American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-10, Issue-01, pp-142-152
www.ajer.org
Research Paper Open Access

Performance of a flanged diffuser augmented wind turbine structure under variable geometrical conditions

Amr M. Abdelrazek
Eng. Mathematics and Physics Dept., Faculty of Engineering, Alexandria University, Egypt

Amira M. Abdelrazik
Mechanical Eng. Dept., Faculty of Engineering, Alexandria University, Egypt

Sadek Z.Kassab
Mechanical Eng. Dept., Faculty of Engineering, Alexandria University, Egypt

ABSTRACT : *In the present paper a study is carried out in order to present and study the variations in the performance of a flanged diffuser augmented wind turbine (FDAWT) structure working under variable geometrical conditions. Nozzle angle, nozzle length, diffuser angle, diffuser length, and flange height are the fife parameters that control the performance of the FDAWT structure. The practical range of each parameter is deduced from the literature review and implemented in the study. Thirty random cases are studied numerically using ANSYS FLUENT6.3. The velocity and pressure distributions and contours of few representative cases are presented. The effect of each parameter on the performance of FDAWT structure is presented and discussed. The maximum and minimum values for the maximum velocity along the structure axis are obtained.*

From the results of the present study the following concluding remarks are deduced:

The velocity ratio of the maximum air velocity along the structure axis, V, and the wind velocity, V_0, V/V_o, reaches a maximum value $V/V_o = 1.83$ and a minimum value $V/V_o = 1.3$. Between these two values there is no observed trend for the results of the 30 random cases. In addition, as the nozzle length increases the mean value of V/V_o is linearly increases regardless of the random variation of the other four parameters. Moreover, there is a slight increase in the velocity ratio, V/V_o, mean value as the nozzle angle increases. Furthermore, the mean trend in the displayed data shows that the mean value of the velocity ratio remains approximately constant regardless of the value of the diffuser length. Also, the mean variation of the velocity ratio, V/V_o, is remained approximately constant regardless the value of the diffuser angle. Finally, the mean value of the velocity ratio, V/V_o, increased slightly as the flange height increases.

Date of Submission: 29-12-2020 Date of acceptance: 10-01-2021

I.INTRODUCTION

Great wind turbines are the main way of producing wind energy with hundreds mega-watt. There are a limited numbers of sites to put these great wind farms where they produce a great amount of energy and this number is reducing over the time. There are less suitable sites, which have a lower average wind speed, but the only way to make it up is to create larger turbines. This is not always possible because there is a limited size for the optimal operation of the surrounding environment. Because of this, off shore wind farms are a suitable solution for the wind turbine designers because the wind turbine size is not a matter to the people as much. The present study focuses on how to make the smaller wind turbine suitable to the urban areas. The main problems that face the urban wind turbines installation are the low wind speeds and the high surface roughness that causes high turbulence intensity. There is a benefit from the urban places. The urban installed wind turbines can take advantage from the buildings to collect and accelerate the flow, which could raise the output power of the wind machine.

Wind turbines in which their rotation of shaft is parallel to the wind is called horizontal axis wind turbine (HAWT). Generally this type of turbine consists of one to three blades. In this type of turbines, the blades are mounted perpendicular to the rotor. HAWT works only with lift principle. Horizontal axis wind turbine (HAWT) has far more efficiency and output than the vertical axis wind turbine, VAWT (in which the rotation of the rotor is perpendicular to the wind direction). That is the reason for their dominance in the wind turbine industry. These turbines are not used in residential area because of its size and noise, Manwell et al. [1].

The power in wind, P, is well known to be proportional to the cubic power of the wind velocity, V_∞ approaching the wind turbine.

$$P = \frac{1}{2}\rho\, V_\infty^3 c_P A$$

Where: ρ is the air density, C_p is the power coefficient, A is the rotor swept area

This means that even a slight increase in wind velocity gives a large increase in energy output. Therefore, many studies tried to find a way to increase the wind velocity. Some of these studies adopt a diffuser-shaped structure surrounding the wind turbine, the other adopt a large flange attached at the exit of diffuser shroud.

The idea of the Diffuser Augmented Wind Turbine (DAWT) is surrounding a wind turbine rotor with a duct. A large flange is attached to the outer periphery of a duct exit. The flange may be considered as an obstacle against the coming flow. However, this flange produces a low pressure region to create separation behind it to collect and accelerate more wind and increase the wind velocity inside the structure compared to a diffuser with no flange. The Flanged Diffuser Augmented Wind Turbine FDAWT presents a significant advancement in the reduction of cost of energy for the augmented wind turbine concept.

Foreman and Gilbert [2-4] and Foreman [5] carried out considerable amount of wind tunnel experiments. They have concluded that the Diffuser Augmented Wind Turbine, DAWT, can produce 4.25 times the power of the same wind turbine that is operating as a bare wind turbine.

De Vries [6] has a first trial to improve theory for DAWT. He characterizes simple diffuser theory from shrouded turbine theory. Foreman and Gilbert [3] conducted three groups of tests on compact diffuser on great scale models, on real turbine and on small-scale turbine using screens instead of real turbine. The first test on non-optimized DAWT produced three times power of bare wind turbine with the same diameter and the optimized diffuser segmented wind turbine produced more than four times power augmentation.

Badawy and Aly [7] obtained a theoretical demonstration of DAWT. There results showed that augmentation ratio and maximum power coefficient are directly proportional to pressure recovery factor, turbine factor, and maximum velocity ratio but inversely proportional to overall recovery factor of diffuser. Ohya et al. [8] showed that the flange plays a major role in accelerating the approaching wind velocity and it also replaces the need of yaw control mechanism. Ohya and Karasudani [9] developed a new wind turbine system which consists of a wind turbine surrounded by a diffuser shroud and a broad-ring brim to it. It demonstrated power augmentation by a factor of two to five when compared with a standard turbine. Sarwar et al. [10] developed a new diffuser augmented wind turbine system with a flange at the exit periphery. They reported that wind speed in the diffuser was greatly affected by the diffuser open angle, flange height, center body length, inlet shape, and hub ratio.

