International Journal of Academic Scientific Research
ISSN: 2272-6446 Volume 8, Issue 4 (November - December2020), PP 15- 23
www.ijasrjournal.org

A numerical study of the effect of turbulence generators in wind turbines diffusers

Mohammad Youssef Al Hashem[1], Nidal Al deeb[2]

[1, 2] *Department of Power Engineering, Faculty of Mechanical Engineering, University of Aleppo*

Abstract: *This research aims to determine how to collect wind energy effectively in order to increase the speed of winds that approach the wind turbine. The generation of energy using wind is proportional to the wind speed cubed. If the dynamic nature of the fluid around a turbine structure and topography were used, the wind speed would be increased. Also, focusing wind energy in the blade area will increase turbine energy output. This study relates to the use of diffusers on wind turbines to improve the wind power system and increase the payoff, and aims to determine which form of diffusers are able to collect wind energy effectively and generate energy with high efficiency from wind. A numerical study was performed on the diffuser's influence on wind turbines using computational fluid dynamics (CFD). A comparison between the performance of the proposed diffuser equipped with ducts and a reference study of different diffusers. The results showed that the proposed diffuser gave an increase in energy output around 21% greater than all the diffusers in the reference study.*

Keywords: *diffuser augmented wind turbine (DAWT), wind turbines.*

1. INTRODUCTION

1.1 Background

The development and application of clean and renewable energy has become an important issue in recent years due to the serious effects of combustion products from power plants and the rapid depletion of fossil fuels. Therefore, in order to address the current energy crisis, work is underway on various alternative energy methods. Wind energy technologies have become one of the fastest growing energy sources in the world, as they are almost an endless resource. However, compared to overall energy demand, the amount of wind energy use is still very meager. This is due to various reasons such as (cost - wind speed - large volumes of turbines ... etc.). Most of the wind energy comes from large wind farms that contain hundreds of wind machines. As with the limited supply of fossil fuels, there is a limited amount of sites where huge wind farms are grown so that they produce a large amount of energy and this number is depleted with each passing year. There are fewer desirable sites, which have lower average wind speeds, but the only way to get a return on investment is to make the turbines larger to compensate for the lower speed. This is not always possible since modern ground-based turbines are about the size limits for optimum operation and keep the effect on the population and surrounding environment to a minimum. This is why this research has focused on taking the opposite approach and making wind turbines smaller to make them available for installation in the urban environments, which increases workable wind energy sites. The main challenge for urban wind turbine installations is low average wind speed and high turbulence due to increased surface roughness for incoming winds. There is the possibility to take advantage of the urban environment so that wind turbines installed in urban areas can benefit from the accelerated flow around buildings and obstacles, which may increase the power output of the wind machine.

Grady and others [1] (2012) conducted a numerical and analytical study on diffuser wind turbines. Their research involves the addition of a diffuser to encapsulate the "nude turbine" and create pressure drop at the trailing edge of the wind turbine. They found that the addition of this specific diffuser increases the air mass flow rate and the efficiency of the wind turbine. Nasir Mahmoud and others [2] (2012) conducted a study to investigate the effect of length and angle of the diffuser section NACA0018 on diffuser design. Simulation was performed using CFD. It was found that the speed inside the diffuser increases with the increase of the diffuser's length and angle of attack. Also, increasing the diffuser angle higher than the critical angle leads to the collapse of the diffuser due to the flow separation.

