© Universiti Tun Hussein Onn Malaysia Publisher's Office

IJIE

http://penerbit.uthm.edu.my/ojs/index.php/ijie
ISSN : 2229-838X e-ISSN : 2600-7916

The International Journal of Integrated Engineering

CFD Investigation of Empty Flanged Diffuser Augmented Wind Turbine

Balasem Abdulameer Jabbar Al-Quraishi[1,2*], Nor Zelawati Asmuin[1], Nurul Fitriah Nasir[1], Noradila Abdul Latif[1], Juntakan Taweekun[3], Sofian Mohd[1], Akmal Nizam Mohammed[1], Wisam A. Abd Al-Wahid[2]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia (UTHM), 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA

[2]Engineering Technical College - Najaf, AL-Furat Al-Awsat Technical University, Najaf, IRAQ

[3]Department of Mechanical Engineering, Faculty of Engineering,
Prince of Songkla University, 15 Karnjanavanich Road, Hat Yai, Songkhla 90110, THAILAND

*Corresponding Author

DOI: https://doi.org/10.30880/ijie.2020.12.03.004
Received 20 May 2018; Accepted 20 September 2018; Available online 27 February 2020

Abstract: Enclosing a wind turbine within a flanged diffuser is an innovative mean to increase the power harvested by turbine blades and it is among the most effective devices for increasing wind turbine energy. The geometric parameters of the empty flanged diffuser contribute efficiently to increase mass flow in the diffuser, hence improve the turbine performance. The study presents developed models of the geometrical parameters of an empty flanged diffuser that suitable for a scaled-down (1-6.5) horizontal axis wind turbine, the geometry parameters were involved the diffuser length, diffuser angle, flange height and flange angle. The geometrical models were verified and CFD investigated in 2-D and 3-D domains. Results obtained from CFD simulations show that, using a compact size of flanged diffuser within optimum geometrical parameters can give well acceptable for flow velocity increase at suggested place for the turbine rotor install. The increase in flow velocity is due to lower pressure at the outlet of the diffuser. As there is also a significant effect of the flange angle on increasing the flow velocity inside the diffuser where the rate of increase in wind velocity at turbine position was calculated for two flange angles (0° and 5°). In another hand, the results also provided information on the velocity contours and velocity streamlines around diffuser geometry.

Keywords: Wind energy harvesting, DAWT, flanged diffuser.

1. Introduction

The need for energy to consume society increases as technology advances in certain areas, so the capability to produce energy must keep pace with increasing demands. Due to the rapid depletion of fossil energy sources, there is a necessary need to seek alternative and sustainable sources of energy. However, wind energy as a renewable and inexhaustible source of energy is now the fastest-growing energy technology worldwide (T. Wei, 2010). Wind power systems, represented by wind turbines have been the focus of interest of scientists and researchers in the past decades. Flowing of wind through the turbine rotor leads to the production of mechanical energy that can be used in many applications especially to produce electricity.

However, power produced by the wind turbine is dependent on the Betz limit; an ideal type can extract only 59.3% of incoming energy in stream-tube by turbine blades (Libii & Drahozal, 2012), (IGRA, 1981). As the energy extracted

*Corresponding author: balasemalquraishi@atu.edu.iq; balasemalquraishi@gmail.com

from wind is increasing as the cube of wind speed increase, consequently, any increment in wind velocity at turbine rotor plane leads to a large increase in wind energy output (IGRA, 1981).

Several innovative concepts have been proposed to augment wind turbine power output. Performance of wind turbines can be improved by various ways such as modification of blades design, the augmentation attempts by application of tip vanes on the rotor blades, vortex type augmentation devices and ducted wind turbine by shrouds, concentrators or diffusers. Most of the researchers show that the Diffuser Augmented Wind Turbine (DAWT) exhibits advantages compared to other augmentation solutions (Monir Chandrala, Abhishek Choubey & Department, 2013), (van Dorst, 2011). Thus, if the turbine is shrouded by a diffuser to capture wind deliver to the rotor, more power will be produced for a given turbine diameter and wind speed thus exceeding Betz limit (M.Natesan, Dr.S.Jeyanthi, & Sivasathya, 2017). In other words, the performance increase of DAWT is proportional to the mass flow through the duct and that larger performances are possible by lowering back pressure levels at the diffuser exit (Ghajar & Badr, 2008).

Several studies were conducted between the 1950s and late 1990s, which studied the use of huge diffusers with large angles, focused heavily on the flow control mechanism to increase the mass flow rate within the diffuser [8-11]. Unfortunately, the high cost of the energy produced by the proposed diffusers doesn't let them reach the commercialization stage. It should be noted that the general economic context where these research had been performed did not promote developing renewable energy. However, in the early 2020s, the diffuser has become an attractive research topic increasing wind turbine power, hence various types and shapes of diffusers have been studied [12-15]. Consequently, in recent years, several studies have focused on the development of DAWT (Schaffarczyk, 2014), particularly the optimal design of wind turbine shroud and the best ways to increase the power augmented.

The performance enhancement of DAWT depends on several factors including the diffuser shape and geometries, blade airfoils, and wind condition at the mounting site. The Geometric features are the main parameters controlling the aerodynamic performances of this wind-energy device. The term of (empty flanged diffuser) indicates DAWT without turbine rotor, in other words only shroud [17]. The geometry of shroud is denoted by dimensions of the diameter of inlet diffuser which is related by rotor diameter (D), diffuser length (L), diffuser angle (Θ) and brim (flange) height (H) as shown in Fig. 1 [18]. Each one of these geometrical parameters has a direct or indirect effect on DAWT performance.

