

DESIGN AND FLOW VELOCITY SIMULATION OF DIFFUSER AUGMENTED WIND TURBINE USING CFD

T. SARAVANA KANNAN*, SAAD A. MUTASHER, Y.H. KENNY LAU

Faculty of Engineering, Computing and Science, Swinburne University of Technology,
Jalan Simpang Tiga, 93350, Kuching, Sarawak, Malaysia
*Corresponding Author: sthangavelu@swinburne.edu.my

Abstract

The main objective of this research is to optimize the diffuser design of Diffuser Augmented Wind Turbine (DAWT). Specifically, this study investigates the effect of wind velocity on different shapes of flanged diffusers to develop the suitable diffuser for the wind turbine. For that purpose, the ideologies of DAWT have been studied. Different diffuser design concepts were developed and the wind speed for each design is simulated using design software Solidworks and Computational Fluid Dynamic (CFD) software ANSYS Fluent. Results show that a remarkable 6.45 m/s final wind velocity with 61.25% increase over the 4 m/s inlet velocity in the re-developed design which was created using 16° diffuser opening angle coupled with a 0.5 m diffuser splitter and a 4° splitter opening angle.

Keywords: Wind turbine, DAWT, Diffuser, Wind velocity, CFD.

1. Introduction

The development and application of renewable, clean energy have become an important issue in recent years due to the serious effects of global warming and rapid depletion of fossil fuels. In order to address the current energy crisis various means of alternative energy are being evaluated [1]. Wind energy technologies have become one of the fastest growing energy sources in the world and it symbolizes a feasible alternative, as it is a virtually endless resource. However, in comparison with the overall demand for energy, the scale of wind power usage is still very meagre. As for the reasons various causes are possible including cost [2]. Therefore, the introduction of a new wind power system that can produce higher power output even in areas where lower wind speeds are expected would be cost effective. Also in conventional wind turbine the air flow is slowed down

and widened. This effect causes a loss in the efficiency of the turbine. By creating a field of low pressure behind the turbine, that effect reduced and the corresponding loss in efficiency can be avoided [3].

One of the most promising concepts in the wind energy field is the development of wind power augmentation systems. By the use of diffuser in the wind turbine, wind power augmentation system is also called as Diffuser Augmented Wind Turbine (DAWT). DAWTs were a hot topic at the 1979 wind energy innovative system conference. In DAWT the diffuser or flanged diffuser generates separation regions behind it, where low pressure regions appear to draw more wind through the rotors compared to a ‘bare wind turbine’ [4, 5].

Wind power generation in wind turbine is directly proportional to the wind speed served. Therefore, a large increase in output is brought about if it is possible to create even a slight increase in the velocity of the impinging wind to a wind turbine. If we can increase the wind speed by exploiting the fluid dynamic nature around a structure or topography, the power output of a wind turbine can be increased substantially. In conventional wind turbine the wind speed served from the location is directly used to generate power. But in DAWT, the available wind speed is increased by the diffuser to generate high power output. Also the smaller diameter blade in DAWT produces the same power output as of a conventional ‘bare wind turbine’ due to the concentration of the wind energy density [5]. Currently very few studies have been reported the effect of diffuser shape in wind speed using computational simulations.

Abe and Ohya [6] investigated the flow fields around flanged diffusers to develop small type wind turbines using CFD. Results showed that the performance of a flanged diffuser strongly depended on the loading coefficient as well as the opening angle because it greatly affected the nature of the separation appearing inside the diffuser. It was suggested that the loading coefficient for the best performance of a flanged diffuser was considerably smaller than that for a bare wind turbine. Chen et al. [7] studied the effects of flanged diffuser on rotor performance to develop small wind turbine for moving vehicles. Results showed that the flanged diffuser significantly increased the power and torque outputs, and rotational speed of the turbine.

Abe et al. [8] was also carried out the experimental and numerical investigation of flow fields behind a small wind turbine with a flanged diffuser. Results showed that the power coefficient of the shrouded wind turbine was about four times that of a bare wind turbine. Ohya et al. [9] examined the optimal form of the flanged diffuser, and demonstrated that power augmentation by a factor of about four to five, compared to a bare wind turbine.