Saravana and others [3] (2013) conducted a numerical study of four different diffuser design concepts using a CFD. It was found that a diffuser with a large flange is able to greatly increase the speed of incoming winds. The lipped diffuser and the diffuser with an internal cone divided into fluxes have a great role in collecting and accelerating incoming winds. Léo Daiki and others [4] (2013) carried out a numerical study of the runoff in diffusers of various geometric shapes using (CFD) for application to (DAWT). A simulation was performed with the intention of identifying the geometry of the diffuser with a good hydrodynamic efficiency coefficient. Masoud [5] (2016) Carried out a design and analysis study using CFD for the wind turbines used to propel small boats. This study focused on designing a portable wind turbine, capable of generating 70 kW to drive a recreational boat with a capacity of 8-10 passengers. The analysis was performed using two circumferential speeds. First: At the inlet velocity of 12 m / s, the air velocity at the turbine is 16 m / s and the power generated by the turbine is 61 kw. Second: At the inlet speed of 6 m / s, the air velocity at the turbine is 10 m / s and the power generated by the turbine is 25 kw. kesby et al. [6] (2016) conducted a study to determine the performance of a diffuser wind turbine using the combination of the CFD & BEM method. The proposed method harnesses both CFD accuracy and BEM speed to predict energy production from diffuser wind turbines with an efficient calculation. The proposed method requires data showing the relationship between velocity at the level of the blade and the axial velocity at the exit of the diffuser to determine the effective diffuser exit zone. A good agreement has been reached between predictions from this new method and experimental data. Prakash and others [7] (2017) conducted a study to improve the performance of publisher's wind turbines using CFD. The diffuser increases the power output of wind turbines primarily by increasing the flow velocity through the blades due to the flow control through the turbine, which makes the pressure at the outlet much lower than the atmospheric pressure, and secondly, limiting the limb phenomenon at the blade's tips. The main objective of this project is to fully assess the effect of the diffuser shape on the angular and instantaneous speed on performance. This effect is observed when installing the turbine inside the diffuser.

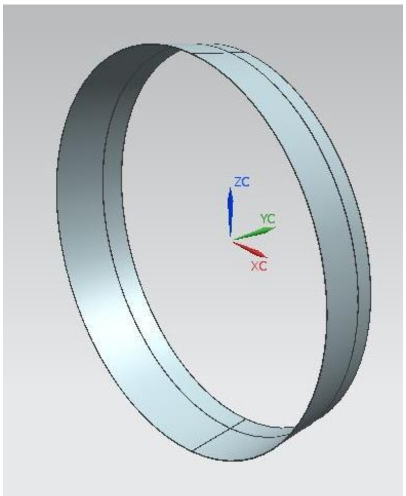
1.2 Purpose

Most regions in Syria have low wind speeds and traditional wind turbines cannot be used, so that the payoff is low in proportion to the cost, so wind turbines had to be equipped with a diffuser to get the highest payoff and at low wind speeds and at the lowest cost. Since the wind power is proportional to the wind speed cubed. The importance of the research is that the output can be increased by approximately 3 times when the diffuser is present. Moreover, the diffuser positioned around the blade rotation area (sound source) achieves excellent silence as it suppresses the blade tip vortex which is the source of noise. Plus a visual sense of safety, since the diffuser is a blade surround collector.