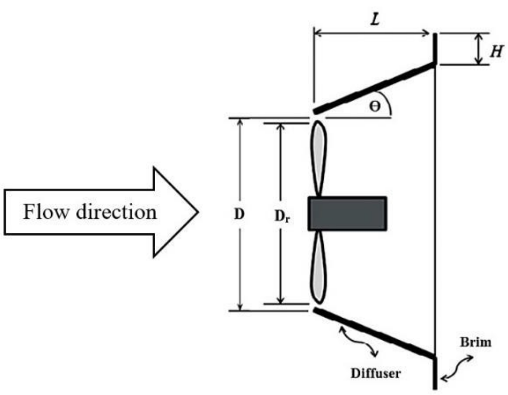

Fig. 1 - Typical design for a wind turbine shrouded by a diffuser-brim [18]

Where the diffuser geometric features are the main parameters to control the flow of wind through the rotor plane, so many experimental and simulation studies on developed its design were done. Ohya et al. (Ohya, Karasudani, Sakurai, Abe, & Inoue, 2008) found that using a diffuser with a proper flange of height attached to its outlet, gives a remarkable increase in wind speed of 1.6-2.4 times hence power augmentation of DAWT up to 4-5 times of bare turbine. Numerical simulations and wind tunnel experiments have been carried out by R. Chaker et al [19] on seventeen open anglesranged from 0° to 35°. Results showed that wind velocity increases as open-angle increase. The optimum angle was 10° for the empty diffuser and 12° for the DAWT with velocity increase of 1.76 times and 1.45 times respectively. Numerical 2D symmetry model was used by M. Lipian et al [20] to study the sensitivity of angle and brim height of diffuser on the DAWT design The CFD analysis result showed the best increase in wind speed was 1.6 at an angle ($2Θ=6^o$). A small wind turbine with a simple frustum diffuser has been simulated by CFD at different sizes of diffuser length (L/D) in range of (0.1-0.4) and area ratio (H/D). The result showed that an increase of H/D with constant L lead to the expansion of flow through the diffuser, hence increase the rotor power. In another hand, the extensive increase of L/D lowers turbine power coefficient. Hence there is an optimum length factor for a frustum diffuser [21]. The terms of empty flanged diffuser was used to expressed on.

In the present paper, a CFD simulation to investigate the effect of geometrical parameters of flanged diffuser ((L/D), (H/D), (Θ) and flange angle ($Θ_f$)) on wind velocity at place of rotor installation and find out an optimum values of these parameters for compact design that achieve optimum increase in velocity ratio (ϵ) at sections of blade (airfoils) for turbine rotor (in scale of 1-6.5). As well as compare the 2D and 3D simulation results in two packages of ANSYS (CFX and Fluent) at different turbulence models.

2. Methodology

Since the study was starting by reviewing many previous studies and selected the best values of the geometrical parameters. In order to investigate the effect of geometrical parameters on the wind acceleration in the flanged diffuser by numerical simulations, a methodological sequence of this study was adopted, starts with verifying the previous study followed by development the diffuser geometry based on the previous model. Then, a new model was used to investigate the optimum geometries for flanged diffuser CFD simulation in 2D and 3D.

2.1 Model Verification

The present study was starting by 2-D axisymmetric diffuser design model presented by El-zahaby [22] which have model parameters L/D=1.5, Θ=3.18^{o} and H/D=0.25 (D is the diffuser inlet diameter) with a range of flange angles with the vertical axis (-25 to +25). Length and height of the simulation model were 15D and 7.5D and diffuser inlet location is 4D, where, the verification results with it as shown in Fig. 2, as well as, the statistic of grid independence which shows the test of mesh metric in term of skewness for the domain, where the best value was in the range of (0.5 - 0.6) at element number of 27784.

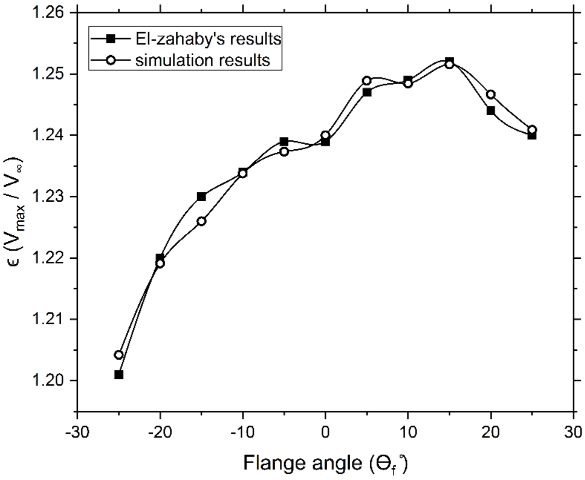

Fig. 2 - Verification curve for 2-D simulation result of ϵ for maximum velocity at the diffuser inlet

2.2 Develop the Geometrical Parameters

In order to investigate the effect of modify the geometrical parameters on diffuser performance, the procedure of modification design of diffuser in this paper was based on achieving the best value of ϵ (ratio of maximum velocity at inlet diffuser to upstream velocity) at a different range of diffuser parameters (L, Θ, H and flange angle Θ_f) in 2-Dimensions axisymmetric by "Fluent CFD" based on El-zahaby's model. Table 1 shows the geometrical parameters that have been investigating with consideration the optimum values of (H and Θ) that has been presented by [23] based on experimental and simulation analysis of 2-D diffuser model, as well as compacting the diffuser length to reduce the cost with optimum performance. The optimum values of geometrical parameters that achieved the highest value of ϵ are cases No.9 and 10.