Kannan et al. [11] investigated the effect of wind speed on various shapes of flanged diffuser. Solidworks and ANSYS fluent were used to model and simulate the design. The inlet velocity was increased by 61.25% in there diffuser. Jafari and kosasih [12] pointed out that, CFD simulations have been carried out on a small commercial wind turbine with a simple frustum diffuser shrouding. Numbers of diffusers were modeled based on length to area ratio and height of the frustum to area ratio. The results showed that these parameters play a prominent role in sub-atmospheric back pressure and flow separation in the diffuser. Corrêa and Guama [13] proposed a mathematical model to study the velocity behavior for three conical diffusers taking into consideration the conditions of the flow around them. The results were compared with experimental results for the three diffusers, and show good agreement. Rochman and Nasution [14] presented CFD studies to get the role of adding a flange to the diffuser exit to the wind velocity inside the diffuser. The results show that the using of a flat flange increased the wind velocity more than 29%. The velocity increase more than 31% in the case of installing the airfoil shaped flange.

Yavuz [15] aimed to optimize the merging of flap wind turbine with a concentrator to achieve high flow speed at the turbine region in the concentrator to increase the output power production rate of the wind turbine. The results got compared with bare wind turbine and airfoil type concentrator. An actuator disc model is used to characterize wind turbine in the concentrator. The results showed that in the case of using concentrator with flap, the flow velocity factored by 1.2. The inner flow of three hollow structures, a nozzle-type, a cylindrical-type model and a diffuser-type was examined by Ohya et al. [8]. The results show that the diffuser structure has the best effect on collecting and accelerating the wind.

As a result of many trials, it is concluded that the velocity is increased when adding an inlet nozzle and a ring type flange to the diffuser exit. The inlet shroud is a curved surface surrounding the diffuser model inlet. The flange is added vertically to the diffuser model exit. As shown in Figs. 1 and 2, by adding the inlet shroud and the flange, a great raise in the wind velocity can be obtained, better than the case of diffuser model only, Ohya et al. [8] and Abe et al. [16].

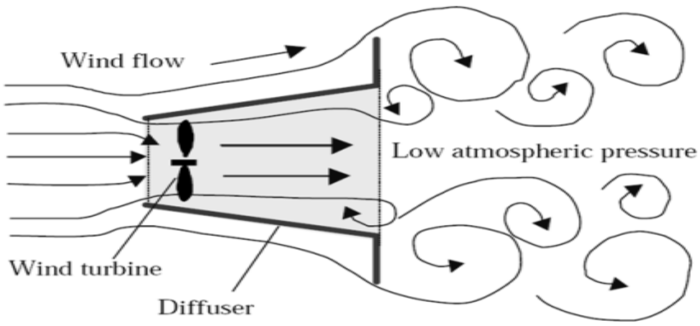


Fig.1. Schematic cross sectional view of a diffuser and wind speed increase mechanism, Abe et al [16].

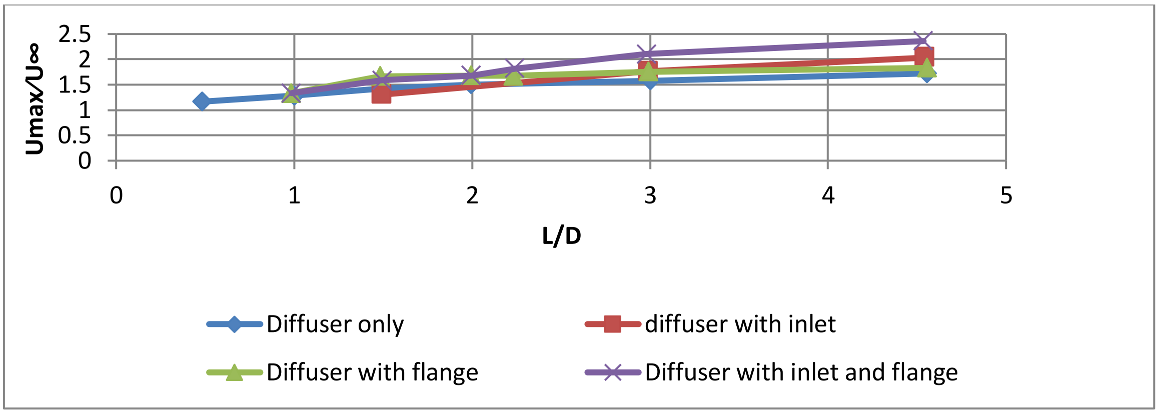


Fig. 2. Increase in the maximum wind velocity by the addition of an element to the diffuser-type model, Ohya et al. [8]

The using of a flanged diffuser produces a great increase in the output power coefficient C_w nearly 5 times that of a standard wind turbine is reached, Ohya et al. [8]. (C_w is the ratio between the actual electrical power produced and the wind power into turbine, $C_w = \frac{P_{out}}{P_{in}}$).

The main objective of the present study is to study and present the variations in the performance of a flanged diffuser augmented wind turbine (FDAWT) structure working under variable geometrical conditions. Nozzle angle, nozzle length, diffuser angle, diffuser length, and flange height are the fife parameters that control the performance of the FDAWT structure as well as its efficiency. The practical range of each parameter is deduced from the literature review and implemented in the present study. Thirty random cases, within the chosen ranges, are studied numerically using ANSYS FLUENT 6.3. The velocity and pressure distributions and contours of few representative cases are presented. The effect of each parameter of the fife geometrical parameter on the performance of FDAWT structure is presented and discussed. The maximum and minimum values for the maximum velocity along the structure axis is obtained and compared for all the studied cases. The matching between the velocity and pressure results within the studied domain is also explored.