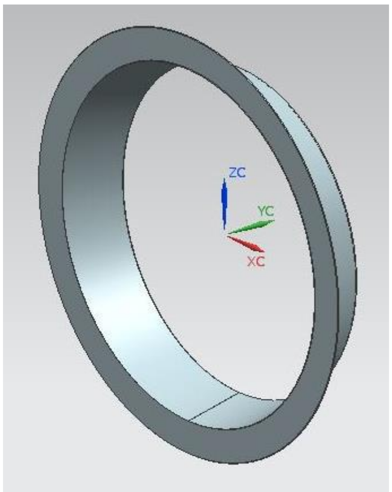
2. Method and procedure

2.1. Summary of the reference study

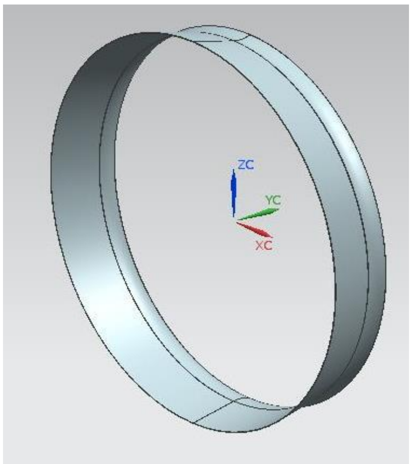
Mr. Gade Sagar Tukaram and others [8] had studied a multi-shaped diffuser that is an accelerator that surrounds a wind turbine, known as a wind lens. Numerical research deals with the effect of the low pressure region caused by the wind lens and thus analyze the strong eddies formed by the edge attached to the diffuser at the outlet. It has been shown that there is a significant increase in velocity when the edging effect is added to the diffuser. The basic 3D model was drawn on CREO software. Straight diffuser, straight diffuser with edge, curved diffuser, curved diffuser with edge, gradient diffuser, and gradient diffuser with edge. 3D models from various diffusers for CFD analysis were used in ANSYS FLUENT version 14.5. The results of this reference study was that when comparing the diffuser without rim and diffuser with rim, it would appear that diffuser with rim will produce more eddy formation without edge, diffuser with turbulence generators with edge will produce 5 times more power output than bare wind turbines, diffuser with turbulence generators without a ledge produces 3 times more power output than bare wind turbines and shows higher power output than all other diffusers without a ledge. The study concluded that the diffuser with turbulence generators is more efficient than all other diffusers.Fig.1 shows the considered diffusers in this research.