2.3 Simulation Description

After verifying the model of the previous study and achieve optimum values for geometrical parameters of the diffuser, a new model was used to investigate the simulation. The described of the CFD simulation model was used in this paper based on dimensions of the test section of Wind Tunnel (125 x 40 x 40 cm) in aerodynamic Lab, Faculty of Mechanical Engineering and Manufacturing (FKMP), University Tun Hussein Onn Malaysia (UTHM) to investigate an empty DAWT suitable for (a scaled-down 1-6.5) of rotor diameter was presented by [24] . The diffuser inlet diameter (D=16 cm), diffuser length (L=0.5 D), flange height (H=0.2 D) and the diffuser expansion angle (Θ) is 12° with 3mm diffuser wall thickness.

2.3.1 Two Dimensional domain

To limit the size of the computational task, a 2D axisymmetric domain was used to the investigating as shown in Fig.3. The diffuser was placed at 60 cm from the inlet test domain. No-slip wall for a diffuser and bottom domain were considered, as well as the model is symmetric about the axis. The 2D simulation carried out using two packages in ANSYS 19.1, Fluent, and CFX by using the same 2D axisymmetric domain with little difference by adding a 1 mm as a thick for the domain in case of simulating by CFX package due to it is unable to consider 2D flow [25]. The mesh type was quadrilateral and independence was tested to check the mesh quality in terms of skewness and Orthogonal quality as shown in Table 2.

Table 1 - The achieved values of (ϵ) as a function of the geometrical parameters of an empty flanged diffuser

No.	L/D	ϴ	H/D	$ϴ_f$	ϵ
1	1.473	3	0.3	0	1.2648
2	1	4.5	0.25	0	1.282
3	0.5	9	0.2	0	1.323
4	0.5	9	0.2	5	1.319
5	0.5	10	0.2	0	1.3367
6	0.5	10	0.2	5	1.337
7	0.5	11	0.2	5	1.346
8	0.5	11	0.2	0	1.349
9	0.5	12	0.2	0	1.361
10	0.5	12	0.2	5	1.3589
11	0.4	9	0.2	0	1.320
12	0.4	10	0.2	5	1.3338
13	0.4	10	0.15	0	1.281
14	0.4	11	0.2	0	1.339
15	0.4	12	0.2	0	1.345
16	0.4	12	0.2	5	1.344
17	0.3	10	0.2	0	1.327
18	0.3	11	0.2	0	1.335
19	0.3	12	0.2	0	1.343
20	0.3	12	0.2	5	1.340

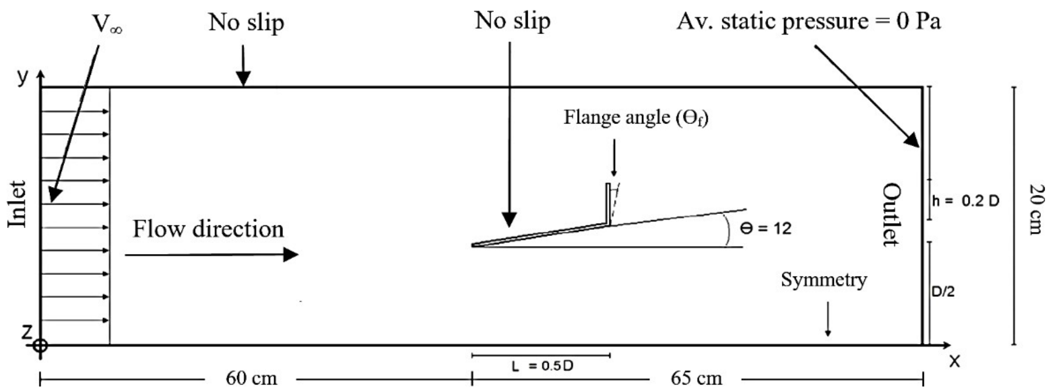

Fig. 3 - The 2-D axisymmetric simulation domain for the flanged diffuser

Table 2 - The statistic of grid independence for 2-Dimention axisymmetric domain

CFD package	Number of elements	Number of nods	skewness	Orthogonal quality
2D Fluent	15743	16131	0.49	1
2D CFX	15767	32308	0.476	1

2.3.2 Three dimensional domain

For high accuracy of flow investigation, the numerical model in a 3D domain by using CFX was carried out for a full geometry of the empty flanged diffuser model and the test section of the wind tunnel (FKMP, UTHM) as shown in Fig.4. The grid independence for 3D diffuser domain is listed in Table 3. The mesh type was Tetrahedron, trial 1 was adopted for suitable numbers of elements and nods, due to the best values for the skewness and orthogonal, high meshing quality was represented by the high value of orthogonal quality and lower value of skewness [26].