II.NUMERICAL PROCEDURE

Many researchers have developed numerical codes to support aerodynamic optimization to improve the energy production of wind turbine, (Lanzafane et al. [17], Sumner et al. [18], and Sargsyan et al. [19]). The flow around a wind turbine is governed by the main principles of Navier-Stokes equations. Unfortunately, these equations are so complex that analytical solutions only have been found for simple cases. Numerical techniques can used to solve these complex equations, Jonkman [20]. CFD models are based on the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations derived from the main principles of conservation of mass and momentum, Sumner et al. [18].

In ANSYS FLUENT core there are two available numerical methods, applied for several conditions. The first solver is pressure-based solver which mainly used for low speed incompressible flows although, the second solver designed as density-based solver, were created for application in high-speed flows. In the present study, includes incompressible flow, pressure-based approach was applied.

Two-equation models are the most turbulent models used in the CFD history. The main features of k-ε models are robustness, the economic submitted in computer terms, and the behavior presented for wide range of turbulent flows generating a reasonable accuracy, ANSYS FLUENT 14.0 [21]. The standard form of the k-ε turbulence model is originally proposed by Launder and Spalding [22]. The turbulent kinetic energy, k, and it is dissipation rate, ε, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial k}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}=\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon+S_k\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)$$
$$=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

In these equations, G_k, shows the production of turbulence kinetic energy due to the average velocity slopes, and is described in Fluent. G_b signs to the turbulence kinetic energy generation due to buoyancy. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants, also σ_k and σ_ε are turbulent Prandtl numbers for k and ε. S_k and S_ε are user-defined sources terms. Thereby, this approximation estimation to the turbulence inputs of k- ε model is used in the present work.

Figure 3 shows the boundary conditions and outer domain for the flanged diffuser structure, where the Inlet boundary (on the left hand side border) is set as a velocity Inlet, the outlet boundary (on the right hand side border) is set as pressure outlet, and the top boundary is set as an open stream while the bottom boundary is set as an axis. The model dimensions have been drawn in reference to the throat diameter D of the flanged diffuser.

In order to ensure the solution convergence, the resulting solution must satisfy the following conditions:
Residual RMS values have been reduced to an acceptable value (typically 10^{-4}). Monitor values of the fields of interest (such as drag, lift and momentum residuals) have to reach a steady solution. These conditions are satisfied in the present study and the details are given in Abdelrazik [23].

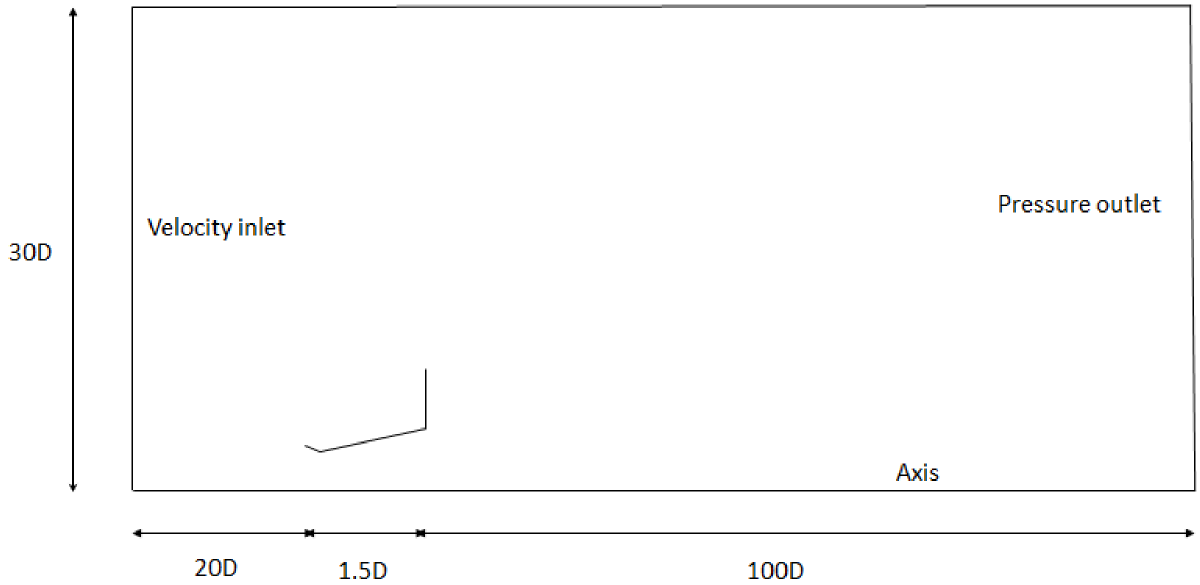


Fig. 3. Computational domain and boundary conditions for the flanged diffuser structure

In the present study, In order to ensure the mesh independency of the solution, four different grids are tested to prove that refining and increasing the number of nodes and elements will not have a big effect on the results accuracy. Grid independence study is performed by choosing one point inside the diffuser and tracking the pressure value for the same point in the four different grids. Each grid has number of nodes as shown in Table 1.

Grid NO.	Number of nodes
1	41,894
2	64,234
3	94,094

4	123,694

Table 1. The number of nodes for the four different grids

As shown in Fig. 4 the pressure value begins (-18.664 pa) then it decreases to about (-21.717pa) and this value can be considered constant from the second to the fourth grid. The third mesh in the grid independence study is used for the flanged diffuser cases, (94,094) nodes are used for the present study to decrease the iterating time.

