Proceedings of the International Conference on Applied Mechanical Engineering Research (IC-AMER2019)
NIT Warangal, India – May 02-05, 2019
Paper No. XXX (The number assigned when the papaer is accepted)

CFD analysis of wind turbine with different flange angles

Sachin Bichitkar, Prasad Buddiyal, Siddhant Chavan, Ajinkya Kulkarni, Dr. V. B. Gawande*
Department of Mechanical Engineering, V.P.K.B.I.E.T, Baramati-413133, Maharashtra, India.
*Corresponding author Email: vipin.gawande@vpkbiet.org

CFD analysis of circular diffuser with flange is carried in present analysis using ANSYS CFD tool. A wind turbine with circular diffuser and flange attached over the periphery of the diffuser is simulated for various flange angle range of 0° to 25°. Provision of diffuser has demonstrated significant augmentation in power and speed of the turbine. The study also shows that the variation of flange angle creates strong vortices behind the flange, resulting in sudden decrease in static pressure in the exit of the diffuser. This will increases the velocity through entrance of the wind turbine and is responsible for power augmentation in the wind turbine. Contour plots shows increase in velocity up to optimum flange angle of 15° and then it decreases gradually afterwards. The results obtained in present analysis are in good agreement with the previous published experimental work.

***Keywords*:** Wind lens turbine; Circular Diffuser; CFD analysis; Flange angle.

1. Introduction

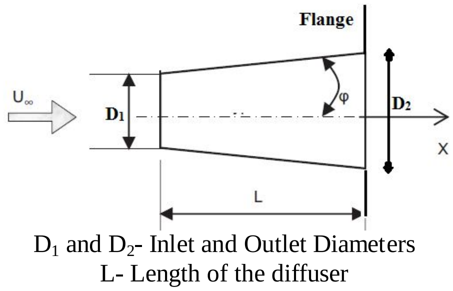
Harnessing the wind is one of the cleanest, most sustainable ways to generate electricity. Wind power produces no toxic emissions and none of the heat-trapping emissions that contribute to global warming. This, and the fact that wind power is one of the most abundant and increasingly cost-competitive energy resources, makes it a viable alternative to the fossil fuels that harm our health and threaten the environment.

Small wind turbines are electric generators in which the energy of the wind is converted into clean, emissions-free energy for individual homes, farms, and small businesses. Efficiency of the wind turbine is affected by the design of wind turbine assembly. The speed of the wind turbine blades could be increased by providing wind lens, which in turn enhances in overall efficiency of the conventional wind turbine. Yuji Ohya[1] and his colleagues 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. This arrangement shows increase in power and wind speed by a factor of about 4-5 compared to standard wind turbine. Yuji Ohya and Takashi Karasudani[2] have experimentally investigated a diffuser shroud with a broad-ring brim at the exit periphery and noted a power augmentation by a factor of about 2–5 compared with a bare wind turbine. Kazuhiko Toshimitsu[3] and his colleagues have experimentally investigated the performance of wind turbine with flanged-diffuser shroud in sinusoidally oscillating and fluctuating velocity flows. Buyung Kosasih and Andrea Tondelli[4] presented effect of diffuser shape on the performance of wind turbine. R. Bontempo, M. Manna[5] has investigated a CFD model for aerodynamic performance of ducted wind turbines. S.A.H.Jafari and B.Kosasih[6] has carried out experimental and CFD analysis of diffuser shrouded horizontal axis wind turbine. Their study proposed a method to design effective frustum diffuser geometries for a small wind turbine. A CFD analysis of wind turbine with a shroud and lobed ejector is carried out by Wanlong Han[7] and his colleagues. Aly M. El-Zahaby[8] and his colleagues has carried out CFD analysis of flow fields for shrouded wind turbine's diffuser model with different flange angles. Hasim A. Heikal[9] and his colleagues have investigate the effects of diffuser flange inclination angle and the diffuser flange depth inside the exit of the diffuser on the wind-lens turbine system performance.