Table 3 - The statistic grid independence for 3-D diffuser domain

Trial No.	No. of elements	No. of nods	skewness	Orthogonal quality
1	851420	153991	0.78	0.996
2	1704436	308071	0.799	0.996
3	3400811	613622	0.799	0.996

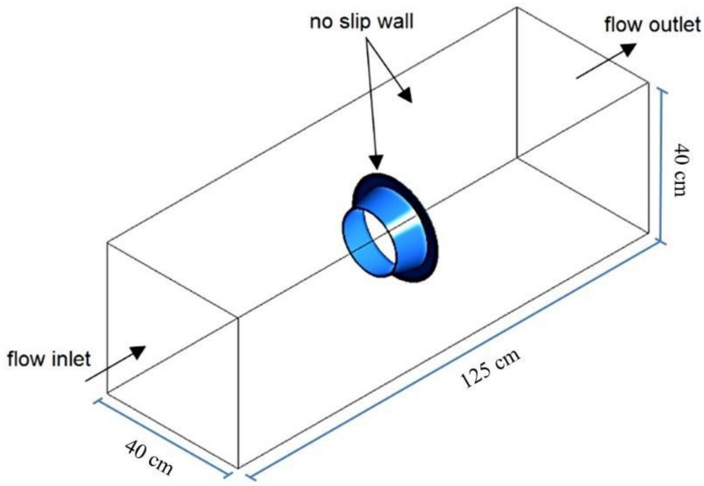

Fig. 4 - The 3-D CFX simulation domain for the flanged diffuser

3D CFX adopted instead of 3D Fluent because CFX uses the vertex-centred method, more precisely dual-median method while Fluent uses a cell-centred method which has a larger number of degrees of freedom but fewer fluxes per unknown making the scheme computationally expensive. The accuracy of the method is affected negatively by the low number of fluxes per unknown making it hard to determine which of methods has better accuracy [27].

The boundary and initial conditions for the two ANSYS 19.1 solvers in 2D and 3D are presented in Table 4. The air was modelled as an ideal gas at 25∘c with reference pressure in all domains equal to 1 atm. The flow was treated as an isothermal and steady-state with a 10-5 convergence criterion. The spatial discretization schemes are high resolution for CFX, and second-order differencing for Fluent which are considered the optimal offered, Although CFX uses the scheme high-resolution which is a combination of the two schemes, however, the results from CFX and second-order agree well between each other. [28].

Table 4 - The boundary and initial conditions for the two ANSYS solvers in 2D and 3D

Boundary	2D Fluent	2D CFX	3D CFX
inlet	Velocity (with zero intensity)	Velocity (with default intensity)	Velocity (with default intensity)
outlet	Average static pressure 0 (Pa)	Average static pressure 0 (Pa)	Average static pressure 0 (Pa)
No-slip wall	Top, diffuser wall	Top, diffuser wall	Top, bottom, the two sides, diffuser wall
symmetry	Bottom	Bottom, the two sides	-
Turbulence model	(k-e)	(k-e) and (SST k-w)	(SST k-w)

3. Results and Discussions

As the amount of wind energy of the air passing the rotor area is proportional to its mass flow rate. So, there are some methods to increase this mass flow. One of the most effective methods is shrouding the rotor with a flanged diffuser. Since the geometrical diffuser parameters control the diffuser efficiency, a different range of these parameters was investigated by CFD, the optimum values for velocity increase were recorded at L=0.5D, H=0.2D, ϴ =12 as shown in Table 1. Where the explanation of this increase in speed due to the decrease in downstream (diffuser exit) pressure significantly because the angle of the large publisher compared to the previous design although it had high (L/D) ratio. In Fig.5 a comparison of the maximum increase in flow velocity at suggested install place for a turbine rotor (0.0625D a distance from inlet diffuser) for a wide range of wind speeds 1-15 m/s. based on previous study model presented by El-zahaby's. It can be observed that there is a significant increase in flow velocity in the modified geometrical models of diffuser compared to the best design that was presented in the previous study. Moreover, the model with a flange angle (Θ_f) = 0˚is little more than that within Θ_f= 5˚.

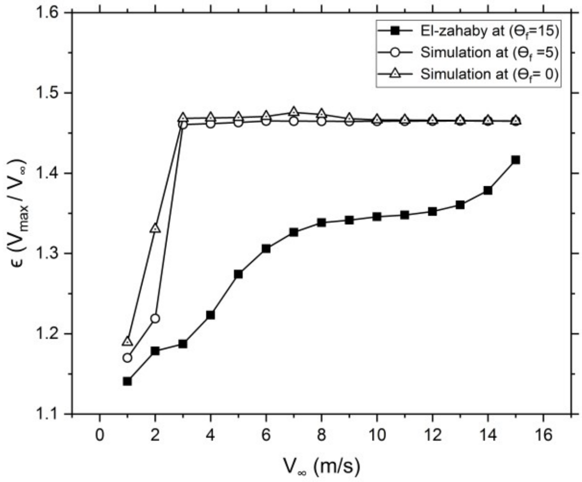

Fig. 5 - The maximum increase of flow velocity at the suggested place of rotor install based on the previous study model

To compare the optimum geometrical design based on 2D simulation domain in Fluent solver using k-e turbulence model, Fig. 6 show the average increase of inlet velocity at suggested install place for a turbine rotor (0.0625 D from diffuser inlet) for upstream velocities ranged 1-15 m/s. It can be observed that diffuser with Θ_f = 0˚ has a little higher efficiency compare diffuser with Θ_f= 5 so, the two geometrical models are applicable.

