

PAPER • OPEN ACCESS

## Development of a Shrouded Wind Turbine with Various Diffuser Type Structures

To cite this article: Y Klistafani and M I Mukhsen 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **676** 012040

View the [article online](#) for updates and enhancements.

# Development of a Shrouded Wind Turbine with Various Diffuser Type Structures

Y Klistafani<sup>1\*</sup> and M I Mukhsen<sup>1</sup>

<sup>1</sup>Mechanical Engineering Department, State Polytechnic of Ujung Pandang, Makassar, 90425, Indonesia

\*E-mail: yiyin\_klistafani@poliupg.ac.id

**Abstract.** The Shrouded wind turbine is an innovative mean to increase power generated by wind turbine. By encompassing the rotor with a diffuser structure, it is possible to increase the wind velocity through the turbine up to 1.8 times of free stream velocity. Specifically, this study is numerical simulations that investigate the effect of wind velocity on various diffuser type structures to develop shroud wind turbine. Numerical study was conducted using Computational Fluid Dynamics (CFD) method. A reasonable agreement between the computed results, available experimental data and previous simulation is obtained. The results show that simulation results have good agreement to experimental results. However, Curved diffuser give the best increment of wind velocity (at centreline diffuser) than flat diffuser and others, which is 84.18%, with the maximum velocity was 9.21 m/s. Regarding the velocity contour results, vortex in downstream curved diffuser is bigger than others. Therefore, suction effect on curved diffuser is strongest and can generate positive impact on the wind velocity quality inside the diffuser.

## 1. Introduction

Alternative energy is very interesting to be studied more deeply considering the limited availability of fossil energy sources on earth. Wind power investigation is very important to address environmental issues. It is well known that wind speed fluctuates depend on natural weather. But if wind speed is very low, it will certainly be one of the inhibiting factors in the utilization of wind energy conversion technology. The power of the wind is proportional to the cubic power of the wind velocity approaching a wind turbine. This means that even small amount of its acceleration gives large increase of the energy generation [1]. Wind turbines usually operate for the rated wind speed of around 8-11 m/s [2, 3]. Therefore, wind turbine innovation is required in optimizing the utilization of wind, as have been done by many research groups that have tried to find a way to accelerate the wind velocity effectively [4-9]. Some interesting studies were reported by Yadav and Kumar [4], Kannan et al. [5], Igra [6], Gilbert and Foreman [7], and Lipian et al [8, 9]. Selection of the diffuser type structure as a good shroud for wind turbine was given by Ohya et al. [10] who employed three typical hollow structures. It was confirmed that the diffuser structure was the most effective for collecting and accelerating the wind than other nozzle and cylinder structures.

It is possible to model engineering problems using Computational Fluid Dynamics (CFD) approaches. The present simulations have been compared and validated against measurement data since then. The simulations range from the simplest 2D Reynolds-Averaged Navier-Stokes (RANS) approach. Wang et al. [11] investigated the dynamic stall behaviour of a pitching airfoil in comparison

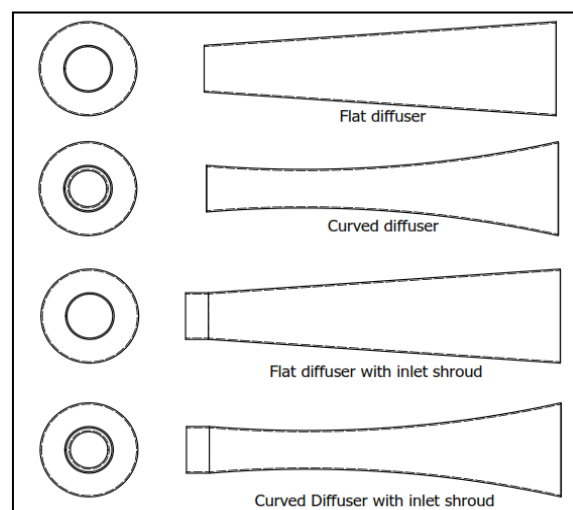


to the experimental data using the Shear-Stress-Transport (SST)  $k-\omega$  [12] and Wilcox  $k-\omega$  [13] turbulence model. They concluded that the Wilcox (or standard)  $k-\omega$  model was too dissipative and could not deliver the prediction accurately. On the other hand, the SST model provided a better agreement against the experimental data. A good agreement of the CFD computations using the SST turbulence model was also obtained in several computations, for example Pape and Lecanu [14], Weihing et al. [15], Jost et al. [16], Klistafani [17, 18] and Klistafani et al. [19]. These encourage the use of CFD for predicting the fluids engineering problems especially with the help of the Menter SST  $k-\omega$  model.

