Wind and Structures, *Vol. 32, No. 1 (2021) 47-53*
DOI: https://doi.org/10.12989/was.2021.32.1.47

A novel high performance diffuser design for small DAWT's by using a blunt trailing edge airfoil

Arturo Alanís[1a], Jesus Alejandro Franco[*1,2], Saul Piedra[3b] and Juan Carlos Jauregui[4]

[1]CIDESI-Centro de Ingeniería y Desarrollo Industrial, Av. Pie de la Cuesta No. 702, Desarrollo San Pablo, 76125, Qro, México
[2]Escuela Nacional de Estudios Superiores Juriquilla, UNAM, Queretaro 76230, Mexico
[3]CONACYT-CIDESI- Centro Nacional de Tecnologías Aeronáuticas Carretera Estatal 200 Querétaro-Tequisquiapan Km. 23, No. 22547, Colón, Qro. México
[4]Universidad Autonoma de Queretaro, Cerro de las Campanas S/N, Queretaro, Qro. México

(*Received October 22, 2019, Revised November 27, 2020, Accepted January 22, 2021*)

Abstract. This paper proposes a novel diffuser design for Diffuser Augmented Wind Turbines (DAWT) based on the blunt trailing edge airfoil AF300. Computational Fluid Dynamics (CFD) simulations are carried out to measure the performance of the AF300 diffuser against diffusers made with the shape of other high performance low wind speed airfoils. The results show that the proposed diffuser produces a greater air mass flow increase through the plane of the turbine than the other diffusers and it can be used to increase the performance of a horizontal axis wind turbine.

Keywords: DAWT; high performance airfoil diffuser; small wind turbines; CFD simulation

1. Introduction

Wind energy is currently considered as one of the most important technologies for electricity generation, especially when we take into consideration economic and environmental circumstances. Globally, the wind power industry is in an age of substantial growth as a result of advances in technology and is set to expand as the world looks for cleaner and more sustainable ways to generate electricity.

The increase in power demand, along with the depletion of fossil fuels and the environmental pollution, have made urgent the need to increase the performance of the wind turbines. Diffuser Augmented Wind Turbines (DAWT) have been proposed by numerous scientists as a way of achieving this power increase. This power increase has been validated for new mathematical models developed for different authors (Liu and Yoshida 2015, Vaz and Wood 2018). These models based on an extension of the Blade Element Theory are able to analyze the performance of DAWT's including diffuser efficiency, axial induction and thrust.

The mechanism behind the DAWT phenomenon is that the presence of the diffuser creates a negative pressure zone downwind of the turbine that increases the air mass flow through the blades of the turbine, thus increasing the power output (van Bussel 2007). DAWT's have shown to produce lower noise, lower visual pollution and to have a lower turbulence sensitivity than the bare wind turbines (Jafari and Kosasih 2014, Ohyad and Karasudani 2010). This, along with the increase in air mass flow through the turbine make them suitable for turbulent and low wind speed environments such as urban areas.

There are mainly three types of diffuser for DAWT's: 1) conical diffusers, 2) flanged diffusers, and 3) airfoil shaped diffusers (Fig. 1). Conical diffusers are the simplest kind of diffusers as they depend only on the length of the wall and its angle, while flanged diffusers are the most complex due to having more parameters, such as the wall shape, its angle, the flange height and the flange angle (El-Zahaby *et al*. 2017). Airfoil diffusers are of great interest to the authors because there is a wide variety of airfoils to choose from and there is a lot of information available publicly on their performance under different wind conditions (Selig 2017). Moreover, there's evidence that shows that both, the inner and outer geometries of the diffuser impact the performance of the system, giving another reason for using airfoils for the geometry of the diffuser (Aranake *et al*. 2013). The geometry of the airfoils deflects the incoming wind into two currents, one that flows over the extrados and another that flows over the intrados. Such deflection is responsible for promoting a low pressure zone over the extrados and a high pressure zone over the intrados. This pressure distribution is responsible for the Lift force generation. A higher lift coefficient C_L implies a large pressure difference between extrados and intrados surfaces. Unlike aircraft wings, the inner surface of the airfoil diffusers corresponds to the low pressure surface (extrados), so that the airflow around the inner surface is accelerated in comparison to the free stream velocity, thus increasing the air mass flow inside of the diffuser (Aranake *et al*. 2015). Furthermore, as the drag coefficient is higher there is a

*Corresponding author, Ph.D. Research Fellow
E-mail: alejandro.francop@unam.mx
[a]Ph.D. Student
E-mail: a.alanis@posgrado.cidesi.edu.mx
[b]Ph.D. Professor
E-mail: saul.piedra@cidesi.edu.mx

http://www.techno-press.com/journals/was&subpage=7
ISSN: 1226-6116 (Print), 1598-6225 (Online)