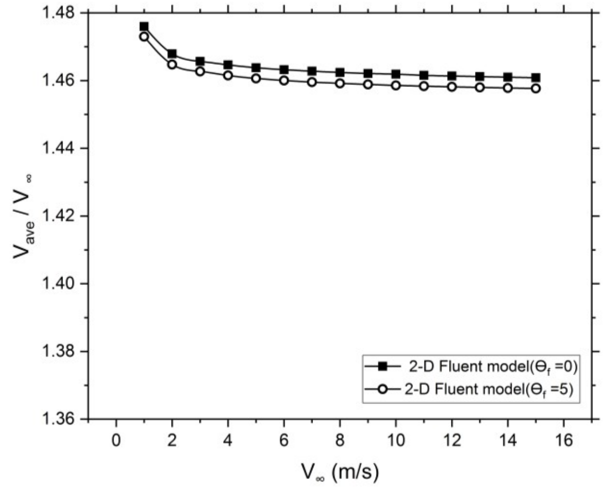

Fig. 6 - The average increase of inlet velocity at suggested install place for a turbine rotor based on 2-D simulation model by Fluent solver

The second set of analyses examined the impact of turbulence models on the flow velocity at 0.0625, a distance from diffuser inlet, for the radial positions (ten sections) along of suggested place of a rotor blade as shown in Fig.7. The simulation domain (based on wind tunnel dimensions) the flow stream velocity was 10 m/s. Two types of turbulence models were simulated, (k-e) and (SST k-w). Due to (k-e) was used in the previous, study, it was used in this study too with 2D-Fluent and 3D-CFX domains to obtain compatibility and convergence of the two models. On another hand, within 2D and 3D domain using CFX Solver, SST k-w turbulence model was used, according to Menter study [29] on different types of turbulence models (SST k-w, K-e, K-w BSL and k-w org.) for engineering applications. the study proved that SST k-w gives a very good prediction for the results matches the actual results; many research adopted SST K-w for 3D simulation [30],[31], [32],[33].

The results, as shown in Fig. 8, indicate that a little difference in increasing of ϵ value among the four models used and the more stability relatively with turbulence model SST k-w at 2D and 3D CFX Solver. Also, it can observe ϵ value along with places of the blade sections gradually increase from the root of a blade with a very simple difference, until it decreases at the position of the tip blade due to the boundary layer of the diffuser wall. On another hand, these results are very important to consider in a suitable turbine blade design for DAWT.

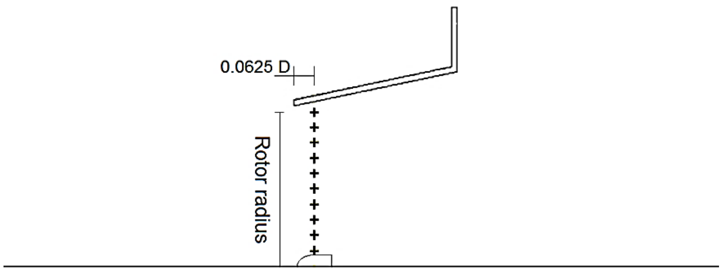

Fig. 7 - The suggestion place of turbine rotor with radial positions along of the blade

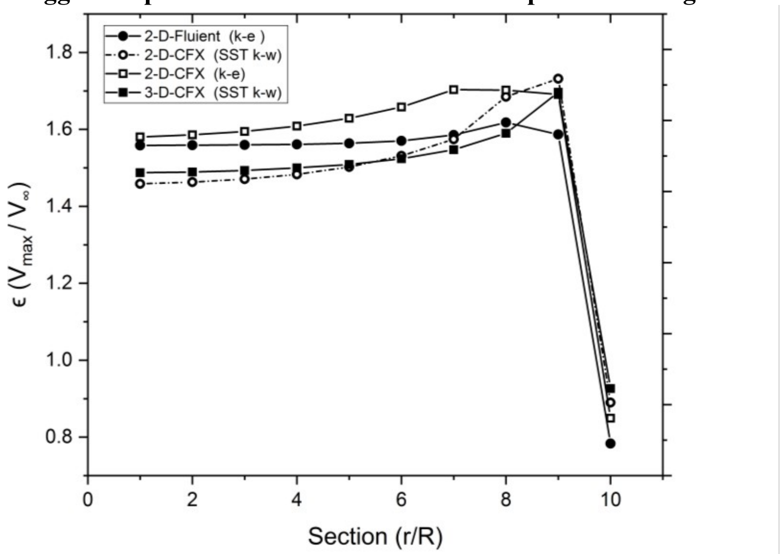

Fig. 8 - A compression of ϵ value at different turbulence model in ANSYS solver

3D CFD results include velocity magnitude at any point in the simulation domain. Also, it is easy to extract velocity streamlines and velocity contours. Figures, 9 and 10 show the velocity contours inside and around the diffuser with flange angles of 5˚and 0˚respectively at free stream velocity of 10 m/s. These figures indicate an increase of velocity at diffuser entrance (especially at the place of rotor install) and the decrease of velocity area behind flange due to the pressure drop at diffuser exit (behind the flange) as shown in Fig.11, pressure distribution at diffuser with Θ_f= 0 ˚.The pressure drop at diffuser exit is due to the generation of vortices behind the flange, as these vortices reduce pressure and thus increase the flow rate through the diffuser. The more these vortices are generated, the more likely the flow of air increase, the speed will be increased at the rotor place installed. The difference between the generation of these vortices for Θf =5˚and Θf =0˚can be observed from Fig. 12 and Fig.13 respectively.

