Scientific Journal of Impact Factor (SJIF): 5.71
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 7, Issue 08, August -2020

TO OPTIMIZE THE DIFFUSER DESIGN OF DIFFUSER AUGMENTED WIND TURBINE

[1]Pramod Kumar, [2]Anoo Dadheech, [3]Jagdish Suthar

[1]M.tech Scholar (TE), Department of Mechanical Engineering, Aravali Institute of Technical Studies, Udaipur, Rajasthan, India
[2]Assistant Professor, Department of Mechanical Engineering, Aravali Institute of Technical Studies, Udaipur, Rajasthan, India
[3]Assistant Professor, Department of Mechanical Engineering, Pacific university, Udaipur, Rajasthan, India

ABSTRACT:- *The main objective of this research is to optimize the diffuser design of Diffuser Augmented Wind Turbine (DAWT). Specifically, this study investigates the effect of wind velocity on different shapes of flanged diffusers to develop the suitable diffuser for the wind turbine. For that purpose, the ideologies of DAWT have been studied. Different diffuser design concepts were developed and the wind speed for each design is simulated using design software Solid works and Computational Fluid Dynamic (CFD) software ANSYS Fluent. Results show that After Numerous study, by keeping flange at the exit velocity increases by 18.75% and by increasing length of diffuser velocity increases by 8-10%.*

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

INTRODUCTION

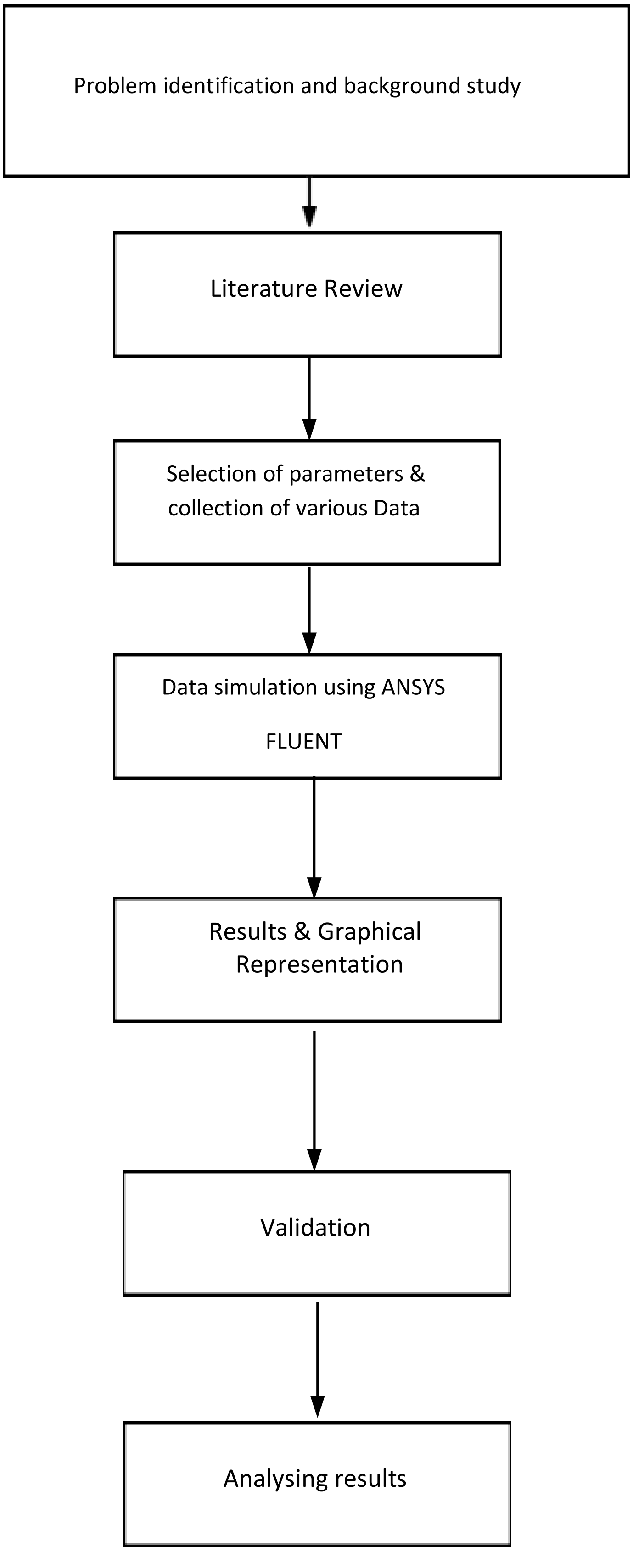
For the application of an effective energy resource in the future, the limitation of fossil fuels is clear and the security of alternative energy sources is an important subject. furthermore, as concerns for environmental issues, i.e. global warming, etc the development and application of renewable and clean new energy are strongly expected. Among others, wind energy technologies have developed rapidly and are about to play a big role in new energy field.