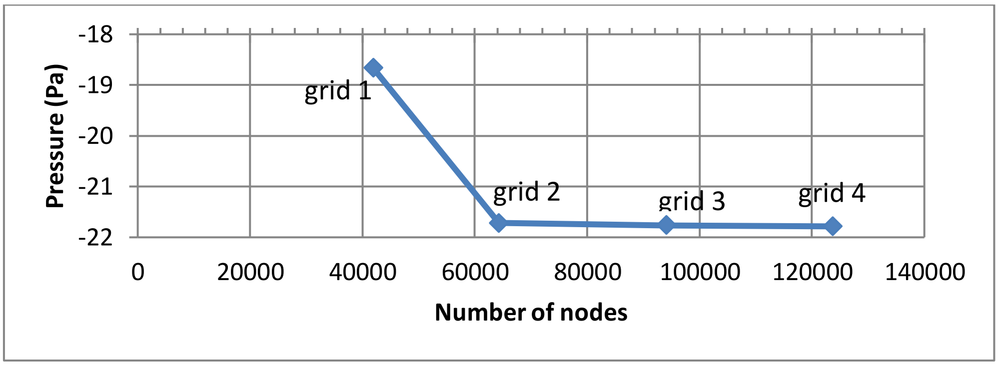


Fig. 4. Grid independence curve

A comparison has carried out between experiments that have been performed in order to find out the maximum velocity that could be generated from different geometric shapes such as the nozzle, cylinder and diffuser, Abe et al. [16], and the results obtained in the present study through the Fluent ANSYS software in order to ensure the validity of the criteria used in the present study as shown in Fig. 5. This figure shows that the results from generated simulations are in agreement with the reference ones. Accordingly, the generated model is valid and could be safely developed and optimized.

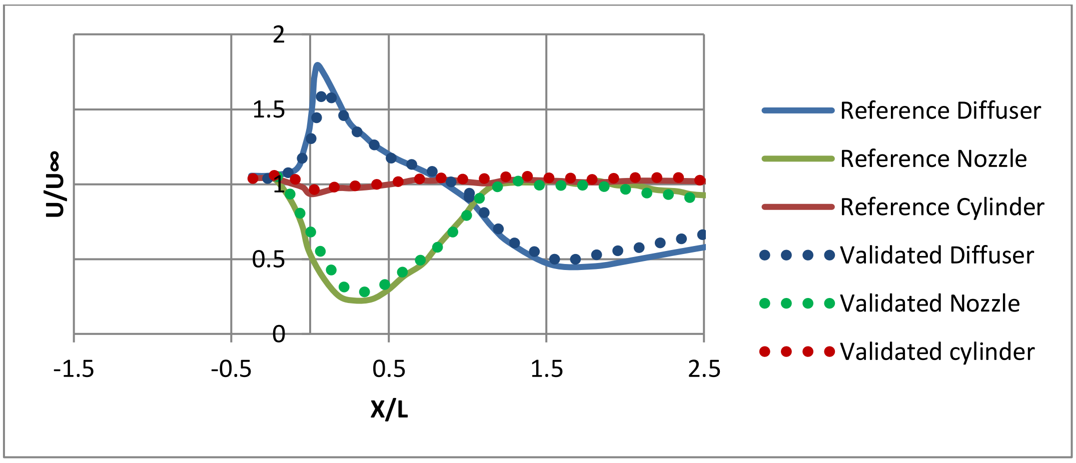


Fig. 5. Validation for performed simulation for the wind velocity on the central axis of three different geometric structures i.e. diffuser, nozzle and cylindrical ducts, present study and Abe et al. [16].

III.RESULTS AND DISCUSSION

The results and discussion related to the performance of a flanged diffuser augmented wind turbine structure under variable geometrical conditions are presented and discussed. Nozzle angle, nozzle length, diffuser angle, diffuser length and flange height, are the parameters that control the performance of the flanged diffuser augmented wind turbine (FDAWT).

According to revising many published reviews (Abdulaziz [24], Putra and Rifai [25], and Barbosa and Vaz [26]), the values of the five factors used in the present study are shown in Table 2:

Nozzle length, L_N	L_N /D = 0.15 - 0.2 - 0.25 - 0.3 - 0.35
Nozzle angle	10 º - 12 º -16º - 20 º - 22º - 24º - 26º - 28º - 30 º
Diffuser length, L_D	L_D / D = 0.63 - 1.05 - 1.15 - 1.25 - 1.35 - 1.45 - 1.76
Diffuser angle	4 º - 6 º - 8 º - 10 º - 12 º - 14 º -16º - 18º - 20 º
Flange height, h	h /D = 0.8 - 0.9 - 1 - 1.1 - 1.25

Table 2. The ranges of the independent variables

Where D is the diffuser inlet diameter.

It is important to point out that running the solution on ANSYS FLUENT6.3 with flange height range from zero to o.7D gave reversed flow or high residuals. Therefore, the cases got solved with flange height range (h/D = 0.8. 0.9, 1, 1.1, 1.25) as presented in Table 2.

Trials are carried out in a successive form, this form is applied by varying the five parameters dimensions in a dimensionless form, where all the selected values were taken as a factor of the diffuser's throat diameter (D) except the angles. Each trial is studied carefully, in order to find out the best case in each attempt. The best case is selected as per the highest wind speed inside the augmented wind turbine structure at or around the throat, where the wind rotor is placed. Thirty random cases are introduced to get the best velocity by using ANSYS-FLUENT 6.3. They are presented in Table 3.

case	diffuser angle	nozzle angle	diffuser length	nozzle length	flange height	velocity at throat (m/s)	V_{max}/V_o
1	14	12	1.15D	0.15D	1.25D	6.58	1.31
2	4	10	1.76D	0.35D	0.8D	7.87	1.57
3	8	22	1.45D	0.25D	1D	8.16	1.63
4	6	16	1.05D	0.2D	0.8D	7.06	1.41
5	20	16	1.15D	0.3D	1.1D	7.39	1.47
6	8	26	1.05D	0.2D	1D	7.87	1.57
7	20	26	1.25D	0.2D	0.9D	7.44	1.48
8	14	30	1.45D	0.25D	1.25D	8.56	1.7
9	6	28	0.63D	0.25D	0.9D	9.15	1.83
10	10	24	0.63D	0.15D	0.8D	6.66	1.33
11	12	12	1.25D	0.25D	1.1D	7.64	1.52
12	16	12	1.35D	0.3D	1D	7.51	1.5
13	18	24	1.05D	0.3D	0.8D	8.18	1.6
14	20	30	1.45D	0.35D	0.9D	9.16	1.83
15	16	20	0.63D	0.25D	0.8D	6.98	1.39
16	18	10	1.45D	0.2D	1D	6.99	1.39
17	12	28	1.05D	0.3D	1.1D	8.55	1.71