Wang et al. [10] investigated a convergent divergent scoop effect on the power output of a small wind turbine. Results showed that the scoop boosts the airflow speed and increased the power output by 2.2 times with the same swept area. Although several other ideas had been reported so far, most of them did not appear to be reaching commercialization. Gardan and Bibeah [11] studied the effect of diffuser on the performance of hydrokinetic turbine using validated momentum source turbine model. Results showed that the diffuser configuration produces 3.1 times more power than the turbine with no diffuser.

Therefore, the development of a wind power system with high output and low cost aims at determining how to collect the wind energy and wind velocity efficiently and what kind of diffuser design can generate energy effectively from the wind speed. The aim of this work is to develop and design a new diffuser splitter that can collect and accelerate the wind approaching better. Namely, we have devised a diffuser shroud with a large flange that is able to increase the approaching wind speed substantially by utilizing various flow characteristics [6-10].

Through this study four different concepts for diffuser design have been developed using Solidwork; and CFD simulations were performed to find wind velocity for different diffuser using ANSYS Fluent. The flanged diffuser shroud plays a role of a device for collecting and accelerating the approaching wind as in Fig. 1.



Fig. 1. Prototype Wind Turbine with a Flanged Diffuser Shroud (500 W). The Rotor Diameter, D , is 0.72 m. The Area Ratio μ (Outlet area/Inlet Area) of the Diffuser is 2.35. [9]

2. Diffuser Augmented Wind Turbine

2.1. Theory

A DAWT would have a duct surrounding the wind turbine blades that increase in cross sectional area in the stream wise direction. The pressure behind the turbine will drop due to the wind turbine being enclosed by the diffuser, thus the wind velocity approaching the wind turbines will be increased. The Vortec 7 was the first full scale DAWT constructed. The aerodynamics of a diffuser are such that more air flows through the blade plane, and more power can be generated compared to a conventional 'bare turbine' of the same rotor blade diameter [12].

A smaller blade diameter in DAWT produces the same power output as of a conventional bare wind turbine due to the concentration of the wind energy density. It can work on a lower cut in wind velocity as well. It was noticed that DAWTs can reduce the sensitivity of the yaw (the misalignment between wind and turbine pointing direction) and can significantly reduce noise. DAWTs should capture more energy than normal conventional wind turbine when the winds are fluctuating and it can align itself with the flow [13]. With these benefits, a diffuser augmented wind turbine can lower the cost to produce energy.

2.2. Ideology

The concept of diffuser is to increase the power output of a wind turbine by accelerating the wind velocity that approaches the wind turbine. To increase the wind velocity, a lower pressure would appear at the back of the wind turbine to act as a vacuum to suck the wind and accelerate it towards the blades as shown in Fig. 2. The region of areas surround or near the vortex would have typically lower pressure where the suction force will be formed by the vortex. Therefore, it will act as an accelerator to accelerate the wind velocity approaching the wind turbine. It was stated that vortex core will collapse the pressure field flow reversal and therefore no augmentation occurs [14]. This statement showed the alternative approach, where the vortex is used to generate a low pressure region in the wake of the turbine. Vortex should appear at the wake of the diffuser but as little as possible inside the diffuser wall.

If the vortices formed inside the diffuser, the pressure inside the diffuser might be lower than the pressure at the back of the wind turbine. In this circumstance, the wind speed exiting the diffuser augmented wind turbine will be decelerated and this only provides very low power coefficients. To get optimum increased acceleration on the wind speed, the pressure inside the wind turbine should not be lower than the pressure at the wake of the diffuser. This can be achieved by reducing or avoiding separation of fluid flow on the inner diffuser wall and cause vortices while creating as much vortices as possible at the back [15].

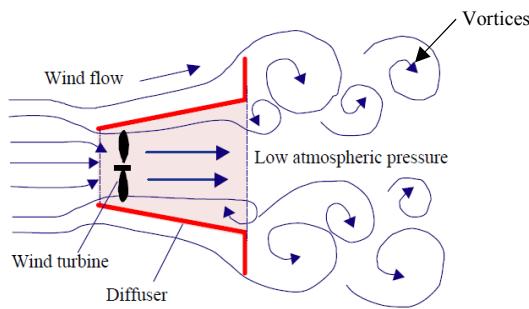


Fig. 2. Schematic Cross-Sectional View of a Flanged Diffuser and Wind Speed Increasing Flow Mechanism [15].