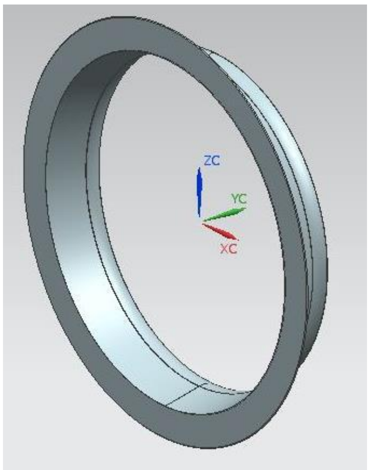
Straight diffuser without brim



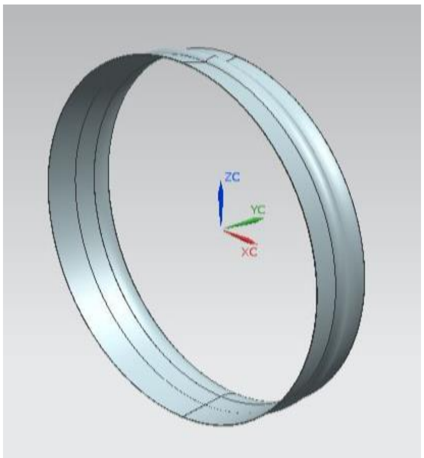
Straight diffuser with brim



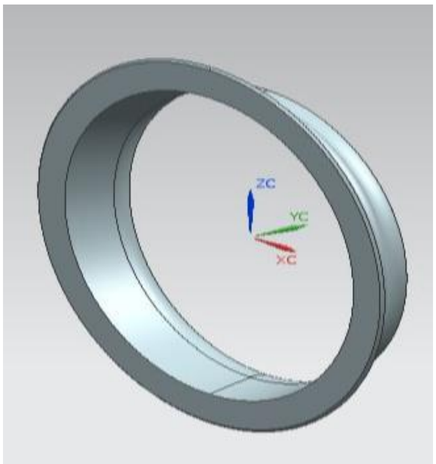
Curved diffuser without brim



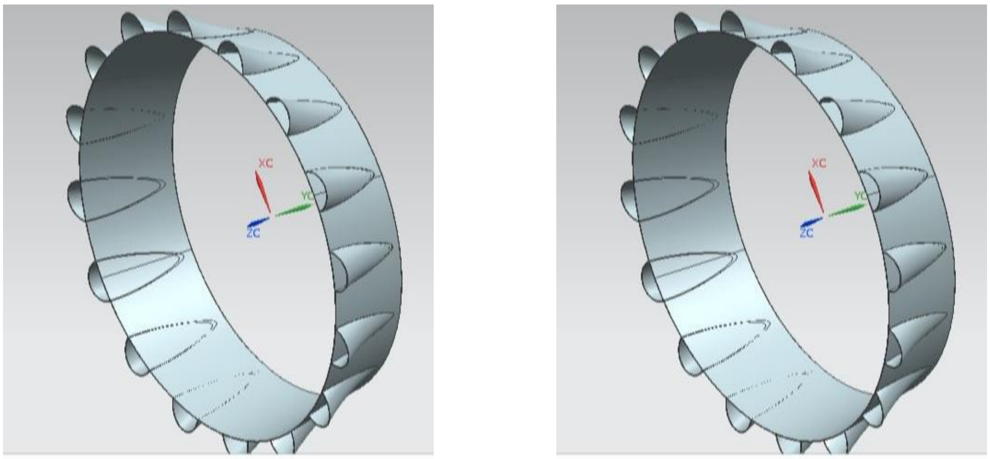
Curved diffuser with brim



stepped diffuser without brim



stepped diffuser with brim



Bumped diffuser without brim

Bumped diffuser with brim

Fig.1 Shapes studied in this research.

2.2. Considered model

The basis for the design of the turbine as a whole is based on patent No. 6877960 registered in the name of Walter M. Presz, Jr., Michael J. Werle, (Fig. 2). This study is evident in the back section of this turbine, which appears to significantly affect performance. The name of the patent is: Lobed convergent / divergent nozzle ejector system / Patent number: 6877960 /. Working principle: The system includes a main lobed orifice followed by a convergent divergent sleeve where the lobed nozzle is separated from the convergent casing diverging by a small distance, which allows secondary flow and during work the current enters the main nozzle and its pressure decreases until it leaves the main nozzle and here it mixes with the secondary flow A vacuum pressure zone is created and the next coating reduces the velocity of the mixture while the pressure rises, which increases the drag performance (behind the turbine) and reduces the energy loss.

Fig.2 IRO 103.

2.3. Governing equations

Mass conservation equation

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{V}\right) = 0 \qquad (1)$$

Momentum Equation

$$\frac{\partial}{\partial t}(\rho\vec{V}) + \nabla.(\rho\vec{V}\vec{V}\ ) = -\nabla p + \nabla(\mu.\nabla\vec{V}) + \vec{f} \tag{2}$$

Where,

$$\nabla = \left(\frac{\partial}{\partial x}, \frac{\partial}{\partial y}, \frac{\partial}{\partial z}\right) \tag{3}$$

2.4. Drawing and designing the models

Models were drawn using SOLIDWORK software. The following figures illustrate SOLIDWORK 3D model of the types of diffusers studied in the article, (Fig. 3) straight diffuser with edge, (Fig. 4) curved diffuser with edge, (Fig. 5) straight diffuser with strike and edge. In addition to the diffusers in our current study, (Fig. 6) A ducted diffuser, (Fig. 7) A ducted diffuser with a second phase convergent divergent. In this analysis we first create the above 3D models with appropriate dimensions. Next these models were used in the CFD analysis in ANSYS FLUENT software.