18	8	26	1.35D	0.15D	1.25D	7.82	1.56
19	10	22	1.25D	0.2D	1.25D	7.93	1.58
20	20	12	0.63D	0.35D	1D	7.37	1.47
21	12	26	0.63D	0.15D	0.8D	6.75	1.35
22	18	30	1.76D	0.3D	1.1D	8.49	1.69
23	16	10	1.25D	0.25D	0.9D	7.43	1.4
24	6	12	1.15D	0.35D	1D	7.59	1.51
25	4	22	1.45D	0.15D	1.25D	7.36	1.47
26	14	16	0.63D	0.3D	0.8D	7.53	1.51
27	10	24	1.05D	0.2D	0.9D	7.83	1.56
28	8	20	1.76D	0.25D	1.1D	8.58	1.71
29	6	28	1.35D	0.35D	1D	8.32	1.66
30	10	30	1.25D	0.15D	1.25D	7.91	1.58

Table 3. Random cases of DAWT on FLUENT

Figures (6-11) show the velocity and pressure contours and distributions inside and outside the flanged diffuser structure in the selected domain for few representative cases. It is unrealistic to present all the thirty cases in one paper (especially because each figure contains four graphs). Therefore few cases will be selected to represent the thirty cases. The results of these selected cases are shown in Figs. (6-11). The results related to the velocity distribution in these figures show that the velocity starts to increase at the structure inlet and reached its maximum value at the diffuser throat and decreased gradually at the structure outlet. Meanwhile, the results related to the pressure distribution show that the pressure begins to decrease at the structure inlet and reaches its lowest value at the diffuser throat and increased gradually at the structure outlet to return back to the atmospheric pressure. These trends, for the velocity and pressure, satisfy and match the energy and Bernoulli's equations.

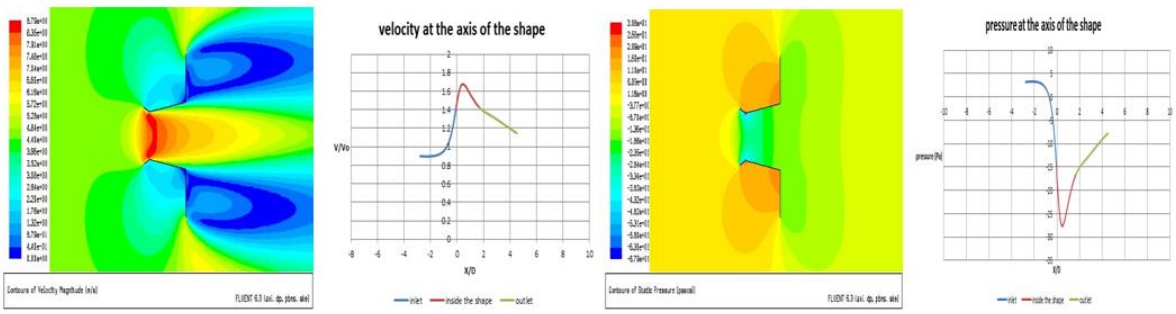


Fig. 6. Velocity and pressure contours and distributions along the axis of the diffuser for case 3

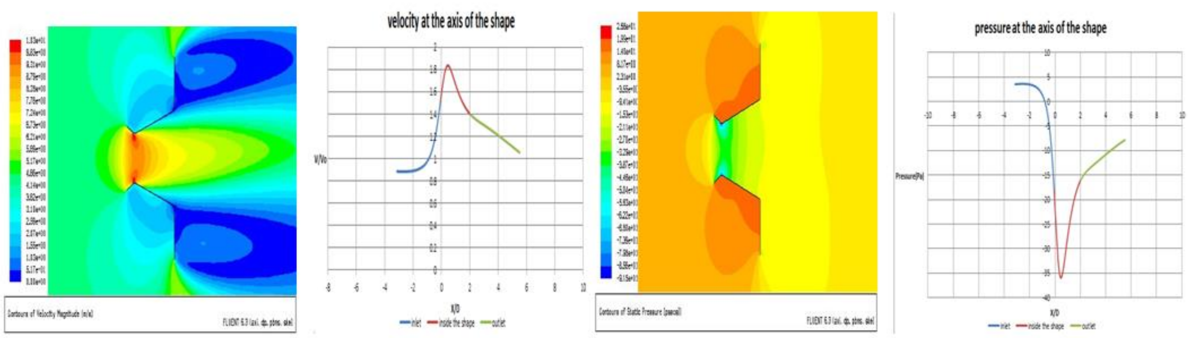


Fig. 7. Velocity and pressure contours and distributions along the axis of the diffuser for case 9

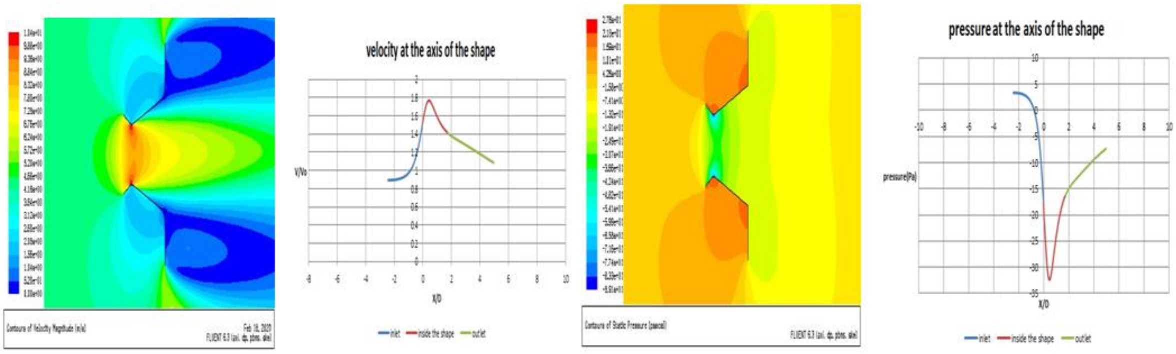


Fig. 8. Velocity and pressure contours and distributions along the axis of the diffuser for case 14