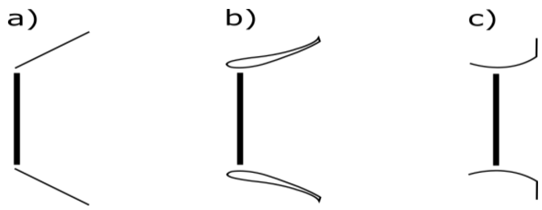

Fig. 1 Wind turbine diffuser types. (a) Conical diffuser (b) Airfoil diffuser (c) Flanged

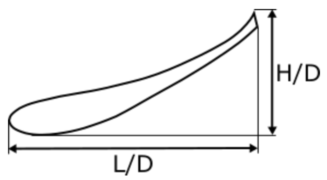

Fig. 2 Wind turbine diffuser parameters

greater resistance to the horizontal component of the wind flow, which in turn decreases the mass flow inside of the diffuser. Thus, a higher aerodynamic performance C_L/C_D will result in a larger mass flow augmentation ratio when and airfoil is used as diffuser geometry.

Additionally, airfoil diffusers are simple to work with because they depend only on the angle of attack α and the chord length. According to this, a method to design a diffuser augmented wind turbine (DAWT) using as a guiding point the optimal pressure drop at the turbine is proposed by Sorribes-Palmer *et al*. (2017). The proposed configuration can extract energy with the same efficiency as the state-of-the-art shrouded wind turbine, however, this design could generate smaller wake than other similar wind turbine systems.

Airfoil diffusers for low wind speed conditions typically make use of high-performance low Re airfoils, such as the S1210, S1223, S2091, SD7034 and SG6043 (D'Angelo 1989). However, the authors are interested in blunt trailing edge airfoils due to its advantages such as manufacturing ease, higher mechanical resistance and more important the increase in C_L with respect to their sharp trailing edge counterparts (Van Dam *et al*. 2008).

There are two ways to create flatback trailing edge airfoils, the first one is to trim the back of the airfoil. The second method consists in increasing the trailing edge thickness in a percentage of the chord (Singh *et al*. 2012). The latter is more accepted because it keeps the airfoil parameters such as the camber and max thickness intact, while the first alters these values by shortening the chord.

Singh *et al*. (2012) created a new blunt trailing edge by increasing the trailing edge of the S1210 airfoil by 3% of the chord. This investigation shows that this airfoil outperforms the C_L and the C_L/C_D of other low speed airfoils, while delaying the stall angle. These characteristics are desirable in DAWT's as it may increase the wind speed range of operation of the diffuser.

This paper presents the performance effects of using the AF300 diffuser in a DAWT by using CFD software together with the Actuator Disk method (AD) to simulate the presence of the wind turbine in a 3D environment.

2. Methodology

2.1 Airfoil validation

A first study was performed to validate that the AF300 airfoil truly outperforms other high performance low Re airfoils that are identified in the literature as the best performing (Singh *et al*. 2012). The comparison was carried out using the so-called panel method implemented in QBlade® software. The airfoils used in the study were the S1210 and S1223. These airfoils were gathered from the available literature for being identified as the best performing low wind speed airfoils (Salgado *et al*. 2016).

2.2 Angle selection for the modeling of the diffuser

The angle of attack for the transversal section of the diffuser was selected based on the parameters that give the best performance results in studies performed by other authors (Jafari and Kosasih 2014, Ohyad and Karasudani 2010, Aranake and Lakshminarayan 2014). This work identified that although these studies use different types of diffusers (conical diffusers and flanged diffusers) and different ways of parameterizing the diffuser shapes, if standardized to Jafari's parametrization, both studies show that the mass flow increase is achieved with a relation between the diameter D of the wind turbine and the height H of the diffuser H/D between 0.075 and 0.125, while the best performance increase is achieved with an H/D=0.1. The other parameter that affects the performance is the relation between the horizontal length of the diffuser and the diameter L/D. Both studies agree that the longer the length, the better performance increases, however, for manufacturing, transportation and installation ease, the author selected the L/D ratio to be 0.3 which is long enough for a good performance increase and is sufficiently small to be practical (Fig. 2).

This parametrization is then applied to airfoils, which dimensions are usually stablished by an angle of attack and the length of the chord but for this purpose, using the angle alone would increase the height of the diffuser while decreasing its horizontal length. Thus, a model was performed to maintain the horizontal length constant while varying both the angle and the chord length until the desired H/D and L/D ratios were met.

As H/D=0.1 was identified as the best performing ratio, the three airfoils angles and chords were sized to meet it. The resulting angles of attack are 14.6° for the AF300 with a chord of 0.3104 m, 15.6° for the S1210 with a chord of 0.3131 m and 14.5° for the S1223 with a chord of 0.3114 m.