3. Methodology

3.1. Different design concepts and simulation model

Matsushima et al. [15] studied the effect of diffuser's shape on wind speed. Results showed that the wind speed in diffuser was greatly influenced by the length and expansion angle of the diffuser, and maximum wind speed increases 1.7 times with appropriate diffuser shape. As per the study, the wind speed in the diffuser was simulated by varying the external dimensions of the diffuser. Solidworks simulation software is used for modeling [16]. Wind speed in the diffuser is simulated for varying dimensions of the diffuser. The shape of the diffuser is shown in Fig. 3. The diffuser was set in the $20\text{ m} \times 10\text{ m}$ analysis space and after the uniform amount of wind was sent from the inflow inlet toward the

outlet as shown in Fig. 4. The following diffuser parameters were taken for simulating the wind speed: The diffuser's main body length (L), entrance diameter (D), its expansion or diffuser open angle (α) flange length (h) and splitter open angle (α_1). Simulations were conducted on the diffuser without a wind turbine in it.

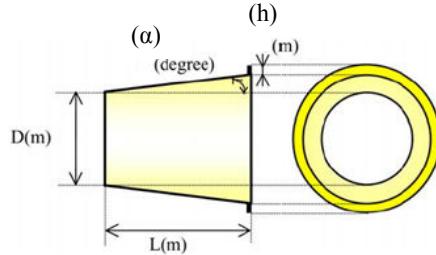


Fig. 3. External View of the Diffuser [15].

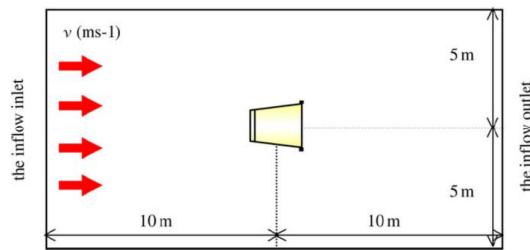


Fig. 4. Analysis Space [15].

Concept 1 is shown in Fig. 5. The dimension of the inlet shroud L , D , h will be fixed as 1 m, 0.4 m and 0.2 m respectively. α will be tested for different angle starting from 4° to 20° on different set of parameters [15].

Concept 2 is shown in Fig. 6. The splitter is a smaller diffuser within the main diffuser. In this concept the vortex would collapse the pressure field. Therefore, another smaller diffuser is placed inside with splitter angle α_1 and splitter length L_1 of 0.5 m and splitter diameter D_1 of 0.2 m with in main diffuser to direct and avoid separation and cause vortex inside the diffuser wall.

Concept 3 is shown in Fig. 7. The separator or splitter is a solid cone shape. To get the optimum air pressure at the wake of the diffuser and minimize separation of flow or vortices in the diffuser, few parameters will be changed and then the wind speed simulated. L , L_1 , D and D_1 are fixed as 1 m, 0.5 m, 0.4 m and 0.2 m respectively and the splitter or cone angle α_1 of 4° , 8° , and 12° were set with different diffuser open angle α of 4° and 8° .

Concept 4 is shown in Fig. 8. It was proposed to change the flow by adding another bigger diffuser, while ensuring a smooth flow inside the diffuser by putting a cone shape splitter. In this concept the ratio for L_2/L , D_2/D , was fixed at 0.5.

Initially, the dimensions L , D , space the length H and space height H_1 were set as 1 m, 0.4 m, 0.4 m and 0.02 m respectively. Then α_1 set as 4° , 8° , 12° and 16° .

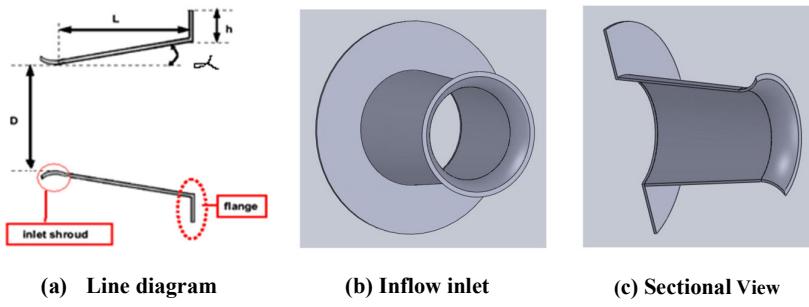


Fig. 5. Concept 1 - Flanged Diffuser with Inlet Shroud.