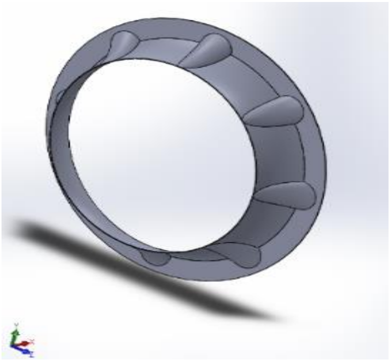


Fig.5 straight diffuser with turbulence and edge

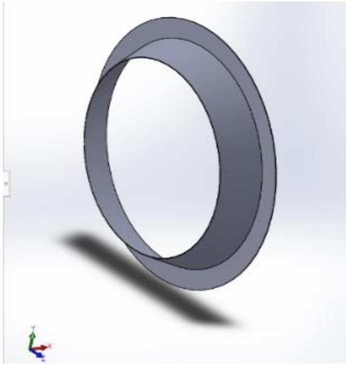


Fig.4 curved diffuser with edge

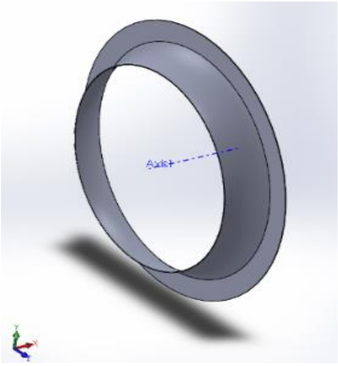


Fig.3 straight diffuser with edge

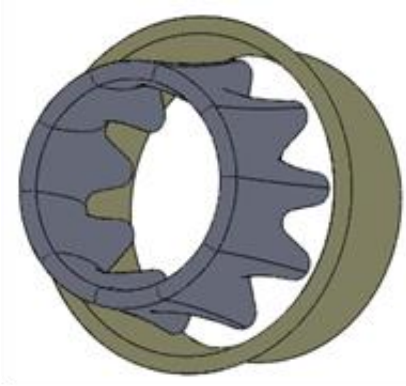


Fig.7 A ducted diffuser with a second phase convergent divergent.

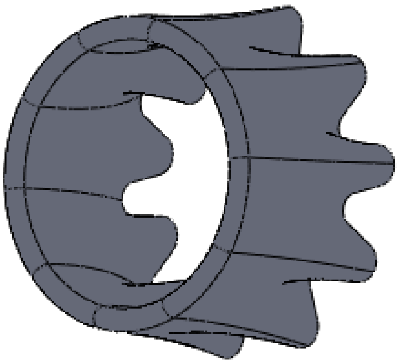


Fig.6 A ducted diffuser.

2.5. Mesh generation

(Fig. 8) shows the computational field in which the numerical study was carried out, where D= 0.35 m.

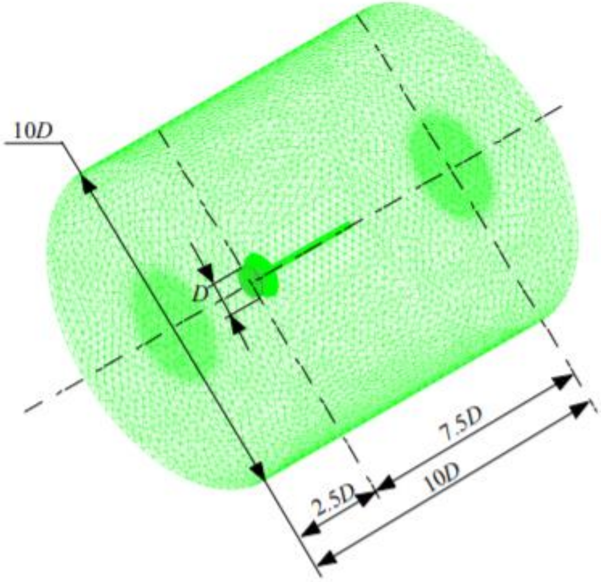


Fig.8 The computational field.

The boundary layer is designed with the following specifications:

- the initial height of the boundary layer = 0.0001 m
- Number of layers = 12 layers.
- Boundary layer growth rate = 1.4
- The total thickness of the boundary layer = 0.014 m

The grid was segmented by forming irregular triangles in the flow field for the calculations in a gradual manner until a cell count equal to 3 million was obtained, and y + less than 5 was preserved. The following inputs are: steady and incompressible flow, turbulence model (k-epsilon), velocity (6 m / s), constant and non-output pressure and reference pressure equal to atmospheric pressure. An improved wall slave application with a non-slip condition on diffuser surfaces, the solution method follows a (SIMPLE) algorithm to correlate pressure with velocity and second-order digital discretization schemes for quantities of motion and turbulence. Affinity the solution up to 10^{-3}.