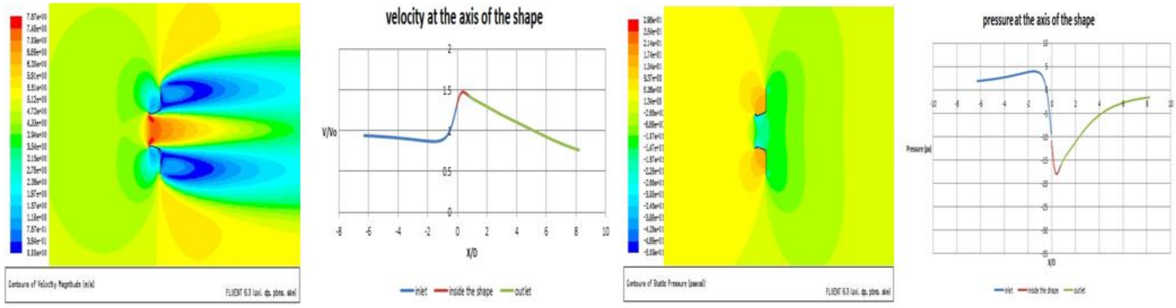


Fig. 9. Velocity and pressure contours and distributions along the axis of the diffuser for case 21

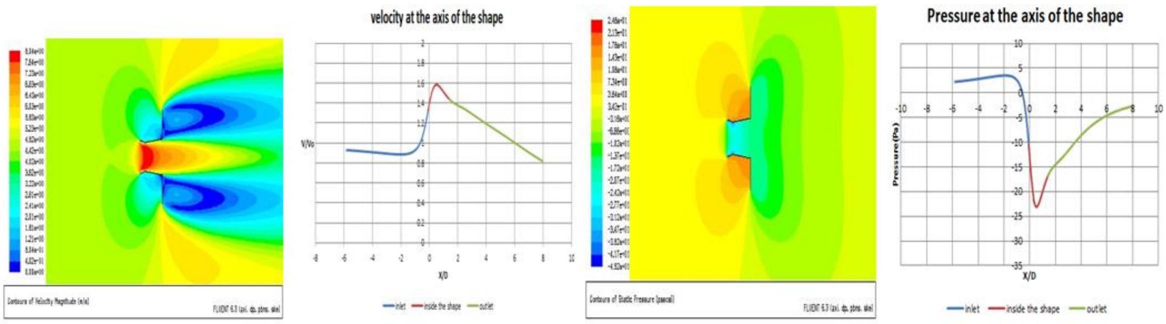


Fig. 10. Velocity and pressure contours and distributions along the axis of the diffuser for case 24

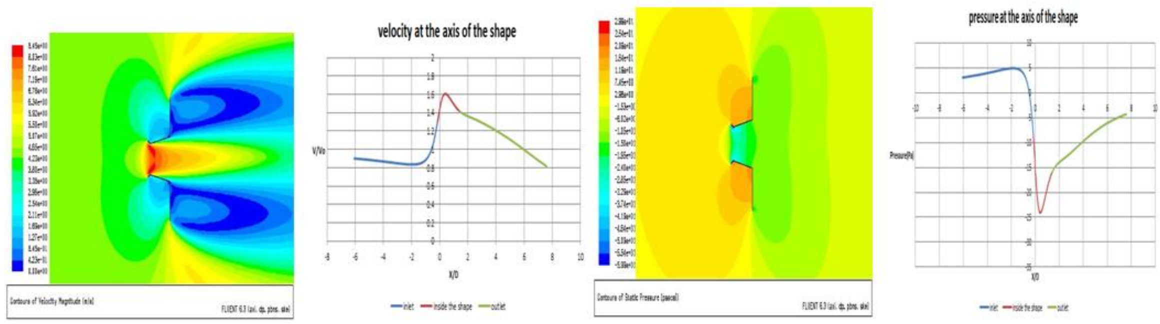
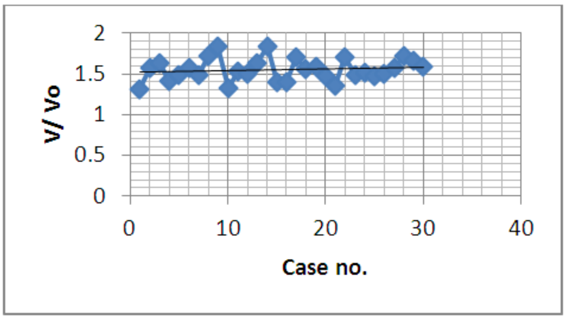


Fig. 11. Velocity and pressure contours and distributions along the axis of the diffuser for case 30

As shown in Fig. 12 the velocity ratio changes for the different cases to reach the maximum value $V/V_o = 1.83$ and the minimum value $V/V_o = 1.3$. There is no observed trend for the results of the 30 random cases.

Figure 13 shows that, the nozzle length changes from 0.15D to 0.35D, the velocity ratio reaches its maximum value V/V_o = 1.83 at $L_{nozzle} = 0.25D, 0.35D$ and its minimum value V/V_o = 1.3 at ($L_{nozzle} = 0.15D$) with velocity ratio increase 53%. It is important to notice that Fig. 13 reveals that as the nozzle length increases the mean value of V/V_o slightly increased linearly regardless of the random variations of the other four parameters.