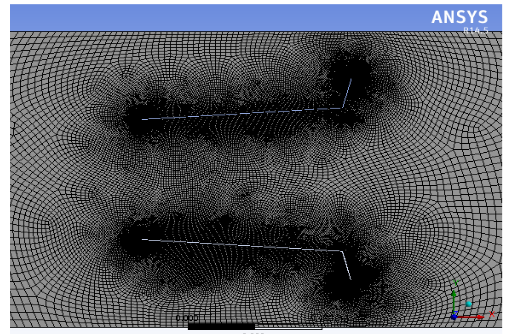
Literature review shows that overall performance of a small wind turbine could be enhanced with an addition of wind lens in the assembly. In present study, CFD analysis of a small turbine is carried out using Ohya's circular diffuser model[1]and flow simulation effects are investigated by varying flange angle (θ).

2. CFD Analysis

In present work, a computational domain of a wind turbine[1] with circular diffuser (D_1= 40 mm and D_2= 48 mm, L/D_1 =1.5) and a flange with a height of, h = 10 cm (h/D_1= 0.25) attached to the outer periphery of the diffuser exit as shown in the **Figure 1 (a)**.



(a) Circular-Diffuser model with flange



(b) Unstructured mesh around diffuser

Fig. 2 (a) Computational domain and (b) meshing on computational domain.

The area ratio (μ = Outlet area/ Inlet area) used is 1.44. The angle of inclination (φ) is 3.7° and wind velocity, U_∞= 5 m/s. ANSYS 14.5 CFD tool is used for pre and post processing. Unstructured mesh (Fig. (b)) is created on computational domain with 65000 nodes. SIMPLE algorithm is used along with k-ε turbulence model in FLUENT solver.

2. 1. CFD Results

Present computational model is verified with the experimental results, carried out by Ohya[1] and is shown in **Figure 3.**

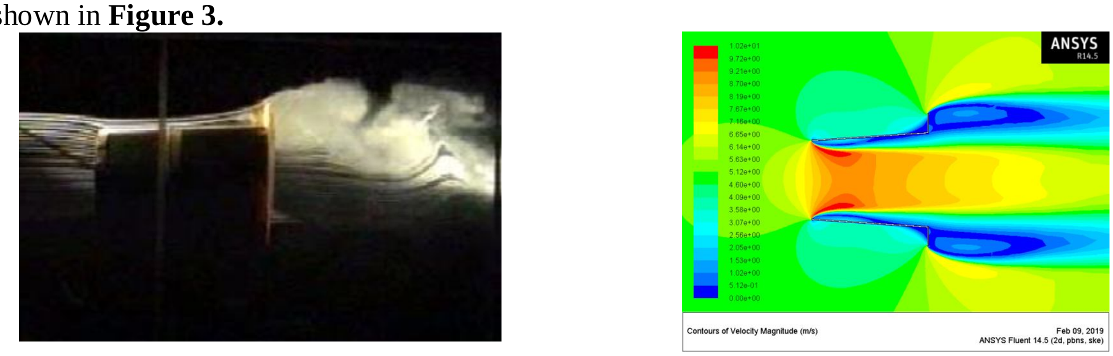


Fig.3. Comparison of Flow around a circular-diffuser experimental model of Ohya[1] with a brim and CFD analysis result.

To predict the performance of a wind turbine, flange angle is varied from θ = 0° to θ = 25° for the specified set of geometrical and operating parameters. Velocity contour plots for flange angle range (0° to 25°) are shown in **Fig. 4.** Based on the CFD results, comparison of velocity increase at diffuser entrance at various flange angles is plotted in **Fig. 5.** The basic principle adopted for the increase in velocity at the upstream of diffuser is vortex formation in the downstream of the flange. A series of vortices are visible in the contour plots along the length of the diffuser. These vortices are responsible for reducing the static pressure in the exit area of the diffuser. Reduction in pressure coefficient in the downstream of diffuser is responsible for increase in velocity at the entry of diffuser. Higher velocity is responsible for power enhancement compared to conventional wind turbine. From **Fig. 5,** it is concluded that the optimum flange angle is 15°, at which there is maximum entrance velocity increase (U_{max}/U_∞= 1.8). After flange angle 15°, there is a gradual drop in the velocity due to reduction in vortices and hence the entrance velocity decreases.