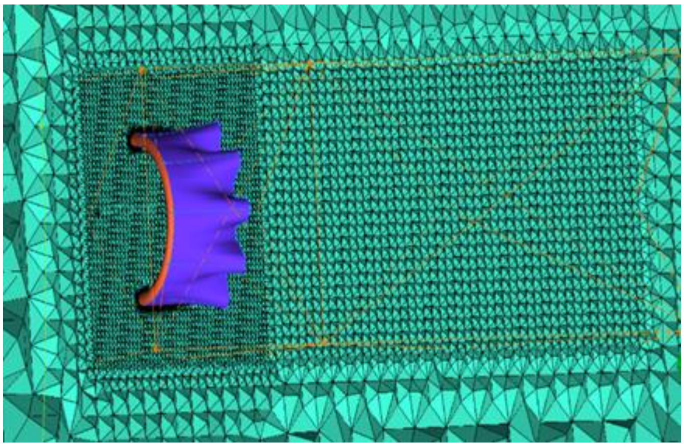


Fig.9 Network formation.

The study of network independence was performed on the divergent diffuser model where the number of cells in the computational field is graded and the mean velocity at input is calculated (where the wind turbine is placed).It can be seen from (Fig. 10) that the average velocity at the entrance became independent of the number of cells after 3 million cells, and therefore the same settings of this mesh were applied to the rest of the studied models.

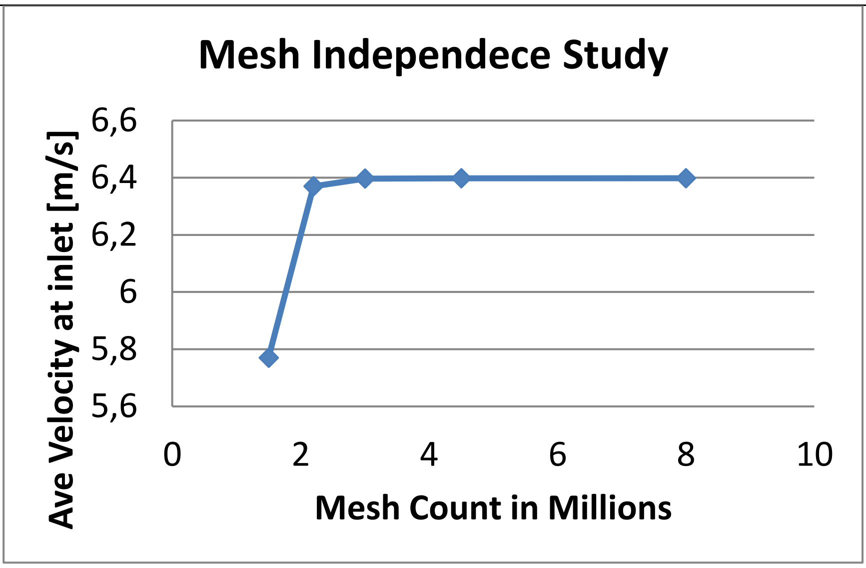


Fig.10 Network independence curve.

3. Results and discussion

The numerical calculation was performed first on the models taken from the reference study and compared with the results obtained from the reference study as shown in (Fig. 11), (Fig. 12) and (Fig. 13). The results were very close as shown in table 1, after that the numerical calculation of the studied model was performed in this paper as shown in (Fig. 14).

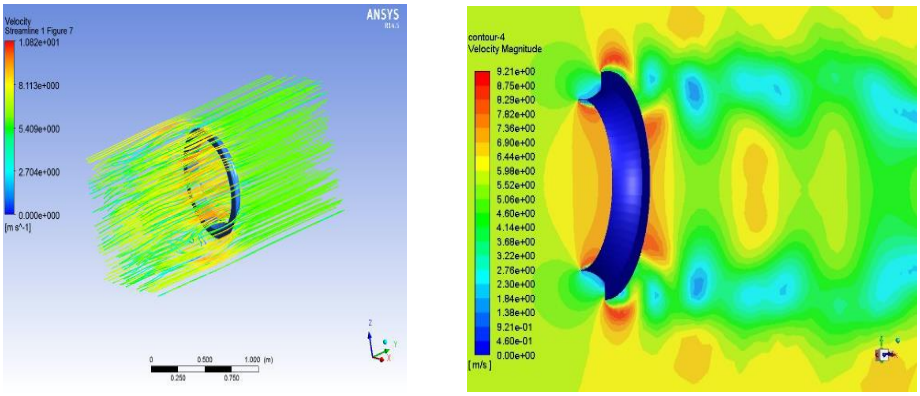


Fig.11 Comparison of the solution of a curved diffuser with edge with the reference study.