Having considered the above background, the development of a wind power system with high output aims at determining how to collect wind velocity efficiently and what kind of diffuser design can generate energy effectively from the wind speed. The computed results were compared with corresponding experimental data [10] to show the fundamental capability of predicting complex turbulent flows.

## 2. Numerical Method

Numerical study is conducted using Computational Fluid Dynamics (CFD) method. The procedures performed on numerical research are pre-processing, solving, post-processing and validations. The first stage on numerical method is designing the diffuser type structure on two dimensional. The structures are illustrated in Figure 1, while detailed information about their dimension is given in Table 1 and Figure 2.



**Figure 1.** Diffuser structures

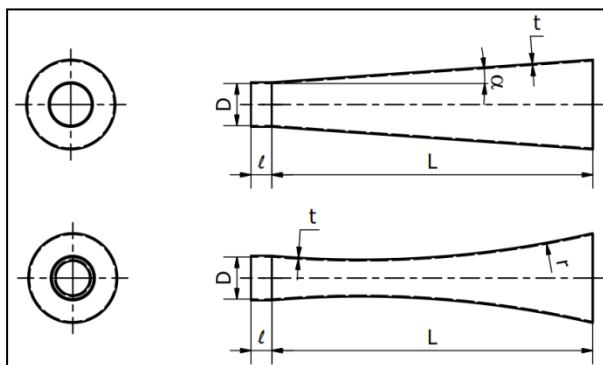
ANSYS Workbench 18.2 is software which used for modelling. The flat diffuser was generated according to the geometry specified in the experimental studies carried out by Ohya et al [10] for the length, diameter, and angle of divergence diffuser. Whereas design of curved shaped diffuser is adjusted flat diffuser dimensions that only interior modified into curved. We designed thickness of diffuser (12.5 mm) by considering recent studies [20] which used 10 layers of plates with each thickness of the plate is 1.25 mm.

The domain of the simulation is shown in Figure 3. The velocity inlet boundary condition was applied at inlet of the flow which is located five times inlet diameter of diffuser ( $D$ ). The flow leaves the computational domain at  $8.5D$  distance from the outlet of the diffuser with the outflow boundary condition. The side walls were set as a non-slip wall that are sufficiently far away from the area of interest to ensure the minimal effect on the flow characteristics near the diffuser. The computations were carried out using the commercial software Ansys Fluent 18.2. The flow was assumed to be steady, incompressibility and heat transfer effect was neglected. This is reasonable because wind

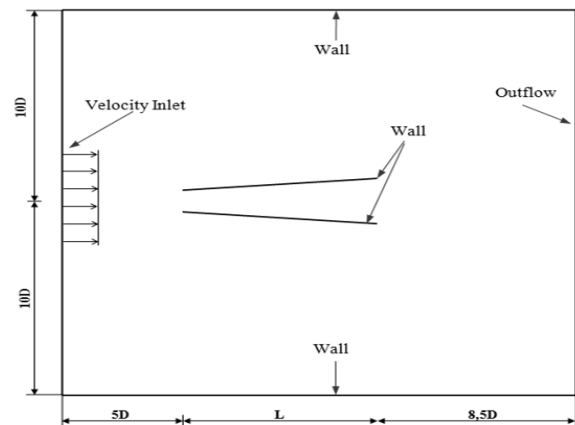
turbines usually operate at a much smaller velocity than the speed of sound. An initial wind velocity of 5 m/s was prescribed at the velocity inlet plane as same as the velocity was employed in experimental [10]. The turbulence model is using the two-equation SST  $k-\omega$  model. The pressure velocity coupling uses the SIMPLE method. All the variables were solved using the second order discretization. The computations were carried out for 10,000 iterations, otherwise convergence was achieved if the residual of the momentum reaches  $1e-6$ .

**Table 1.** Diffuser type structure dimensions.

Specification	Diffuser Type Structure			
	Flat diffuser	Curved diffuser	Flat diffuser with inlet shroud	Curved diffuser with inlet shroud
Inlet diameter ( $D$ )	400 mm	400 mm	400mm	400mm
Diffuser length ( $L$ )	3080mm	3080mm	3080mm	3080mm
Diffuser thickness ( $t$ )	12.5 mm	12.5 mm	12.5 mm	12.5 mm
Inlet shroud length ( $l$ )	-	-	200mm	200mm
Diverging angle ( $\alpha$ )	4°	4°	4°	4°
Circle radius of arch ( $r$ )	-	$10^4$ mm	-	$10^4$ mm