Moreover, the results from the 3D domain which carried out using CFX solver have been observed more details for flow through the diffuser. Fig. 14 shows the average increase of inflow velocity at 1 cm (0.0625 D a distance from diffuser inlet) along the position of rotor radius (7.7 cm) for an empty diffuser with flange angles (5˚and 0 at velocities 1-15 m/s, where it can be observed that the overall average of increase in average velocity almost equal in all range of velocities where it is 1.44 for Θ_f = 5 ˚,while it reaches to 1.47 for Θ_f = 0 ˚.These predictions are identical to those of previous studies by Ohya [15], R. Chaker [19], and M. Lipian [20] which they achieved a significant increase in wind velocity about 1.6-2.4 times, 1.76 times, and 1.6 times, for the three studies respectively

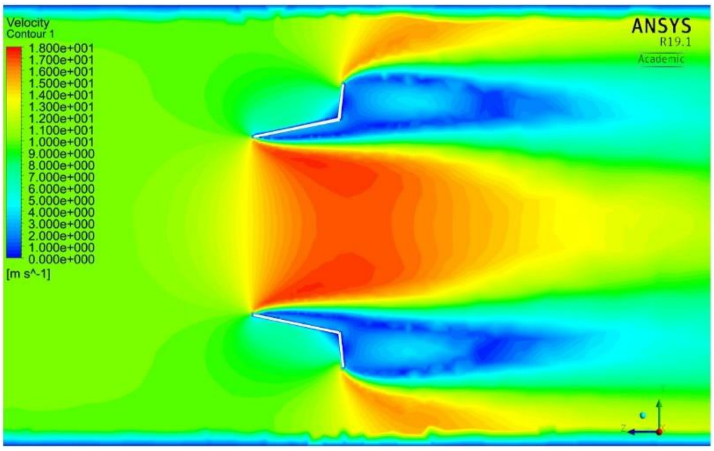

Fig. 9 - Velocity contour of the flow at diffuser with $\Theta_f = 5$

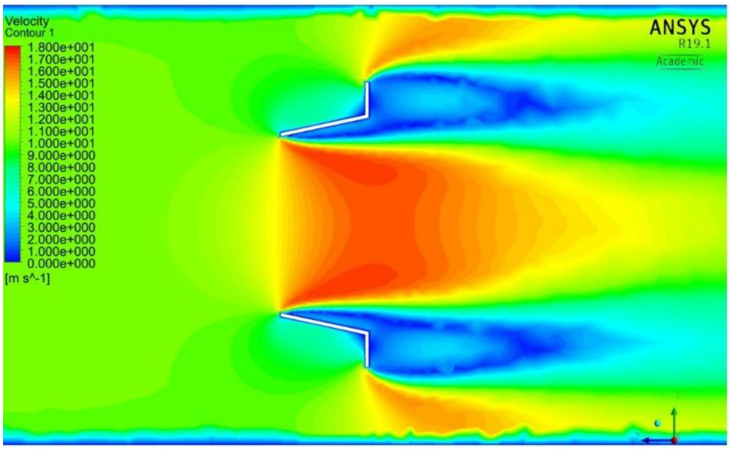

Fig. 10 - Velocity contour of the flow at diffuser with $\Theta_f = 0$

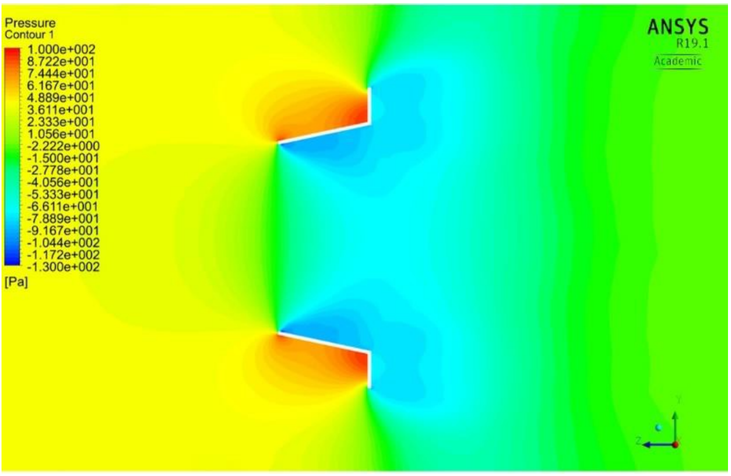

Fig. 11 - Pressure contour of the flow at diffuser with $\Theta_f = 0^{\circ}$

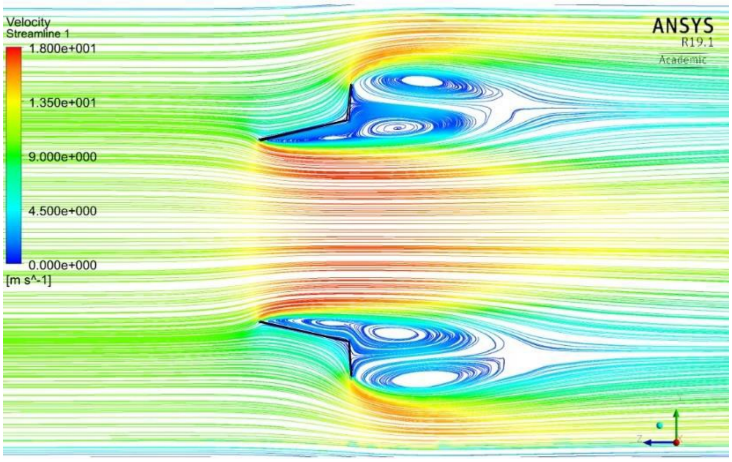

Fig. 12 - Velocity streamline of the flow at diffuser with Θ_f =5

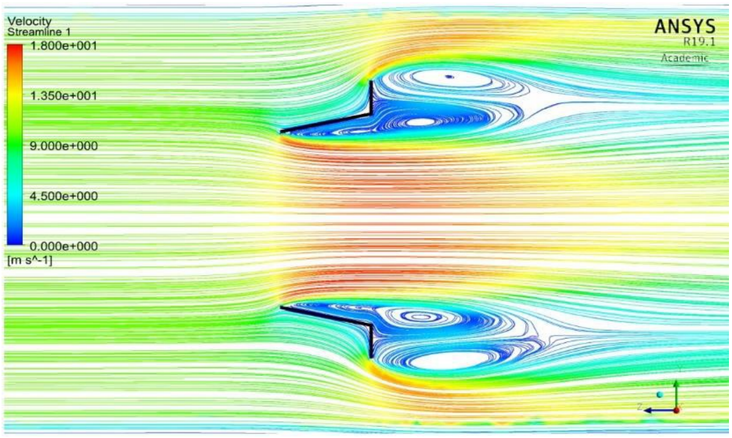

Fig. 13 - Velocity streamline of the flow at diffuser with Θ_f =0

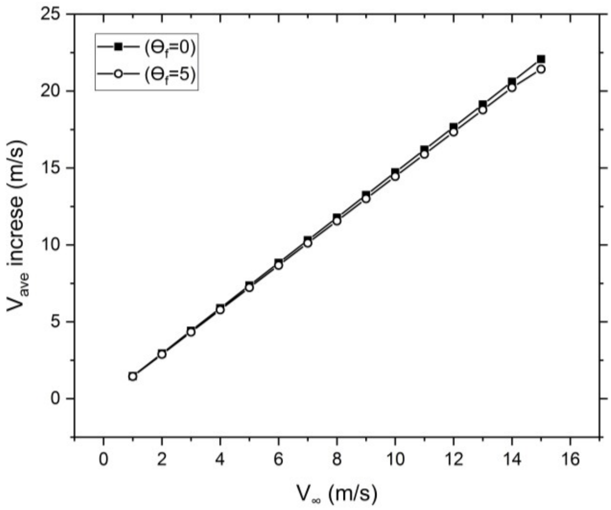

Fig. 14 - The average increase of flow velocity at the suggested place of rotor install

4. Conclusion

This paper was aimed to introduce a developed geometry model of empty flanged diffuser suitable as a shroud for scaled-down (1-6.5) horizontal axis wind turbine, it is able to increase the flow velocity at suggested place rotor install. Two modified geometries models of an empty diffuser were investigated CFD simulation in a two ANSYS solver (CFX and Fluent) at different turbulence model to study the effect of geometrical parameters of the empty diffuser in 2-D and

3-D. The two geometries models had the same geometrical parameters except for their differences in flange angle, where one of them have Ɵf =5 ˚,while the other one had Ɵf = 0˚. The simulation results were showed the ability of the compact geometry model of the flanged diffuser to increase the flow velocity at the turbine place for a wide range of velocities. The overall average of increase in flow velocity reaches up to 1.47 times, where