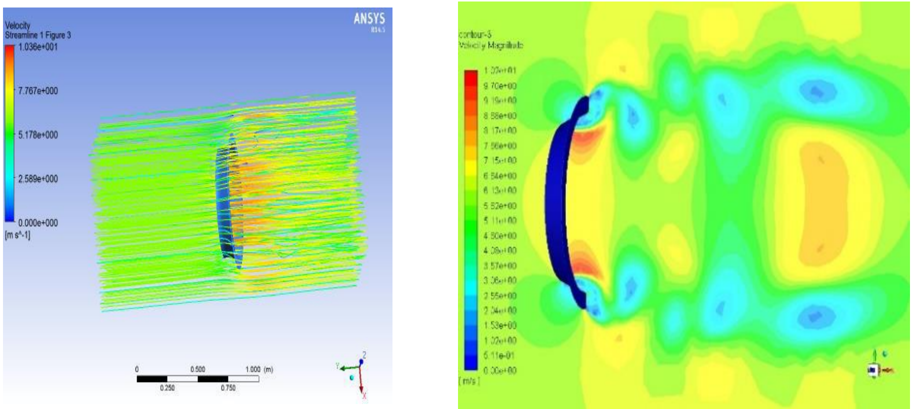


Fig.12 Comparison of the solution of a straight diffuser with edge with the reference study.

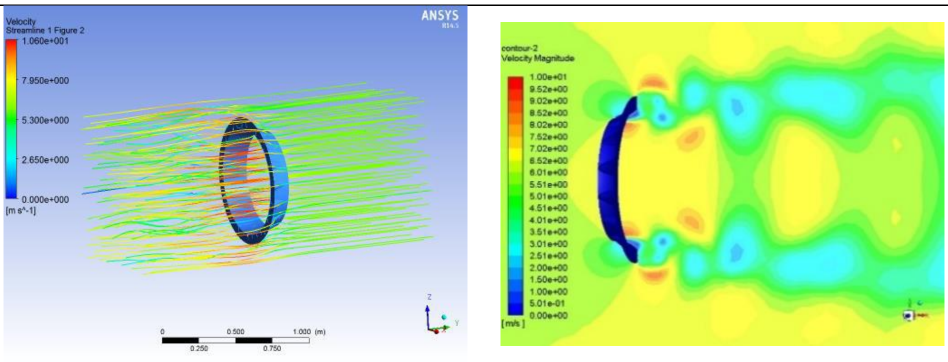


Fig.13 Comparison of the solution of a straight diffuser with turbulence and edge with the reference study.

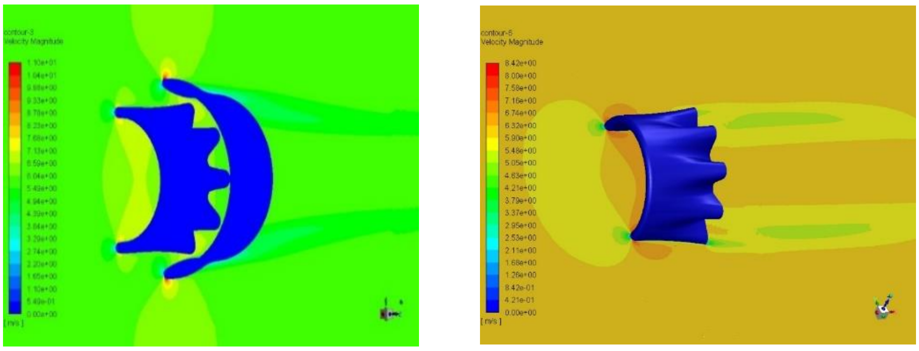


Fig.14 Numerical calculation for a diffuser with single turbulence generators and with a second stage.

table 1 shows a comparison between the results in the reference study and their solution with this study, in addition to the results of a diffuser with turbulence generators in this study.