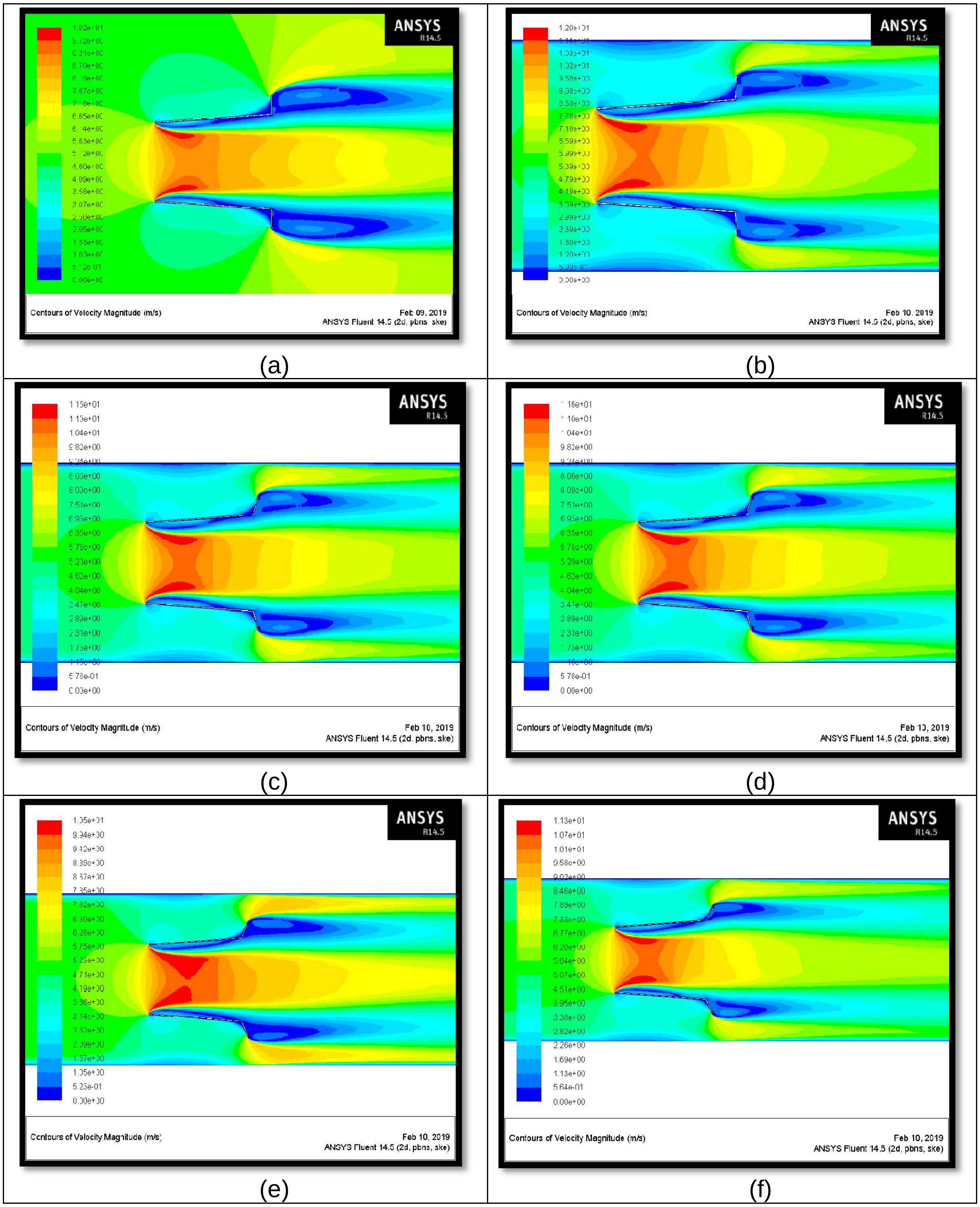


Fig. 4. Velocity contour plots of flow fields at flange angle of (a) 0°, (b) 5°, (c) 10°, (d) 15°, (e) 20° and (f) 25°.