this increase in velocity is due to the presence of flange at an appropriate height at the outlet of the diffuser which causes the generation of vortices behind the flange that produces a pressure drop at the outlet of the diffuser. On another hand, the simulation results were calculated the maximum increase in flow velocity at different positions (ten positions) along of turbine blade, where these results are very important to consider in a suitable turbine blade design for DAWT.

Acknowledgement

The authors would like to acknowledge the Universiti Tun Hussein Onn Malaysia (UTHM), AL-Furat Al-Awsat Technical University, and Prince of Songkla University (PSU).

References

[1] T. Wei. (2010). *Wind Power Generation and Wind Turbine Design*.

[2] Libii, J. N., & Drahozal, D. M. (2012). The influence of the lengths of turbine blades on the power produced by miniature wind turbines that operate in non-uniform flow fields. *World Transactions on Engineering and Technology Education*, *10*(2), 128-133.

[3] IGRA, O. (1981). Research and Development for Shrouded Wind Turbines. *Energy Conversion and Management*, *21*, 13-48.

[4] Monir Chandrala, Abhishek Choubey, B. G., & Department. (2013). CFD analysis of horizontal axis wind turbine blade for optimum value of power, *4*(5), 825-834.

[5] van Dorst, F. A. (2011). *An Improved Rotor Design for a Diffuser Augmented Wind Turbine*. Eindhoven University of Technology.

[6] M.Natesan, Dr.S.Jeyanthi, & Sivasathya, U. (2017). a Review on Design of Augmented Wind Turbine Blade for F or Low Wind Speed, *8*(7), 685-691.

[7] Ghajar, R. F., & Badr, E. A. (2008). An Experimental Study of a Collector and Diffuser System on a Small Demonstration Wind Turbine. *International Journal of Mechanical Engineering Education*, *36*(1), 58-68.

[8] Lilley, G. M., & Rainbird, W. J. (1956). *A Preliminary Report on the Design and Performance of a Ducted Windmill*. report 102, College of Aeronautics. Cranfield, UK.

[9] A. Kogan and A. Seginer, "Final report on shroud design," *Dep. Aeronaut. Eng. Tech.*, p. Report No. 32A, 1963.

[10] IGRA, O. (1984), "Research and Development for Shrouded Wind Turbines," *Eur. Wind Energy Conf. 22-26 October, 1984, Hamburg, Germany*.

[11] Foreman, K.M. (1981) "Preliminary Design and Economic Investigations of Diffuser Augmented Wind Turbines (DAWT)," Final Report, Research Department Grumman Aerospace Corporation Bethpage, New York.