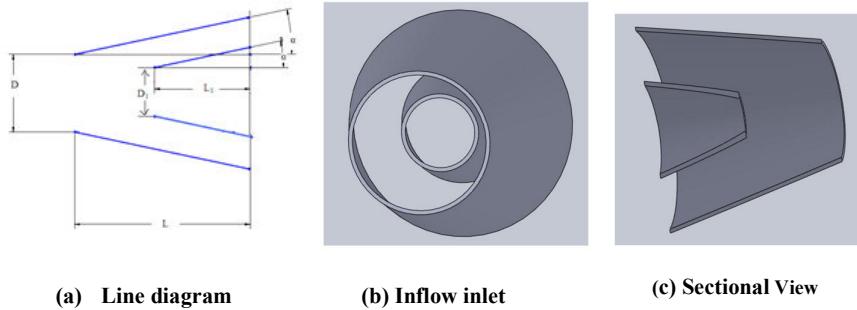


Fig. 6. Concept 2 - Small Inner Diffuser Splitter.

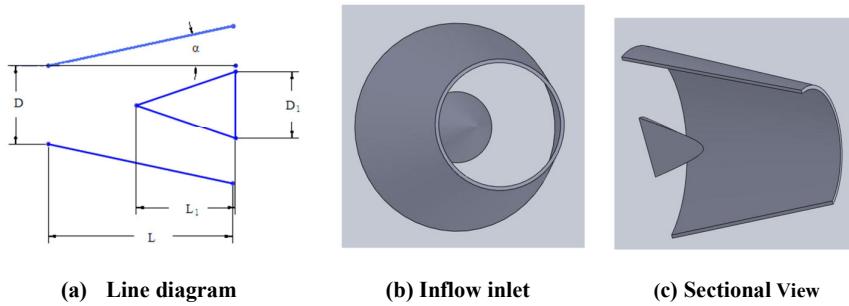


Fig. 7. Concept 3 - Cone Shape Splitter.

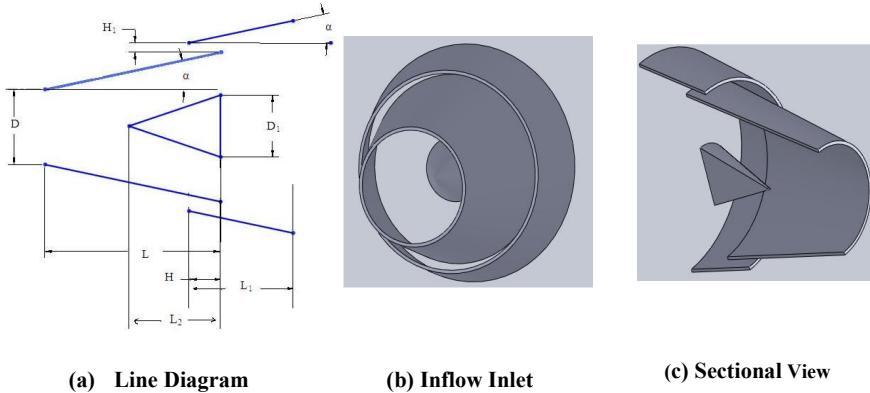


Fig. 8. Concept 4 - Double Diffuser with Cone Shape Splitter.

3.2. Computational Fluid Dynamics

ANSYS Fluent is used for simulation and prediction of fluid flow velocity. ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications [17].

In this simulation, it was assumed to be an symmetric incompressible steady flow with the following boundary conditions: initial wind velocity 4 m/s (with respect to the Malaysia average wind speed) as velocity inlet at the front boundary diffuser; inlet pressure surrounding the diffuser was set to atmospheric pressure; pressure inlet with 0 pressure input; diffuser wall was as stationary wall [18]. Surface meshing setting was set at 0.1 unit sizes. A single reference frames (SRF) model was used to simulate the incompressible and steady-state flow field. The incompressible steady flow is represents in the continuity equation given below:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (1)$$

In Eq. (1), ρ , t and \vec{u} denote density, time and velocity respectively.