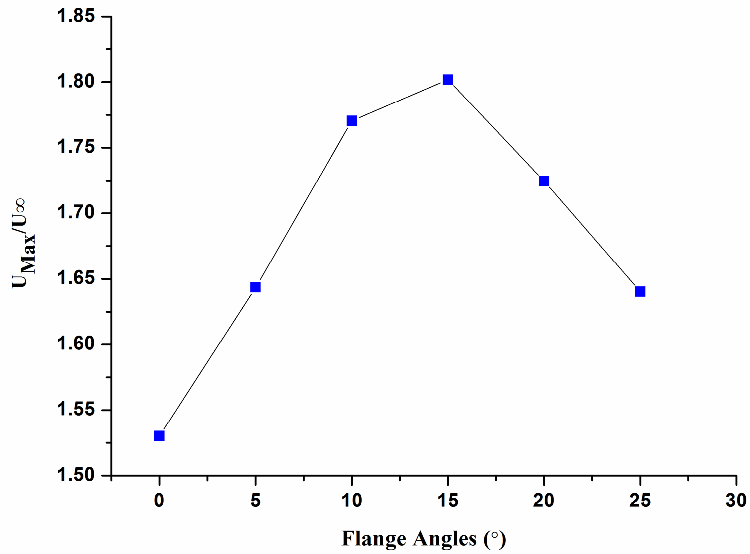


Fig. 5. Comparison of velocity increase at diffuser entrance at various flange angles.

3. Conclusion

Present study is carried out using ANSYS CFD tool for simulation of flow characteristics in circular diffuser with varying flange angles. Following outcomes could be derived from the present analysis-

1) Insertion of flange results in the formation of strong vortices on the downstream of flange, which in turn results in the increase velocity through the entrance of the wind turbine.
2) Maximum entrance velocity is recorded for flange angle of 15° ($U_{max}/U\infty$= 1.8).
3) After flange angle of 15°, the static pressure in the exit area of the diffuser decreases gradually and hence the entrance velocity decreases.
4) Present study shows that the speed of the wind turbine blades could be increased by providing wind lens, which in turn enhances in overall efficiency of the conventional wind turbine.

References

1. Yuji Ohya, Takashi Karasudani, Akira Sakurai, Ken-ichi Abe, Masahiro Inou. Journal of Wind Engineering and Industrial Aerodynamics. 96 524–539 (2008).
2. Yuji Ohya, Takashi Karasudani. Energies. 3, 634-649 (2010).
3. Kazuhiko Toshimitsu, Hironori Kikugawa, Kohei Sato, Takuya Sato. Open Journal of Fluid Dynamics. 2, 215-221 (2012).
4. Buyung Kosasih, Andrea Tondelli. Evolving Energy-IEF International Energy Congress (IEF-IEC2012), (2012), Procedia Engineering. Jan 1;49:92-8.
5. R. Bontempo, M. Manna. Applied Energy. 136, 405–416 (2014).
6. S.A.H.Jafari, B.Kosasih. Journal of Wind Engineering and Industrial Aerodynamics. 125, 102-110 (2014).
7. Wanlong Han, Peigang Yan, Wanjin Han, Yurong He. Energy. 89, 687-701 (2015).
8. Aly M. El-Zahaby, A.E. Kabeel , S.S. Elsayed, M.F. Obiaa. Alexandria Engineering Journal. 56, 171–179 (2017).
9. Hasim A. Heikal, Osayed S.M. Abu-Elyazeed, Mohamed A.A. Nawar, Youssef A. Attai, Maged M.S. Mohamed. Renewable Energy. (2018)-In Press.