**Figure 2.** Detailed dimensions of diffusers with inlet shroud

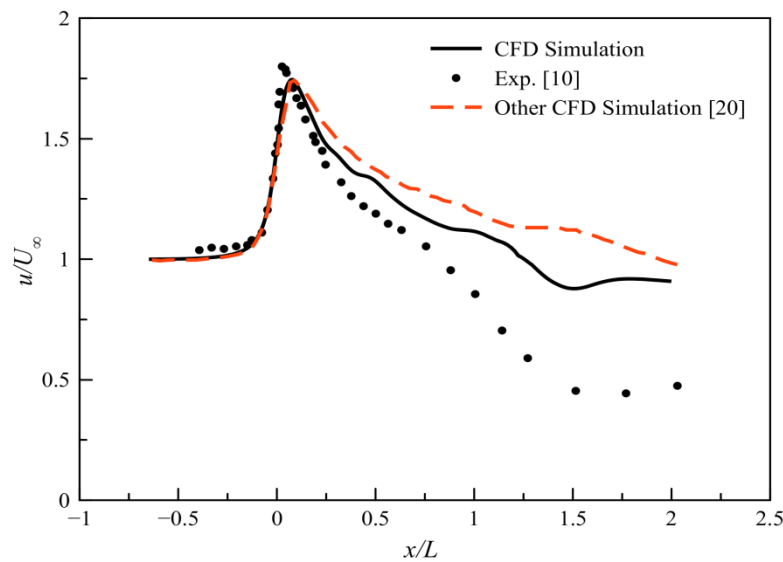


**Figure 3.** Computational domain and boundary conditions

The mesh was generated using ANSYS Workbench 18.2 software. Mesh parameters and controls are shown in Table 2. Grid independence studies were carried out in advance to ensure that the results are independent of the mesh resolution. The computational results are plotted in Figure 4 where the streamwise velocity ratios ( $u/U_\infty$ ) of flat diffuser at the centreline are compared with experimental results [10] and previous CFD Simulation conducted by Wibowo et al. [21]. It can be seen that simulation results has good agreement to experiment results with the error relative to the experiment is about 6.01%. The numerical method used greatly affects the computational results, despite using same software. This numerical simulation used second order and second order upwind for the methods and defined stricter convergency criteria than previous simulation [21]. Present simulation determine the bigger iteration limit, it is ten thousand iterations, than previous which only one thousand iterations. Therefore, those make present computations results closer to the experimental results than previous simulation.

**Table 2.** Mesh parameters and controls.

Mesh Parameters		Mesh Controls	
Size function	Curvature	Method	Quadrilateral dominant
Relevance center	Fine	Element order	Use global setting
Target skewness	0.900000	Free face mesh type	All quad
smoothing	High	Refinement	3
Inflation Option	Smooth transition	Number of divisions	90
Growth Rate	1.2	Size function	Uniform
Maximum layers	2	Bias factor	5.0

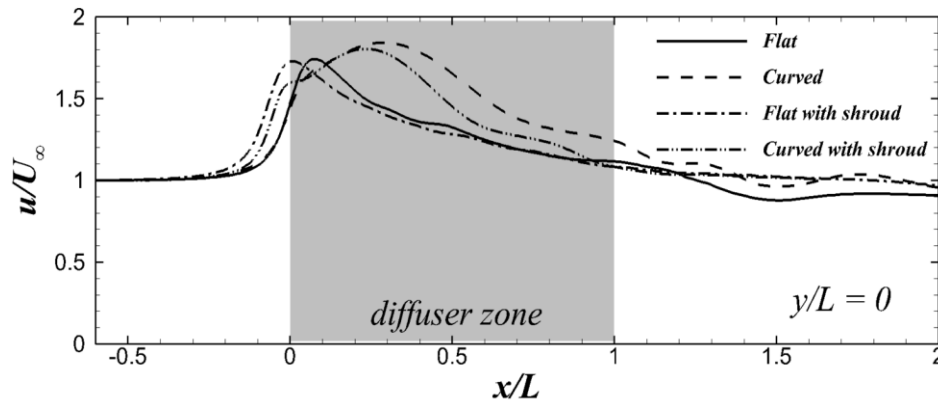
**Figure 4.** Streamwise flow velocity distribution for the flat diffuser.

Comparison of streamwise flow velocity distribution is intended to validate the results of numerical computations with experimental results. Wind speed increment data compared is data for flat diffuser, where the specification of the flat diffuser dimensions in the simulation refers to the diffuser specifications used by Ohya et al [10]. The highest velocity increase in experimental results reached 1,795 times of the freestream velocity (5 m/s), while in the numerical results the highest wind velocity increase reached 1,742 times. This indicates that the difference in numerical predictions with experiments in terms of the highest wind velocity increase about 2.95%.