[12] Hansen, M. O. L., Sørensen, N. N., & Flay, R. G. J. (2020). Effect of Placing a Diffuser around a Wind Turbine. *Wind Energy*, *3*(4), 207-213. https://doi.org/10.1002/we.37

[13] J.M Werle and Presz W.M. (2008) "Ducted Wind/Water Turbines and Propellers Revisited," *J. Propuls. Power*, vol. 24, no. 5, pp. 1146-1150.

[14] Kumar A., Sachendra. (2017) "Development in Augmented Turbine Technology," *Int. J. Trend Res. Dev.*, vol. 4, no. 1, pp. 2394-9333.

[15] Ohya, Y., Karasudani, T., Sakurai, A., Abe, K. ichi, & Inoue, M. (2008). Development of a shrouded wind turbine with a flanged diffuser. *Journal of Wind Engineering and Industrial Aerodynamics*, *96*(5), 524-539. https://doi.org/10.1016/j.jweia.2008.01.006

[16] Schaffarczyk, A. P. (2014). Introduction to Wind Turbine Aerodynamics. *Green Energy and Technology*, *153*, 7-21. https://doi.org/10.1007/978-3-642-36409-9

[17] M. Kardous, R. Chaker, F. Aloui, and S. Ben Nasrallah. (2013) "On the dependence of an empty flanged diffuser performance on flange height: Numerical simulations and PIV visualizations," Renewable Energy, vol. 56, pp. 123-128.

[18] Kosasih, B., & Tondelli, A. (2012). Experimental study of shrouded micro-wind turbine. *Procedia Engineering*, *49*, 92-98. https://doi.org/10.1016/j.proeng.2012.10.116

[19] Chaker, R., Kardous, M., Aloui, F., & Nasrallah, S. Ben. (2014). Open angle effects on the aerodynamic performances of a flanged Diffuser Augmented Wind Turbine (DAWT). *Conférence Internationale Des Energies Renouvelables*, *2*.

[20] Lipian, M., Karczewski, M., & Olasek, K. (2015). Sensitivity study of diffuser angle and brim height parameters for the design of 3 kW Diffuser Augmented Wind Turbine. *Open Engineering*, *5*(1), 280-286. https://doi.org/10.1515/eng-2015-0034

[21] Jafari, S. A. H., & Kosasih, B. (2014). Flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations. *Journal of Wind Engineering and Industrial Aerodynamics*, *125*, 102-110. https://doi.org/10.1016/j.jweia.2013.12.001

[22] El-zahaby, A. M., Kabeel, A. E., Elsayed, S. S., & Obiaa, M. F. (2016). CFD analysis of flow fields for shrouded wind turbine ' s diffuser model with different flange angles. *Alexandria Engineering Journal*, 0-8. https://doi.org/10.1016/j.aej.2016.08.036

[23] Kardous, M., Chaker, R., Aloui, F., & Abidi, I. (2016). Locations of vortices and their impacts on the aerodynamic performances of a diffuser and a DAWT. *2016 3rd International Conference on Renewable Energies for Developing Countries, REDEC 2016*. https://doi.org/10.1109/REDEC.2016.7577536

[24] Lee, M.-H., Shiah, Y. C., & Bai, C.-J. (2016). Experiments and numerical simulations of the rotor-blade performance for a small-scale horizontal axis wind turbine. *Journal of Wind Engineering and Industrial Aerodynamics*, *149*, 17-29. https://doi.org/10.1016/j.jweia.2015.12.002

[25] Michal, L., MacIej, K., Jakub, M., & Krzysztof, J. (2016). Numerical simulation methodologies for design and development of Diffuser-Augmented Wind Turbines-analysis and comparison. *Open Engineering*, *6*(1), 235-240. https://doi.org/10.1515/eng-2016-0032

[26] Mentin Ozen, “Mesh Metric Spectrum Quality,” California: Ozen Engineering.

[27] Rutvika Acharya, “Investigation of Differences in Ansys Solvers CFX and Fluent,” Master thesis in Fluid Dynamics, Mechanics Institution, Royal Institute of Technology, KTH Stockholm, June 2016

[28] Acharya, R. (2016). Investigation of Differences in Ansys Solvers CFX and Fluent. *Royal Institute of Technology, KTH Stockholm*, (June).

[29] Menter F.R. (1994) “Two-equation eddy-viscosity turbulence models for engineering applications,” *AIAA J.*, vol. 32, no. 8, pp. 1598-1605.

[30] Tourlidakis, A., Vafiadis, K., Andrianopoulos, V., Kalogeropoulos, I. (2013) “Aerodynamic design and analysis of a flanged diffuser augmented wind turbine,” Proceedings of ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, San Antonio, Texas, USA.

[31] Kokasih, B., Saleh Hudin, H. (2016) “Influence of inflow turbulence intensity on the performance of bare and diffuser-augmented micro wind turbine model,” Renewable Energy, vol. 87 , pp. 154-167.

[32] Balasem Abdulameer J. Alquraishi, N. Z. Asmuin, S. Mohd, W. A. Abd Al-Wahid, and A. N. Mohammed, “Review on Diffuser Augmented Wind Turbine (DAWT)”, *Inter. Journal of Integrated Eng.* (*IJIE*), vol. 11, no. 1, May 2019.

[33] A. Sadikin, et al “A Comparative Study of Turbulence Models on Aerodynamics Characteristics of a NACA0012 Airfoil”, *Internation. Journal of Integrated Enginering (IJIE)*, vol. 10, no. 1, Apr. 2018.