4. Results and Discussion

Table 1 and Fig. 9 show the comparison of velocity at the entrance for each open angle α in concept 1. It was found that with an open angle of 12° , the wind velocity increased most significantly by 67.12% which is almost similar to Matsushima et al. [15] achievement of 68%. This is because of the flanges work well when combined with α of 12° to block 'back-flow' from outer wall of the diffuser while the inlet shroud direct better streamlines of the air flow through the diffuser. The lowest pressure occurs at the entrance of the diffuser is also increased the wind velocity at the entrance. It was noticed that the pressure at the rear of the diffuser was lower than atmospheric pressure and ideally lower than the inflow inlet for the diffuser to have optimum efficiency.

Table 1. Concept 1.

Diffuser open angle (α)	4°	8°	12°
Wind velocity at entrance (m/s)	5.37	6.28	6.68
% Increment (initial velocity 4 m/s)	34.25%	56.97%	67.12%

Table 2. Concept 2.

Splitter open angle (α_1)	4°	8°	12°
Wind velocity at entrance (m/s)	5.27	5.41	5.27
% Increment (initial velocity 4 m/s)	31.75%	35.25%	31.75%

Table 3. Concept 3(a).

Cone or splitter open angle (α_1)	4°	8°	12°
Wind velocity at entrance (m/s)	4.66	4.35	3.88
% Increment (initial velocity 4 m/s)	16.5%	8.75%	-2.91%

Diffuser open angle, $\alpha=4^{\circ}$ **Table 4. Concept 3(b).**

Cone or splitter open angle (α_1)	4°	8°	12°
Wind velocity at entrance (m/s)	4.83	4.93	4.68
% Increment (initial velocity 4 m/s)	20.75%	23.25%	17%

Diffuser open angle, $\alpha=8^{\circ}$ **Table 5. Concept 4.**

Cone or splitter open angle (α_1)	4°	8°	12°	16°
Wind velocity at entrance (m/s)	3.78	5.06	5.96	6.18
% Increment (initial velocity 4 m/s)	-0.06%	26.5%	49%	54.5%

From Table 2 and Fig. 10, the highest wind velocity at the entrance was recorded with an 8° open angle diffuser in concept 2. However, it was not the most efficient design concept because of the lack of flange which functions to blow the air flow outside the diffuser. The flange allows the accelerated air flow in the diffuser to flow outside without being interfered by the different velocity air flow. Based on the Table 2 and Fig. 10 it could be concluded that concept 2 is not a recommended design, but further design improvement would gain better results.

From Table 4 Concept 3(b) and Fig. 11, the highest recorded intake air velocity at the diffuser entrance occur using diffuser with 8° diffuser opening angle α and 8° cone opening angle α_1 . This is higher than Concept 3(a) in Table 3. But, this combination only managed 4.93 m/s at the entrance mid-point. The increased amount of the cone's opening angle was unable to increase the intake air velocity because of the increased area of back flow occurring right at the diffuser exit. However, it was discovered that the cone has the ability of focusing the air flow. As the diffuser and the cone opening angle are increased; the highest air flow tends to occur at the diffuser exit. This is because the cone shape alters the diameter of the area through which the air flows. This causes the pressure drop and thus increases the air velocity. If the cone is adapted as the turbine blades' center cap design, air flow across the turbine blade can be optimized by focusing the increased velocity air flow towards the blades.

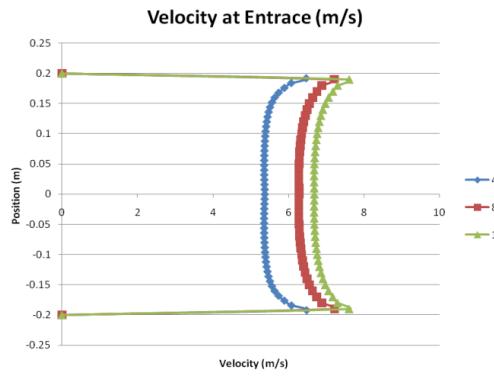


Fig. 9. Concept 1 - The Wind Velocity at the Entrance of the Diffuser with Different Diffuser Open Angles (α).

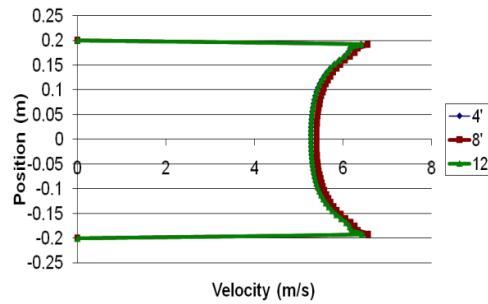


Fig. 10. Concept 2 - The Wind Velocity at the Entrance of the Diffuser with Different Splitter Open Angles (α_1).