### 3. Results and Discussion

The dimensionless streamwise velocity at centreline diffuser along the axial positions plots for all diffuser type structures are presented in Figure 5. Curved diffuser generates wind velocity increment highest than other diffuser type structures, which is up to 1.842 times ( $x/L = 0,285$ ) of initial wind velocity set up (5 m/s). It is found that the inlet shroud has no significant impact on the augmentation of the wind velocity, but it just makes wind velocity increase occurs earlier. However, the highest wind speed between flat diffuser and curved diffuser occurs at different sections. The smallest cross-sectional area both of them is different, so that makes it happen. Flat diffuser has an inlet section as a smallest cross-sectional area, while the smallest cross-sectional area of curve diffuser has distance of several mm from the entrance section. It is noted, however, that the curved diffuser has a better performance than the flat diffuser, provided that location of the rotor is not at the near entrance but

around  $x/L \approx 0.36$ . Detailed information regarding the comparison of the wind velocity at centreline of all diffuser type structures is shown in Table 3.



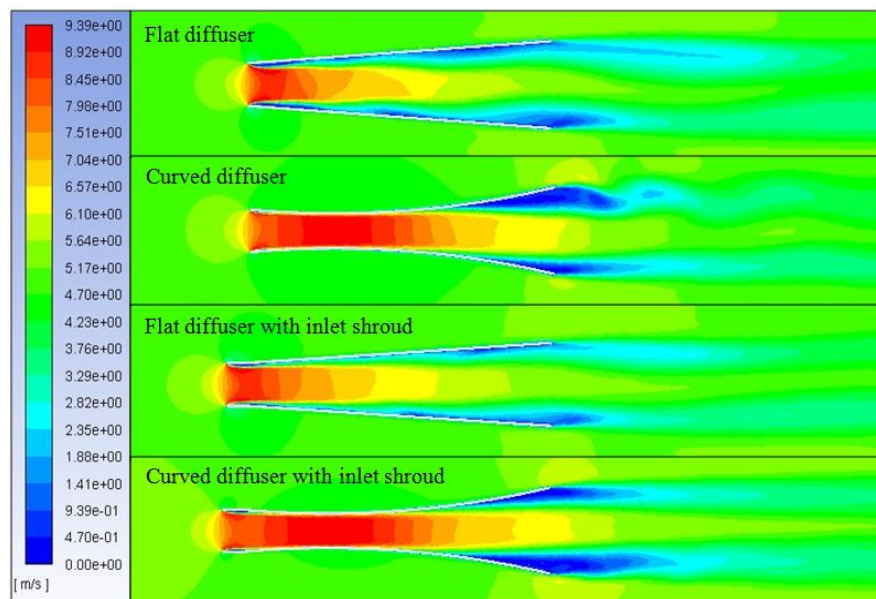
**Figure 5.** Streamwise flow velocity distribution at centreline diffuser for four diffuser type structures

**Table 3.** Wind velocity at centreline for all diffuser type structures

Value	Flat diffuser	Curved Diffuser	Flat diffuser with inlet shroud	Curved diffuser with inlet shroud
Wind velocity at inlet diffuser ( $m/s$ )	7.34	7.20	8.63	7.98
Maximum wind velocity ( $m/s$ )	8.71	9.21	8.64	9.02
Location of maximum wind velocity ( $x/L$ )	0.074	0.285	0.009	0.233
Increment	74.23%	84.18%	72.80%	80.39%

The velocity contour for all diffuser type structures is shown on Figure 6. As seen in Figure 6, the wind flows into the diffuser as it is inhaled. The vortex formed in the downstream area can give the effect of suction fluid into the diffuser. The greater the flow vorticity that occurs, the better the increase in wind velocity enters the diffuser. The flow vortices that occur in the downstream area for all four diffusers vary depending on the type of diffuser structure. Vortex size that formed in the downstream curved diffuser area is greater than the flat diffuser's, therefore curved diffuser is better in increasing wind velocity.

As seen on Figure 6, inlet shroud that attached on flat diffuser instead the size of the vortex in the downstream area becomes smaller than flat diffuser without inlet shroud. The addition of inlet shroud makes the suction flow effect decrease. Therefore, as seen in Table 3, the increment of wind velocity generated by the flat diffuser is 1.43% larger than the flat diffuser with inlet shroud. The same thing happened to the curved interior diffuser, where the curved diffuser has better flow suction power compared to curved diffuser with inlet shroud which the difference of wind velocity increment about 3.97%.