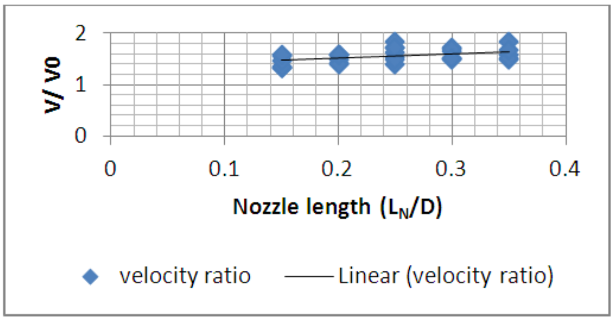
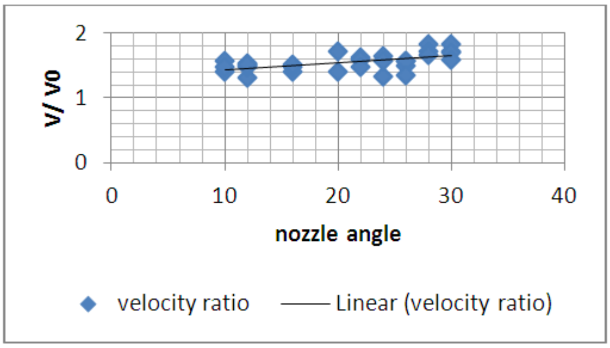


Fig. 12. Velocity ratio changes with the thirty cases **Fig. 13. Velocity ratio changes with the nozzle length**

Meanwhile, Fig. 14 shows that as the nozzle angle changes from 10° to 30°, the velocity ratio reaches its maximum value V/V_o = 1.83 at $nozzle\ angle = 30°$ and its minimum value V/V_o = 1.3 at $nozzle\ angle = 12°, 24°, 26°$. This figure shows also a slight linear increase in the velocity ratio value as the nozzle angle increases despite the fact that the change in the other parameters is random.

Considering the diffuser length, Fig. 15 shows that as the diffuser length changes from 0.63D to 1.76D, the velocity ratio reaches its maximum value V/V_o = 1.83 at $L_{diffuser}$ = 0.63D ,1.45D and its minimum value V/V_o = 1.3 at $L_{diffuser} = 0.15D$. The mean trend in the displayed data shows that the mean value of the velocity ratio remains approximately constant regardless the value of the diffuser length.



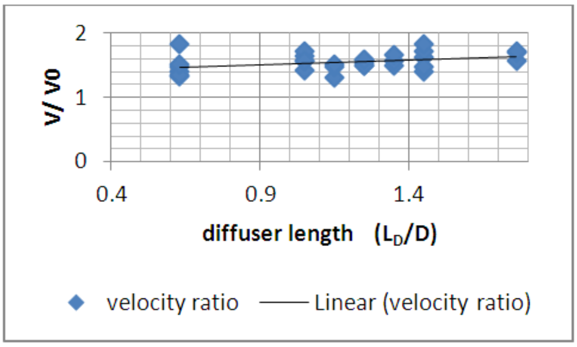


Fig. 14. Velocity ratio changes with the nozzle angle **Fig. 15. Velocity ratio changes with the diffuser length**

Furthermore, Fig. 16 shows that the diffuser angle changes from 4° to 20°, the velocity ratio reaches its maximum value V/V_o = 1.83 at $diffuser\ angle = 6°, 20°$ and its minimum value V/V_o = 1.3 at $diffuser\ angle = 10°, 14°$. This figure also shows that the mean value of the velocity ratio remains approximately constant regardless of the value of the diffuser angle.

Finally, considering the flange height, Fig. 17 shows that the flange height changes from 0.8D to 1.25D, the velocity ratio reaches its maximum value V/V_o = 1.83 at flange height $= 0.9D$ and its minimum value V_{max}/V_o = 1.3 at flange height $= 0.8D$, 1.25D. The mean trend of Fig. 17 is that the mean variation of V/V_o is linearly increased slightly as the flange height increases.

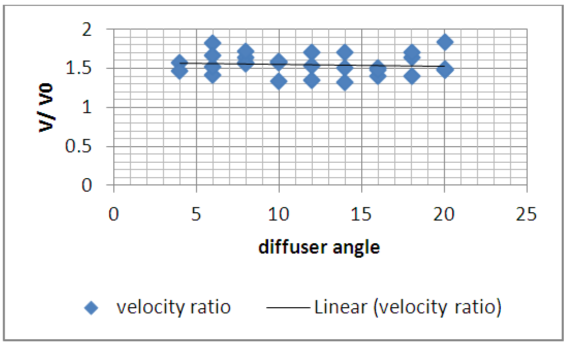


Fig. 16. Velocity ratio changes with the diffuser angle

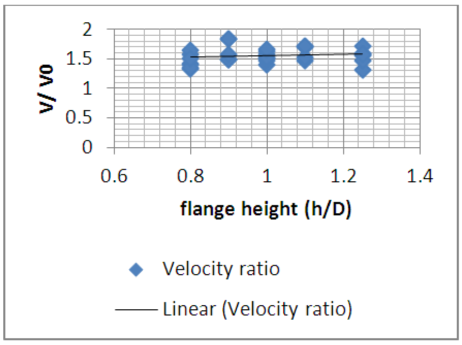


Fig. 17. Velocity ratio changes with the flange height

One should note that each point in Figs. 12-17 and Table 3 represents different values for the 5 used parameters. This means that the presented results in these figures show the mean trend of each parameter when it used with the other 4 parameters in random order.

IV.CONCULUSION

In the present study, a structure consisting of a nozzle, a diffuser and a flange has been constructed and modeled using numerical methods. The aim was to study the variations in the performance of a flanged diffuser augmented wind turbine (FDAWT) structure working under variable geometrical conditions. Nozzle angle, nozzle length, diffuser angle, diffuser length, and flange height are the parameters that control the performance of the FDAWT structure. Thirty random cases are studied numerically using ANSYS FLUENT6.3. The effect of each parameter on the performance of FDAWT structure is presented and discussed.