Table 5 shows that wind velocity at diffuser inlet is proportional to the open angle in concept 4. Diffuser with open angle 16° was shown to be the highest increment of wind velocity in concept 4 with an increment of 54.5% because of low pressure inside the diffuser. But diffuser with open angle α of 4° happens to have a little decrement of 0.06%. It could be assumed that higher the open angle greater the wind velocity. Even though by increasing the angle, the wind velocity will increase but it can be predicted that there will be decrement at certain angle above 16° . It was observed from this concept that adding inlet shroud, flange and separator into the diffuser, the wind velocity growth was significant.

After performing all the simulation for the initial design, it was found that there are still rooms for improvement. It was realized from the analysis and study on the characteristic of the flow that inlet shroud can direct and provide a better flow direction and streamlines. While with flanges at the end of the diffuser, the back-flow which caused vortices inside the diffuser can be reduced significantly. However, this only happens at optimum angles because of higher the diffuser splitter angle generate more pressure and vortices inside the diffuser. From concept 2 and 3 it was noticed that by adding splitter inside the diffuser, the pressure at the exit of the diffuser will drop significantly. But, it only happens at optimum angles.

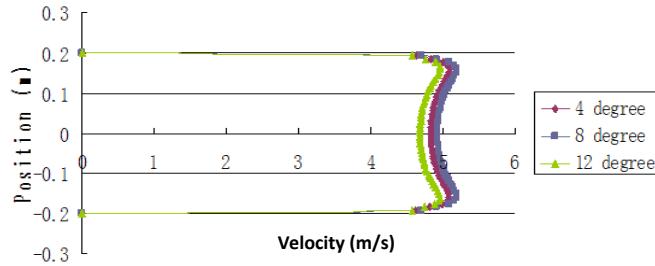


Fig. 11. Concept 3(b) - The Wind Velocity at the Entrance of the Diffuser with Different Cone Open Angles, α_1 , and Diffuser Open Angle, $\alpha=8^\circ$.

With further analysis, the characteristics provided by the diffuser type of splitter would be ideal than cone. The diffuser type of splitter tends to act like a nozzle at the end of the diffuser as the area circled in Fig. 12 and draw more air out ultimately reducing the back-flow from the diffuser. This idea is used in redeveloped design of Concept 4 to achieve better flow velocity. In that redeveloped design different open angle α of 12°, 16° and 20° and different α_1 of 12°, 16° and 20° was taken to analyze the wind velocity. An impressive 6.45 m/s, which is 61.25% increase over the 4 m/s air velocity before inlet was achieved with the 4° splitter opening angle α_1 and 16° diffuser open angle α as shown in Table 6 and Fig. 13. Unfortunately, the reduced splitter length did not help in increasing the efficiency although it did help to increase the air velocity of the diffuser with 12 ° splitter opening angle. The wind velocity contour and wind velocity vector profile for re-developed design were shown in Fig. 14 and Fig. 15. From the analysis and study on the characteristic of the flow, it was realized that inlet shroud can direct and provide a better flow direction and streamlines. While with flanges at the end of the diffuser, the back-flow which caused vortices inside the diffuser can be reduced. Also, flow field inside the diffuser is highly dependent on the motion of the turbine blades. In future study, momentum transfer to the blades will be considered to simulate the wind velocity.

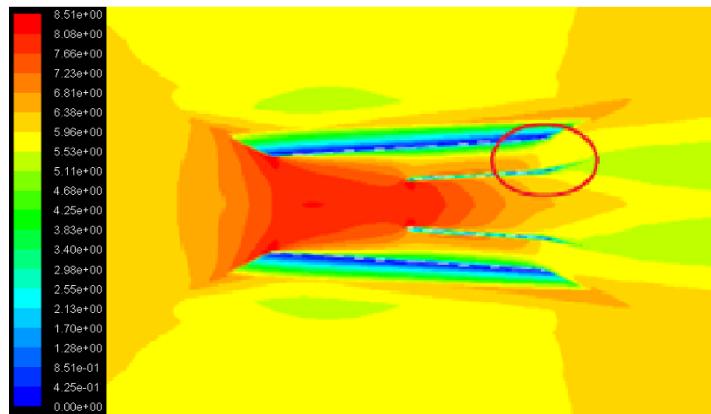


Fig.12 Sectional (Circled) Area Acts as Nozzle to Draw the Air Out in Concept 4.