Wind power generation is proportional to the wind speed cubed. Therefore, a large increase in output is brought about if it is possible to create even a slight increase in the velocity of the approaching wind to a wind turbine. If we can increase the wind speed by utilizing the fluid dynamic nature around a structure or topography, namely if we can concentrate the wind energy locally, the power output of a wind turbine can be increased substantially.

LITERATURE REVIEW

In the study conducted by **Y. Ohya *et al.* (2008),** they have developed a wind turbine system that consists of a diffuser shroud with a broad-ring flange at the exit periphery and a wind turbine inside it. The flanged-diffuser shroud plays a role of a device for collecting and accelerating the approaching wind. Emphasis is placed on positioning the flange at the exit of a diffuser shroud. **T. Saravana Kannan *et al.* (2013):** The main objective of this research is to optimize the diffuser design of Diffuser Augmented Wind Turbine (DAWT). Specifically, this study investigates the effect of wind velocity on different shapes of flanged diffusers to develop the suitable diffuser for the wind turbine. **Abhishiktha Tummala *et al.* (2014)** This paper presents review of on different types of small scale wind turbines i.e., horizontal axis and vertical axis wind turbines. The performance, blade design, control and manufacturing of horizontal axis wind turbines were reviewed. **Kamyar Mansour *et al.* (2014):** in this Computation was done for flow fields around flanged diffusers to study small-type wind turbines. In these calculations, Spalart Allmaras & k–ε RNG turbulent models were used for solving corresponding Reynolds Average equations. Comparison of the computed results shows agreement with the corresponding experimental data. The purpose of the study conducted by **Aly M. El-Zahaby *et al.* (2016)** is to development and analysis of 2-D axisymmetric CFD model of flanged diffuser that was used as a casing for developed small wind turbines to increase the generated power. Study conducted by **Nugroho Agung Pambudi *et al.* (2017):** The main objective of this research is to investigate the effect of attaching nozzle lens with difference diameter and number of blades in non-twisted NACA 4415 on the power output of the Horizontal Axis Wind Turbine (HAWT). **Aierken Dilimulati *et al.* (2018):** This paper reviews the state-of-the-art of urban wind energy by examining the various types of urban wind turbine designs, with a view to understand their performance and the synergy between the turbines and the urban environments.

METHODOLOGY



RESULTS & DISCUSSION

Optimization of Diffuser geometry:
Different cases: case 1

L/D	H/D	Diffuser opening angle(degree)
1	0	4

Case 2 :

L/D	H/D	Diffuser opening angle(degree)
1	0.25	4

Case 3:

L/D	H/D	Diffuser opening angle(degree)
1	0.5	4

Case 4:

L/D	H/D	Diffuser opening angle(degree)
1.5	0	4

Case 5:

L/D	H/D	Diffuser opening angle(degree)
1.5	0.25	4

Case 6:

L/D	H/D	Diffuser opening angle(degree)
1.5	0.5	4

Case 7:

L/D	H/D	Diffuser opening angle(degree)
2	0	4
2	0.25(without converging portion)	4

Case 8:

L/D	H/D	Diffuser opening angle(degree)
2	0.25	4

Case 9:

L/D	H/D	Diffuser opening angle(degree)
2	0.5	4

Table 1. Different Geometry of Diffuser

Effect of Converging portion:

By considering case 7: With converging portion

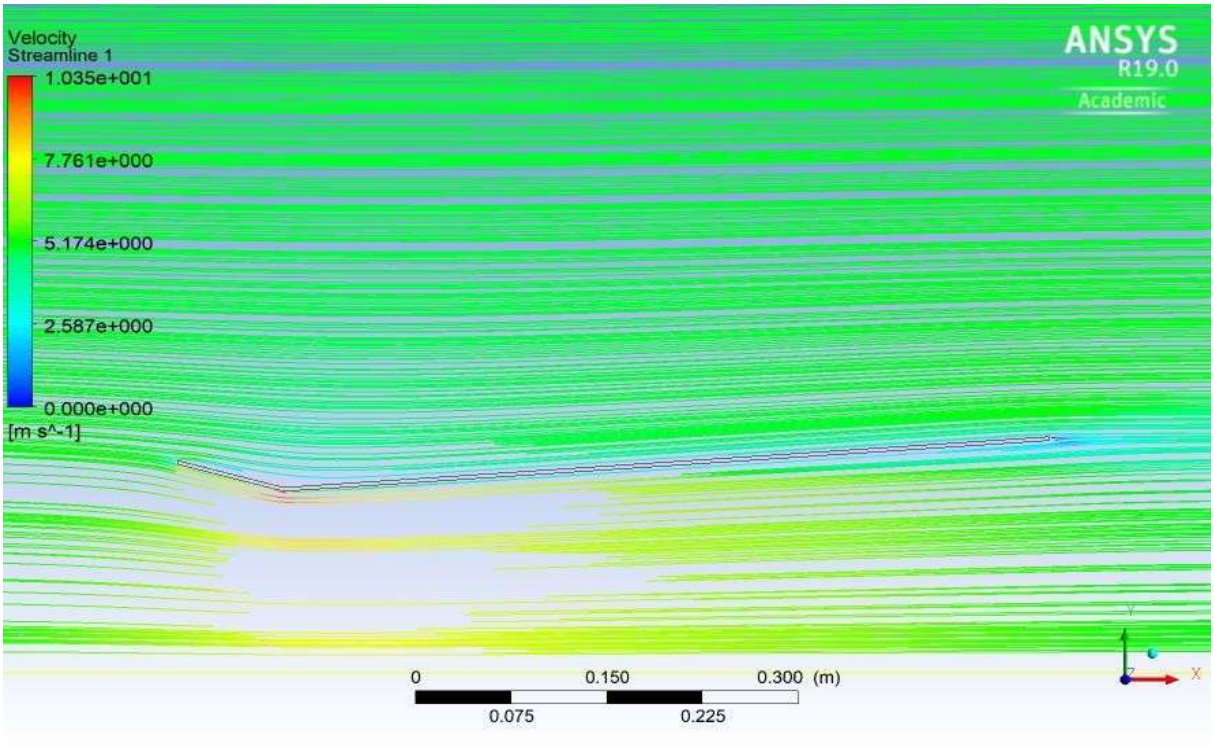


Fig 1. Velocity Streamline for L/D=2, H/D=0

Case 7 (without converging portion**)**

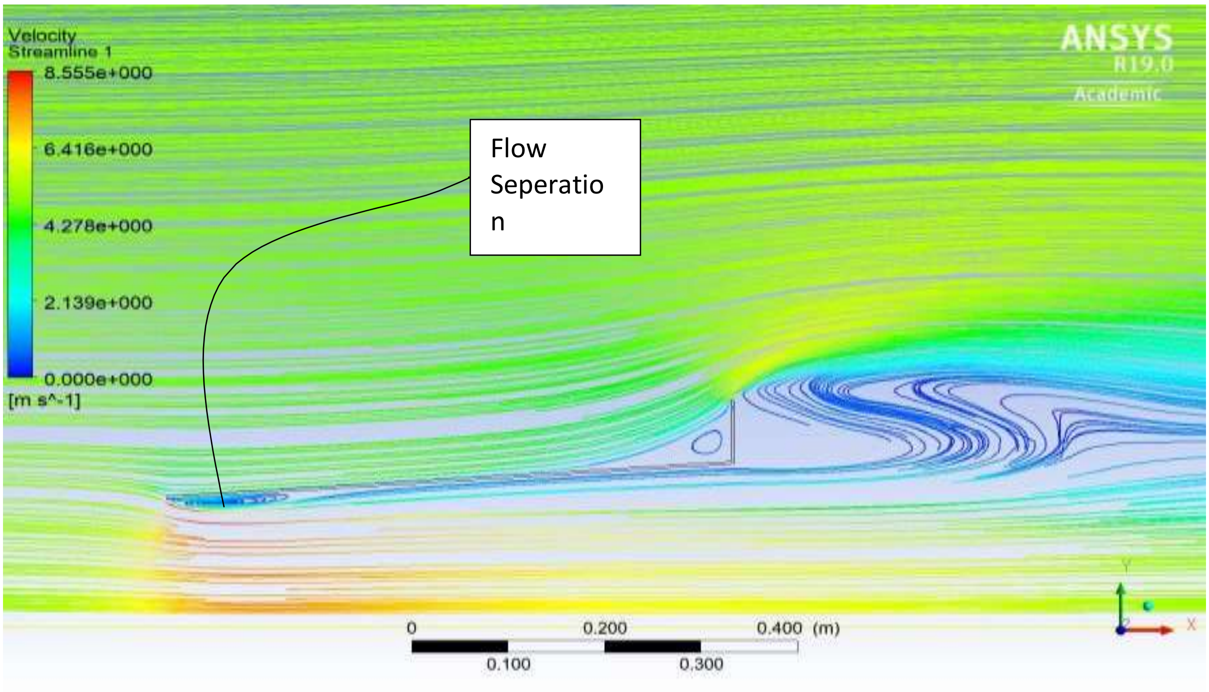


Fig 2. Velocity Streamline for L/D=2,H/D=0.25

By Figure 1 and 2 it is found that velocity in increased only by 1.2% with converging portion which agrees Abe ohya et al result (1.21% increment in velocity), but the effect of the inlet shroud is that it restrains flow separation at the entrance fairly well and the wind flows in more smoothly and Static Environment can be avoided at the entrance by attaching convergent portion.

Effect of flange:

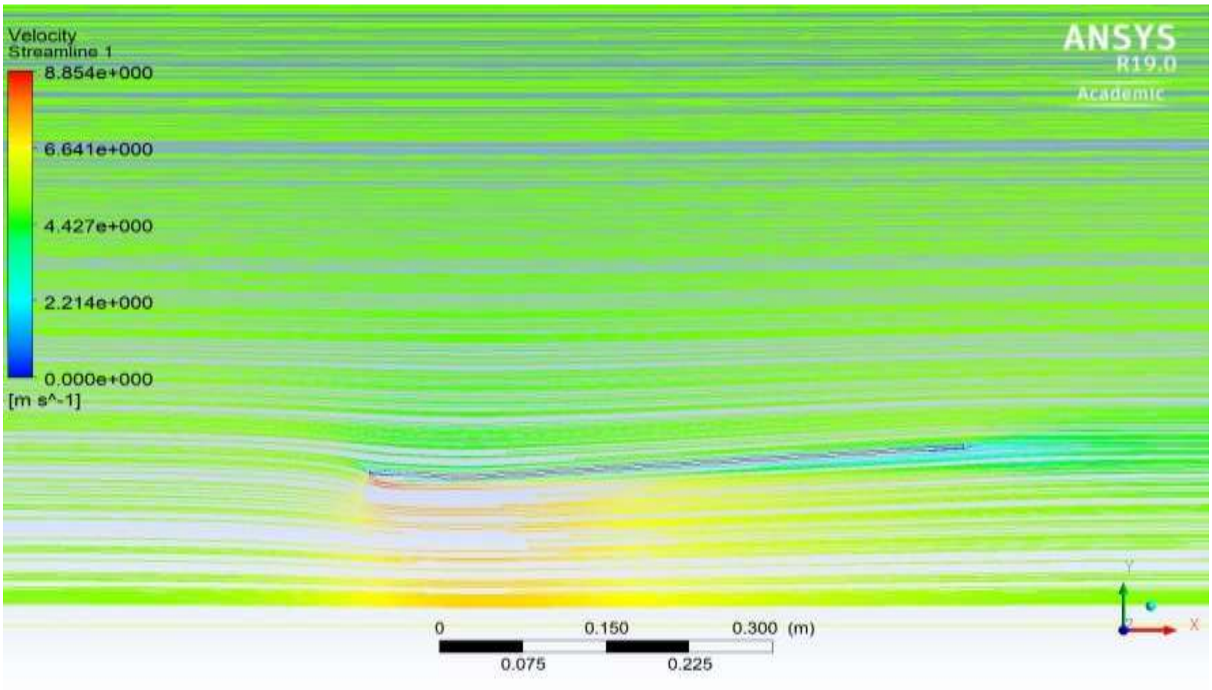


Fig 3. Velocity Streamline for L/D=1.5, H/D=0

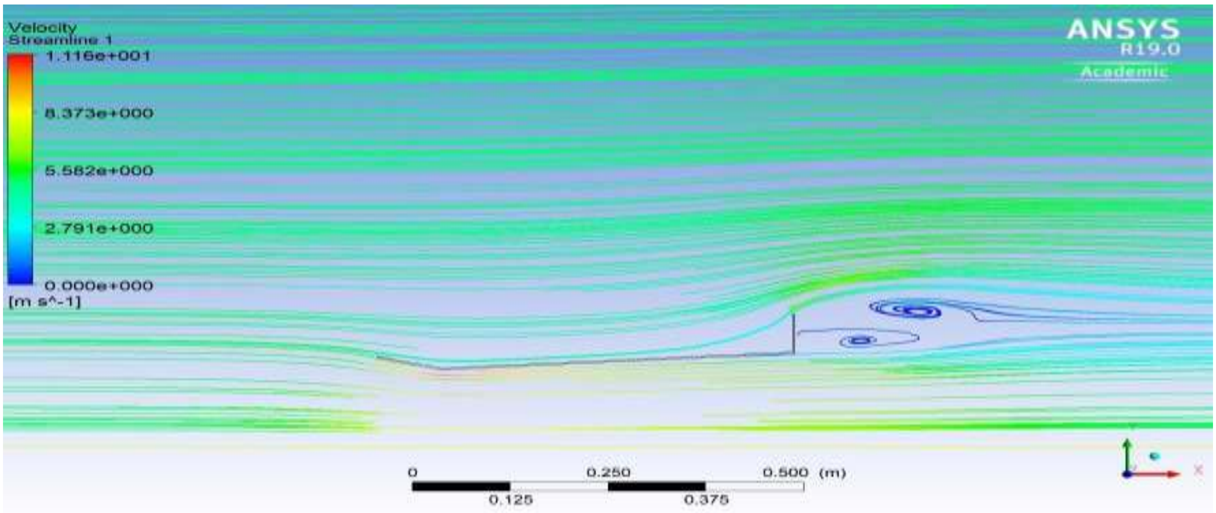


Fig 4. Velocity Streamline for L/D=1.5, H/D=0.25

By Figure 3 there is no flange attach to it so there is no wake formation behind diffuser. In figure 4 there is wake formation or flow separation happened because of flange. Due to Flange there is increase in velocity from 7.044 m/sec to 8.365 m/sec, which is approximate 18.75% more than the former value, which is shown in Figure 5, which agrees with work of Abe and ohya et al.