Within the range of the variations of the geometrical parameters studied in the present study, the following concluding remarks are deduced:

- The velocity ratio of the maximum air velocity along the structure axis, V, and the wind velocity, V0, V/Vo, reaches a maximum value V/Vo = 1.83 and a minimum value V/Vo = 1.3. Between these two values there is no observed trend for the results of the 30 random cases.

- As the nozzle length increases the mean value of V/Vo is linearly increased regardless of the random variations of the other four parameters.
- There is a slight increase of the velocity ratio, V/Vo, mean value as the nozzle angle increases despite the fact that the change in the other parameters is random.

- The mean trend in the displayed data shows that the mean value of the velocity ratio, V/Vo, remains approximately constant regardless of the value of the diffuser length.

- The mean variation of the velocity ratio, V/Vo, is remained approximately constant regardless the value of the diffuser angle.

- The mean value of the velocity ratio, V/Vo, is increased slightly as the flange height increases.

REFERENCES

[1]. Manwell. F., McGowan., G. and Rogers L., “Wind Energy Explained; Theory, Design and Application”, John Wiley & Sons Ltd, 2002.
[2]. Foreman. M. and Gilbert. L., “Fluid Dynamics of Diffuser-Augmented Wind Turbines”, Journal of Energy, Vol. 2, No. 6, pp. 368-374, 1978.
[3]. Foreman. M. and Gilbert L., “Experimental Demonstration of the Diffuser-Augmented Wind Turbine Concept”, Journal of Energy, Vol. 3, No. 4, pp. 235-240, 1979.
[4]. Foreman. M. and Gilbert L., “Experiments with a Diffuser Augmented Wind Turbine”, Journal of Energy Resources, vol. 1, pp. 46-54, 1983.
[5]. Foreman. M., “Preliminary Design and Economic Investigations of Diffuser Augmented Wind Turbines (DAWT)”, Research Department Grumman Aerospace, New York, 1981
[6]. De Vries. O., “Fluid Dynamic Aspects of Wind Energy Conversion”, AGARDograph No. 243, 1979.
[7]. Badawy. M. and Aly. M., “Gas dynamic analysis of the performance of diffuser augmented”, Sadhana, pp. 453–461, 2000.

[8]. Ohya Y., Karasudani. T. and Sakurai. A., "Development of a Shrouded Wind Turbine with a Flanged Diffuser", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96, pp. 524-539, 2008.
[9]. Ohya Y. and Karasudani. T., "A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology", Energies, Vol. 3, pp. 634-649, 2010.
[10]. Sarwar. M., Nawshin. N. and Imam. M., "A New Approach to Improve the Performance of an Existing Small Wind Turbine by using Diffuser", International Journal of Engineering & Applied Science, Vol. 4, pp. 31- 42, 2012.
[11]. Kannan. S., Mutasher. S. and Kenny. L., "Design and Flow Velocity Simulation of Diffuser Augmented Wind Turbine using CFD", Journal of Engineering Science and Technology, Vol. 4, pp. 372-384, 2013.
[12]. Jafari. S. and Kosasih. B., "Flow Analysis of Shrouded Small Wind Turbine with a Simple Frustum Diffuser with Computational Fluid Dynamics Simulations", Journal of Wind Energy, pp.102-110, 2014.
[13]. Corrêa. R. and Guamá., "An Investigation of a Mathematical Model for the Internal Velocity Profile of Conical Diffusers Applied to DAWTs", Annals of the Brazilian Academy of Sciences , pp.1133-1148, 2015.
[14]. Rochman. M. and Nasution. A., "CFD Studies on the Flanged Diffuser Augmented Wind Turbine with Optimized Curvature Wall", Springer Science & Business Media Singapore, pp.347-355, 2017.
[15]. Yavuz. T., "Effect of Flap on the Wind Turbine-Concentrator Combination", international journal of renewable energy research, pp. 551-560, 2019.
[16]. Abe. K., Nishida M., Sakurai. A. and Ohya. Y., "Experimental and Numerical Investigations of Flow Fields behind a Small Wind Turbine with a Flanged Diffuser", Journal of Wind Engineering and Industrial Aerodynamics, Vol.93, pp. 951-970, 2005.
[17]. Lanzafame. R., Mauro, S. and Messina, M., "Wind turbine CFD modeling using a correlation-based transitional model", Renewable Energy, pp. 31 – 39, 2013.
[18]. Sumner. J., Watters. and S. and Masson. C., "CFD in wind energy: The virtual, multiscale wind tunnel", Energies, pp. 989 – 1013, 2010.
[19]. Sargsyan, A., "Simulation and modeling of flow field around a horizontal axis wind turbine (HAWT) using RANS method", Master's thesis, Florida Atlantic University, 2010.
[20]. Jonkman, J. M. "Modeling of the UAE wind turbine for refinement FAST-AD, Technical Report", National Renewable Energy Laboratory, 2003.
[21]. Fluent (2011a), ANSYS FLUENT 14.0 Documentation, ANSYS, Inc.
[22]. Launder. E. and Spalding. B., "Lectures in Mathematical Models of Turbulence", Academic Press, London, England, 1972.
[23]. Abdelrazik, A. M., "Optimization of a diffuser augmented wind turbine structure performance using design of experiments technique", M. Sc. Thesis, Alexandria University, Egypt, 2021.
[24]. Putra. F. and Rifai. D, "Multilevel Diffuser Augmented for Horizontal Axis Wind Turbine", Web of Conferences 42, pp. 1-6, 2018.
[25]. Abdulaziz. A., "Study of the Performance of a Flanged Diffuser Augmented Wind Turbine", M. Sc. Thesis, Alexandria University, Egypt, 2017.
[26]. Barbosa. D. and Vaz. J., "An Investigation of a Mathematical Model for the Internal Velocity Profile of Conical Diffusers Applied to DAWTs", Research gate, pp. 1-17, 2015.