**Figure 6.** Velocity contour of all diffuser type structures (velocity magnitude,  $m/s$ )

#### 4. Conclusions

Numerical simulations were carried out for flow fields around various diffuser type structures to develop shrouded wind turbine. The main conclusions derived from the study are as follows:

- Computational results (present study) are in good agreement with the corresponding experimental data. It has been confirmed from those validation that present computational procedure is very useful to investigate flow field of the diffuser type structure.
- The curved diffuser gave the best increment of wind velocity than flat diffuser and others, which was 84.18%, with the maximum velocity was 9.21 m/s.
- The vortex in downstream curved diffuser is bigger than others. Therefore, suction effect on curved diffuser is strongest and can generate positive impact on the wind velocity quality inside the diffuser.

#### Acknowledgments

The research leading to these results has received funding from the directorate of research and community service, directorate general of strengthening research and development, ministry of research, technology and higher education of Indonesia.

#### References

- [1] Abe K and Ohya Y 2004 An investigation of flow fields around flanged diffusers using CFD *J. Wind Engineering and industrial aerodynamics* **92** 315.
- [2] Bangga G, Hutomo G, Wiranegara R and Sasongko H 2017 Numerical study on a single bladed vertical axis wind turbine under dynamic stall *J. Mech. Science and Tech.* **31** 261.
- [3] Bangga G, Lutz T, Jost E and Krämer E 2017 CFD studies on rotational augmentation at the inboard sections of a 10 MW wind turbine rotor *J. Renewable and Sustainable Energy* **9** 023304.
- [4] Yadav A J and Kumar D 2017 Review of a shrouded wind turbine for low wind speeds International digital library of technologi & Research **1**.
- [5] Kannan S A and Lau Y H K 2013 Design and flow velocity simulation of diffuser augmented wind turbine using CFD *J. Eng. Science and Tech.* **8** 372.
- [6] Igra O 1981 Research and development for shrouded wind turbines *Energy Convers. Manage.* **21** 13.

- [7] Gilbert B L and Foreman K M 1983 Experiments with a diffuser-augmented model wind turbine *J. Energy Resour.* **21** 13.
- [8] Lipian M, Karczewski M and Olasek K 2015 Sensitivity study of diffuser angle and brim height parameters for the design of 3kW diffuser augmented wind turbine *Open Eng.* **5** 280.
- [9] Lipian M, Karczewski M and Molinski J 2016 Numerical simulation methodologies for design and development of diffuser-augmented wind turbines – analysis and comparison *Open Eng.* **6** 235.
- [10] Ohya Y, Karasudani T, Sakurai A, Abe K and Inoue M 2008 Development of a shrouded wind turbine with flanged diffuser *J. Wind Eng. and Industrial Aerodynamics* **96** 524.
- [11] Wang S, Ingham D B, Ma L, Pourkashanian M and Tao Z 2010 Numerical investigations on dynamic stall of low reynolds number flow around oscillating airfoils *Computers & Fluids* **39** 1529.
- [12] Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications *AIAA Journal* **32** 1598.
- [13] Wilcox D C 1993 Turbulence modelling for CFD *DCW Industries*.
- [14] Pape A L and Lecanu J 2004 3D Navier–stokes computations of a stall regulated wind turbine *Wind Energy* **7** 309.
- [15] Weihing P, Letzgun J, Bangga G, Lutz T and Krämer E 2016 Hybrid RANS/LES capabilities of the flow solver FLOWer—Application to flow around wind turbines *Proc. Symposium on Hybrid RANS-LES Methods Springer Champ* 369.
- [16] Jost E, Fischer A, Bangga G, Lutz T, and Krämer E 2017 An investigation of unsteady 3-D effects on trailing edge flaps *Wind Energy Science* **2** 241.
- [17] Klistafani Y 2017 Studi numerik steady RANS aliran fluida di dalam asymmetric diffuser *J. INTEK* **4** 20.
- [18] Klistafani Y 2018 Karakteristik aliran fluida di dalam asymmetric diffuser dengan penambahan vortex generator *J. INTEK* **5** 21.
- [19] Klistafani Y, Mukhsen M I, and Bangga G 2018 Assessment of various diffuser structures to improve the power production of a wind turbine rotor *Technische Mechanik* **38** 256.
- [20] Hu J and Wang W 2015 Upgrading a shrouded wind turbine with a self adaptive flanged diffuser *Energies* **8** 5319.
- [21] Wibowo A T H, Wahyuono R A and Nugroho G 2013 Studi numerik pengaruh geometri dan desain *diffuser* untuk peningkatan kinerja DAWT (*diffuser augmented wind turbine*) *J. Teknik Mesin* **14** 90.