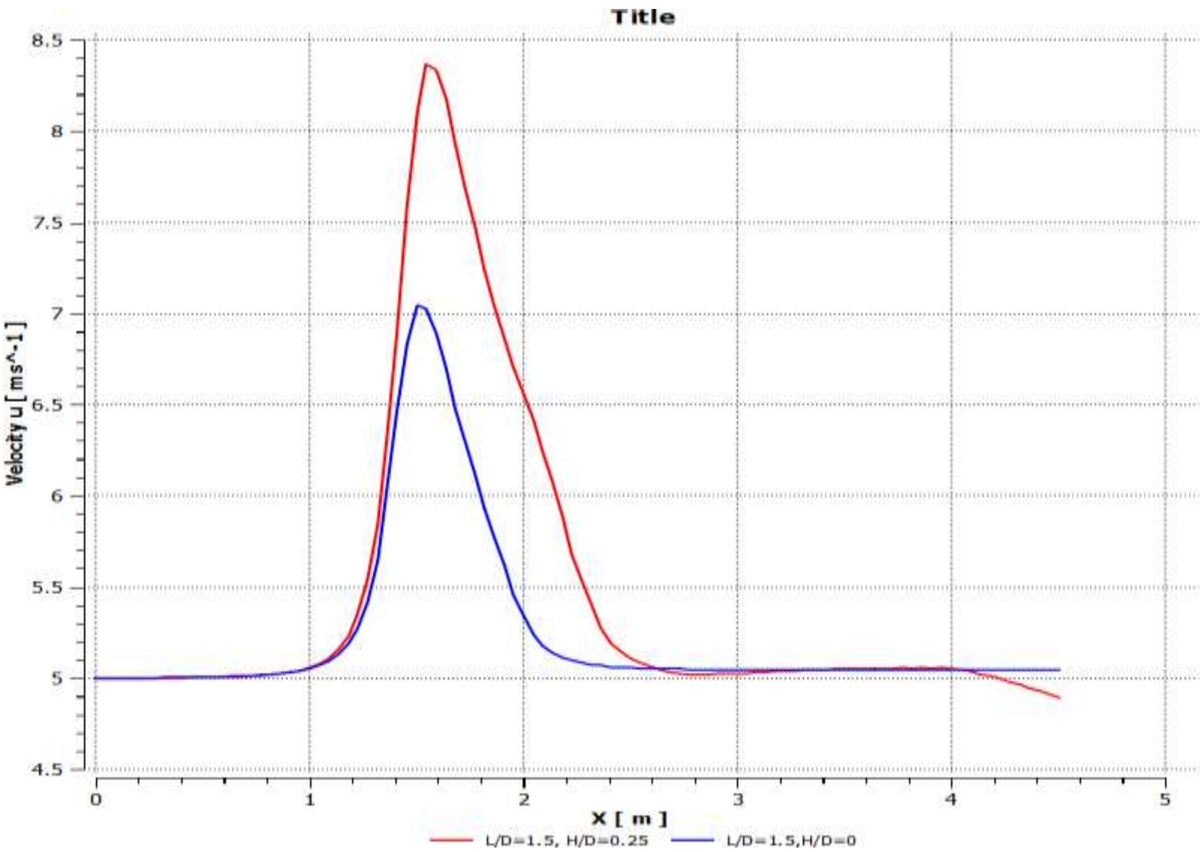


Fig 5 Comparison of velocity for different flange height

CONCULISON

Concluding remarks:

Numerical investigations were carried out for flow fields around flanged diffusers to develop small- type wind turbines under 1.5 kW: The main conclusions derived from the study are as follows:

- Computational results were in good agreement with the corresponding Abe et al work. It has been confirmed from this fact that the present computational procedure is very useful to investigate flow fields of this kind.
- CFD calculation was done for flow fields around flanged diffusers to study small-type wind turbines. In this calculations, SST-kw model were used for solving corresponding Reynolds Average equations.
- I have considered the 9 Different cases of diffuser geometry for numerical study.
- After Numerous study, by keeping flange at the exit velocity increases by 18.75% and by increasing length of diffuser velocity increases by 8-10%.
- Out of 9 above mentioned cases 9 (L/D=2,H/D=0.5) shows Optimize Result.
- I have validated the result of present calculation with the existing literature ref[1] for above mentioned 9 cases.
- The wind turbine equipped with a flanged and inlet diffuser shroud demonstrated by realizing remarkable increase in wind speed of 1.5–1.9 times that of the approaching wind speed therefore the power augmentation for a given turbine diameter and wind speed by a factor of about four compared to a standard (bare) wind turbine.

REFERENCES

[1] Abe, K., Nishida, M., Sakurai, A., Ohya, Y., Kihara, H., Wada, E., Sato, K., 2005 Experimental and numerical investigations of flow fields behind a small wind turbine with a flanged diffuser. J. Wind Eng. Ind. Aerodyn. 93, 951–970.

[2] Abe, k., Ohya, y., 2004. An investigation of flow fields around flanged diffusers using CFD. J. Wind Eng. Ind. Aerodyn., 92315–92330.

[3] Bet, F., Grassmann, H., 2003. Upgrading conventional wind turbines. Renew. Energy 28, 71–78

[4] Igra, O., 1981. Research and development for shrouded wind turbines. Energy Convers. Manage. 21, 13–48

[5] Mansour K., Meskin khoda P., May 29–June 1 2012. Numerical analysis of flow fields around flanged diffusers. In: Proceedings of the Sixth International Conference on Thermal Engineering: Theory and Applications. Istanbul, Turkey.

[6] Ohya, Y., Karasudani, T-A, 2010a. Shrouded wind turbine generating high output power with wind-lens technology. Energies 3, 634–649.