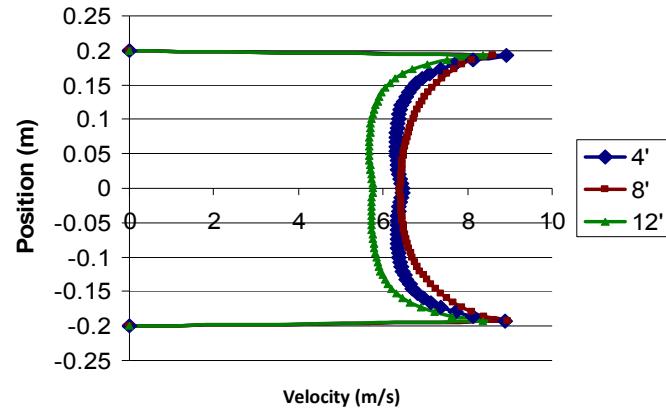


Fig. 13. Re-developed Design (Concept 4) - Wind Velocity at the Entrance of the Diffuser with Different Open Angle of the Splitter.

Table 6. Re-developed Design (Concept 4).

Splitter open angle (α_1)	4°	8°	12°
Wind velocity at entrance (m/s)	6.45	6.42	5.76
% of increment	61.25%	60.50%	44.00%
$\alpha = 16^\circ$, $L = 1.0$ m, $L_1 = 0.5$			

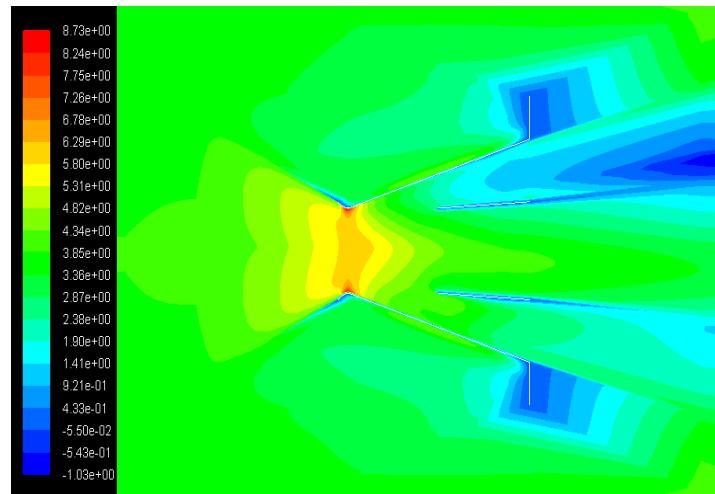


Fig. 14. Re-developed Design (Concept 4) - The Wind Velocity Contour for $L_1=0.5$ m and $\alpha_1=4^\circ$.

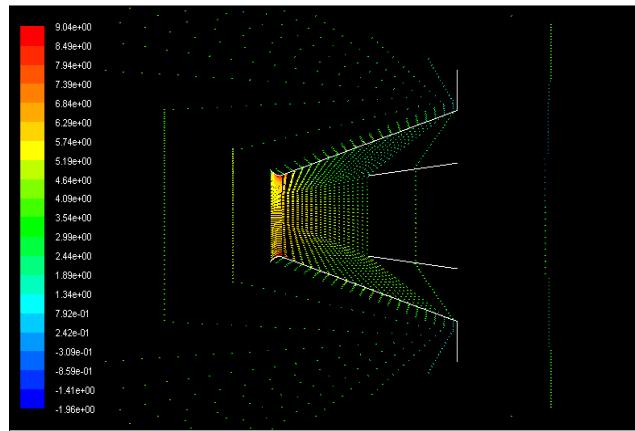


Fig. 15. Re-developed Design (Concept 4) - The Vector of the Wind Velocity.

5. Conclusions

This paper studies the different designs of Diffuser Augmented Wind Turbine using Solidworks and ANSYS Fluent. Results show that each aspect from each concept developed has perfection in diffuser efficiency and wind outlet velocity. A remarkable 6.45 m/s wind velocity with 61.25% augmentation was obtained using a 16° diffuser opening angle coupled with a 0.5 m splitter and 4° splitter opening angle in the redeveloped design.