Table 1: comparison of results.

Turbine	Velocity (m/s)		Power (Watt)
	I/P	CENTER	½ C_P ρ A V^3
Bare wind turbine From the previous study	6	6.1	19
Straight with edge From the previous study	6	9.99	80.96
Straight with edge From this study	6	9.3	
Curved with edge From the previous study	6	10.36	90
Curved with edge From this study	6	10.2	
Bumped with edge From the previous study	6	10.6	96.43
Bumped with edge From this study	6	10	
A ducted diffuser	6	8.4	52.14
A ducted diffuser with a second phase convergent divergent	6	11	117.08

CONCLUSION

This study enables us to obtain the following conclusions:

1. The results of the numerical calculations of the straight diffuser with an edge, the curved diffuser with a rim, and the suggested diffuser with turbulence generators with an edge showed a good agreement.
2. The results of the numerical calculations to the considered diffuser in one stage with turbulence generators, shows an increase in the velocity at the entrance of the diffuser equal to (8.4 m / s), which is equivalent to 1.4 of the input velocity and thus an increase in power by 2.7 times from the bare turbines.
3. Adding the second stage to the diffuser, gave an increase in the velocity at the entrance to the diffuser equal to (11 m / s), equivalent to twice the speed of the wind, and thus an increase in power by 6 times the bare turbine.
4. The considered diffuser gave an increase in energy about 21% greater than all the diffusers studied in the reference study.

REFERENCES

[1] Grady M. Isensee , Hayder Abdul-Razzak, **Modeling and analysis of diffuser augmented wind turbine**, Department of Mechanical and Industrial Engineering, Texas A&M University, Kingsville 700 University Blvd., Kingsville, USA, IJES Vol.2 Iss.3 2012 PP.84-88.

[2] Nasir Mehmood, Zhang Liang and Jawad Khan, **CFD study of NACA 0018 for diffuser design of tidal current turbines**, *Research Journal of Applied Sciences, Engineering and Technology* 4(21): 4552-4560, 2012, China.

[3] T. Saravana kannan, Saad a. Mutasher, Y.H. Kenny Lau, **Design and flow velocity simulation of diffuser augmented wind turbine using CFD**, *Journal of Engineering Science and Technology* Vol. 8, No. 4 (2013) 372 – 384, School of Engineering, Taylor's University – Malaysia.

[4] Léo Daiki Shinomiya, Déborah Aline Tavares Dias do Rio Vaz, **Numerical study of flow around diffusers with different geometry using CFD applied to hydrokinetics turbines design**, *22nd International Congress of Mechanical Engineering (COBEM 2013)* November 3-7, 2013, Ribeirão Preto, SP, Brazil.

[5] Masoud Arabbeiki Zefreh, **Design and CFD analysis of airborne wind turbine for boats and ships**, Department of Mechanical Engineering, Payam Noor University of Isfahan, Isfahan, Iran, *International Journal of Aerospace Sciences* 2016, 4(1): 14-24.

[6] J E Kesby, D R Bradney and P D Clausen, **Determining diffuser augmented wind turbine performance using a combined CFD/BEM method**, School of Engineering, Faculty of Engineering and the Built Environment, The University of Newcastle, *Australia Journal of physics: conference series*, volume 753 (2016) , Published under licence by IOP publishing ltd.

[7] Mr. Prakash ,Sreeju ,Vijaychandran, Sridharan, **Optimization of wind turbine with diffuser using CFD tool**, Coimbatore – 641 035, tn, India, International journal of engineering sciences & research, Technology 6(1): January, 2017.

[8] Mr.Gade Sagar Tukaram,. 2013. Design & Analysis of Diffuser Augmented Wind Turbine. *International Engineering Research Journal* Page No 226-237.

[9] ANSYS Fluent documentation: User guide, 2018.

[10] ANSYS Fluent documentation: Theory guide, 2018.