turbine will have a shorter lifetime compared to the first turbine. This problem happens due to the higher turbulent intensity produced by the vortex formation of the first turbine. Figure 12 shows the turbulent intensity for the three models just before the second wind turbine. The turbulent intensity is highest at 5D and it decreases as the distance increases. The most important structural effect on a wind turbine in the wake of a neighbouring machine is fatigue. That is due to the combined effect of increased turbulence, wind speed deficit and shear, and changes in turbulence structure that cause dynamic loading, which may excite the wind turbine structure. Thus, in order to calculate the dynamic loading acted on the downstream wind turbine, static pressure of the wind turbine had been digging out. The maximum pressure acted on the wind turbine for models with distance 5D, 10D and 15D were 1779.62, 284.78 and 49.79 Pascal respectively. It could be said that second wind turbine that located nearest to the first wind turbine experiencing the high pressure or loading due to highest velocity deficit and turbulent intensity occur. The higher the pressure experienced by the wind turbine, the shorter the life of the blade.This can be concluded that the vortex formation by the upstream wind turbine will reduce the power produced by the downstream wind turbine and also cause in increasing the aerodynamic loading of downstream wind turbine. Velocity deficits at the wake of upper turbine results in power losses by the downstream wind turbine.

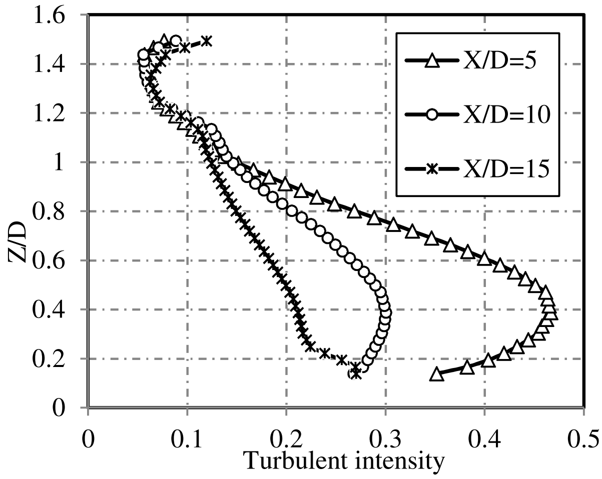


Fig. 12. Graph of turbulent intensity with comparing two turbines with spacing of 5D, 10D and 15D downstream before second turbine

4. Conclusion

The effect of vortex formation on the performance and aerodynamics loading of small wind turbine and the numerical model for effectively analysing the effect of vortex formation on downstream small wind turbine performance has been successfully investigated and well established respectively. According to the findings obtained from the modelling work, it can be concluded that the velocity at the wake deficits due to the presence of turbulence created by the vortex formation which later recover the velocity after some distance from the downstream of the upstream wind turbine. The velocity deficit magnificent was observed downstream of the single wind turbine. The structure of the wake was investigated using velocity streamline, and turbulent intensity. The maximum velocity deficit was occurred at 1D from the single turbine.

Acknowledgement
This research is supported by GUP grant under the VOT number of Q.J130000.2724.11J50. The authors fully acknowledged Universiti Teknologi Malaysia for the financial support on this research.

References

[1] Kardooni, R., Yusoff, S.B., Kari, F.B. "Renewable energy technology acceptance in peninsular Malaysia." *Energy Policy* 88, (2016): 1-10.

[2] Sanderse, B. Aerodynamics of Wind Turbine Wakes. Energy Research Centre at The Netherlands. Technical Report ECN-E–09-016, 2009.

[3] Hansen, M.O.L. Aerodynamics of Wind Turbines, 2nd Ed., Earthscan, 2008.

[4] Ammara, I., Leclerc, C., Masson, C. "A viscous three-dimensional differential/actuator-disk method for the aerodynamic analysis of wind farms." *Journal of Solar Energy Engineering* 124, no. 4 (2002): 345-356.

[5] Højstrup, J. "Spectral coherence in wind turbine wakes." *Journal of Wind Engineering and Industrial Aerodynamics* 80, no. 1 (1999): 137-146.

[6] Zawawi, F.M. Aeroelastic Analysis of Flexible Composite Proprotor Blades for Convertible Micro Air Vehicles. PhD Thesis, l'Institut Sup´erieur de l'A´eronautique et de l'Espace (ISAE), 2014.

[7] Davis, C.J. Computational Modelling of Wind Turbine Wake Interactions. Master Thesis, Colorado State University at Fort Collins, Colorado, 2012.

[8] Maia, L.A.B. Experimental and Numerical Study of a Diffuser Augmented Wind Turbine-DAWT. PhD diss., 2014.

[9] Vermeer, L.J., Sørensen, J.N., Crespo, A. "Wind turbine wake aerodynamics." *Progress in aerospace sciences* 39, no. 6 (2003): 467-510.