The velocity contour plot diagram of the redesigned concept shows that the high velocity contours are actually occupying the mid area of the wind turbine blades. There is an agreement that the splitter and shroud concepts might prove valuable if the turbine fan design is integrated into the power augmentation analysis. It is realized that inlet shroud can provide a better flow direction and streamlines with the flanges at the end of the diffuser. Also the back flow which produces vortices inside the diffuser is also reduced. As suggestion, further work is necessary to optimize the diffuser design in terms of power coefficients by considering momentum transfer to the turbine blade.

Acknowledgement

The authors gratefully acknowledge Faculty of Engineering, Computing and Science, Swinburne University of Technology, Kuching, Sarawak for support by providing access to lab facilities and software such as ANSYS and Solidworks.

References

1. Ackermann, T.; and Soder, L. (2002) An overview of wind energy-status, *Renewable and Sustainable Energy Reviews*, 6(1-2), 67–127.
2. Frankovoc, B.; and Vrsalovic, I. (2001) New high profitable wind turbines, *Renewable Energy*, 24(3-4), 491-499.

3. Ueno, H.; and Serada, T. (2002) Wind turbine generation system & photovoltaic power generation system. *NTT Power Build Facil J*, 39(231), 42-45.
4. Van Bussel, G.J.W. (2007). The Science of making more torque from wind: diffuser experiments and theory revisited. *Journal of Physics: Conference Series*, 75, 1-12.
5. Bet, F.; and Grassmann, H. (2003). Upgrading conventional wind turbines. *Renewable Energy*, 28(1), 71-78.
6. Abe, K.-I.; and Ohya, Y. (2004) An investigation of flow fields around flanged diffusers using CFD. *Journal of Wind Engineering and Industrial Aerodynamics*, 92(3-4), 315-330.
7. Chen, T.Y.; Liao, Y.T.; and Cheng, C.C. (2012) Development of small wind turbines for moving vehicles: Effects of flanged diffusers on rotor performance. *Experimental Thermal and Fluid Science*, 42, 136-142.
8. Abe, K.; Nishida, M.; Sakurai, A.; Ohya, Y.; Kihara, H.; Wada, E.; and Sato, K. (2005) Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser. *Journal of Wind Engineering and Industrial Aerodynamics*, 93(12), 951–970.
9. Ohya, Y.; Karasudani, T.; Sakurai, A.; Abe, K.-I.; and Inoue, M. (2008). Development of a shrouded wind turbine with a flanged diffuser. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(5), 524-539.
10. Wang, F.; Bai, L.; Fletcher, J.; Whiteford, J.; and Cullen, D. (2008). The methodology for aerodynamic study on a small domestic wind turbine with scoop. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(1), 1-24.
11. Gardan, D.L.F.; and Bibeah, E.L. (2010). A numerical investigation into the effect of diffusers on the performance of hydro kinetic turbines using a validated momentum source turbine model. *Renewable Energy*, 35(6), 1152-1158.
12. Phillips, D.G.; Flay, R.G.J.; and Nash, T.A. (1999) Aerodynamic analysis and monitoring of the Vortec 7 diffuser augmented windturbine. *Transactions of the Institution of Professional Engineers New Zealand*, 26(1), 13-19.
13. Phillips, D.G.; Richards, P.J.; and Flay, R.G.J. (2000). CFD modelling and the development of the diffuser augmented windturbine. *In Proceedings of the Computational Wind Engineering*, Birmingham, 189-192.
14. Foreman, K.M. (1980). Preliminary design and economic investigations of diffuser augmented wind turbines (DAWT). *Final Report, Research Department, Grumman Aerospace Corporation*, Bethpage, New York and Solar Energy Research Institute, Colorado.
15. Matsushima, T.; Takagi, S.; and Muroyama, S. (2006). Characteristics of a highly efficient propeller type small wind turbine with a diffuser. *Renewable Energy*, 31(9), 1343–1354.
16. Solidworks User Manual by Solidwork Corp.
17. ANSYS Fluent User Manual by ANSYS Inc.
18. Sheila, W. (2009) *Potential flow calculations of axisymmetric ducted wind turbines*. Massachusetts Institute of Technology.