2.3 Model Implementation

2.3.1 Actuator disk model

The main objective of this work is studying the effect on the wind velocity due to the presence of a proposed diffuser in order to improve the efficiency of a wind turbine. Since it is not reliable only to solve the flow around the diffuser, the turbine was included in the computational simulations. For

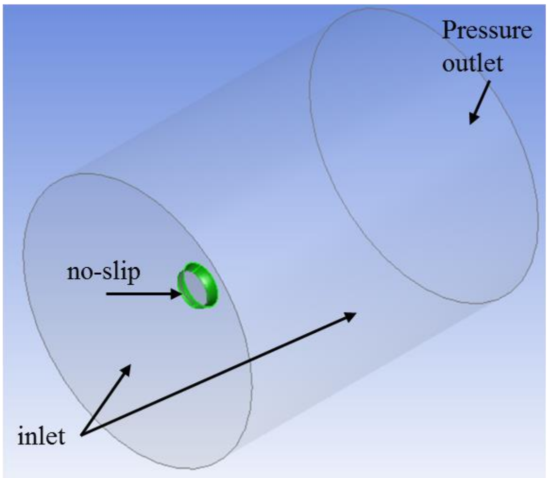

Fig. 3 Computational domain sketch and boundary conditions

this purpose, the Actuator Disk (AD) model was implemented to account for the wind turbine-diffuser system (Chaker *et al*. 2013). A hypothetical wind turbine with 1 m in diameter with almost the same features of the NREL Phase VI wind turbine was selected for the simulations. It is important to highlight that in a strict sense, the wind turbine selection is not relevant in the present study, since the analysis is focused on the diffuser performance and the presence of the turbine in the model is only necessary for completeness of the analysis. In the AD model the aerodynamic effect of the blades rotation motion is modeled by a pressure discontinuity through a thin disk with an area equal to the rotor swept surface of the wind turbine. The disk is modeled as a porous medium that allows the flow to pass through the rotor. The pressure in the front face of the disk is larger than that in the back face, however, the velocity evolves continuously throughout the thin disk. The complete theory of the actuator disk model can be found in Javaherchi (2010). In the present work, the implementation of the AD model was performed in ANSYS Fluent. The flow in the porous medium can be computed using a simplified momentum conservation equation that combines the classical Darcy's law and additional inertial loss term:

$$\Delta p = \left(\frac{\mu}{k} v + \frac{c_2}{2} \rho v^2\right) \Delta m \quad (1)$$

where μ is viscosity, k is permeability through the thickness of the porous disk, c_2 is the pressure jump coefficient, v is flow velocity normal to the porous face, ρ is density and Δm is the thickness of the media. As can be observed, the permeability and pressure jump must be computed in order to calculate the pressure drop through the porous media. The same strategy presented by Javaherchi (2010) was followed in this work for calculating both coefficients. The pressure drop as function of the velocity at the turbine plane was calculated using the AD model equations, then a second order polynomial fit was done to compute the pressure jump and permeability coefficients (Hartwanger and Horvat 2008). Finally, those coefficients were incorporated into the porous zone model in ANSYS Fluent to complete the CFD implementation for the wind turbine-diffuser system.

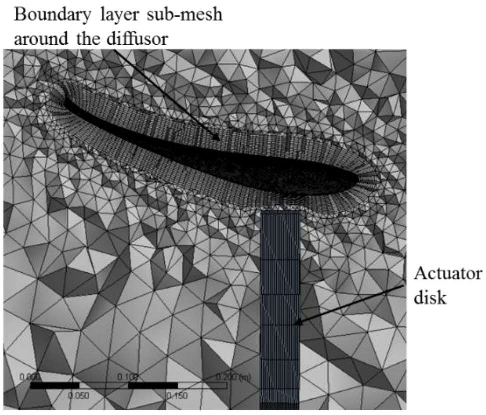

Fig. 4 Mesh refinement for the porous zone (actuator disk) and around the diffuser

2.3.2 CFD Implementation

Computational simulations with a full flow solver were implemented in order to analyze the flow and the aerodynamic performance of the wind turbine-diffuser system. In this score, the industry relies on numerical simulations that solve turbulent flows utilizing the Reynolds Averaged Navier-Stokes (RANS) equations (Spalart and Venkatakrishnan 2016). This section describes the implementation of steady-state RANS simulations for the flow around and throughout the wind turbine-diffuser system. The Shear Stress Transport k-ω turbulent model was used to carry out the simulations (Menter *et al*. 2009). The governing equations includes mass and momentum conservation equations to compute the averaged pressure and velocity fields in the whole domain and transport equations must be solved for the specific dissipation rate (ω) and turbulent kinetic energy (k).

A tetrahedral consistent mesh was constructed for each one of the three cases (AF300, S1210 and S1230 diffusers). In Fig. 4, a slice of the computational domain is shown to visualize the mesh close to the diffuser and actuator disk. Appropriate refinement was applied on the zones close to diffuser wall to compute accurately the flow at those zones. In addition, a sub-mesh 16 cells thick and aligned with the diffuser surface was constructed to capture boundary layer effects. A separated mesh was built for the porous zone that corresponds to the actuator disk. From the last considerations and the sensitivity analysis we obtained a computational mesh with around 6 million of control volumes. Since the mesh was developed to implement RANS simulations with wall functions, the value of y+ was fixed to 80, it was computed based on the freestream velocity to keep the approximation at the logarithmic region of the boundary layer.

3. Results and discussion

In the previous section, the mathematical model and the computational simulation methodology were presented. As

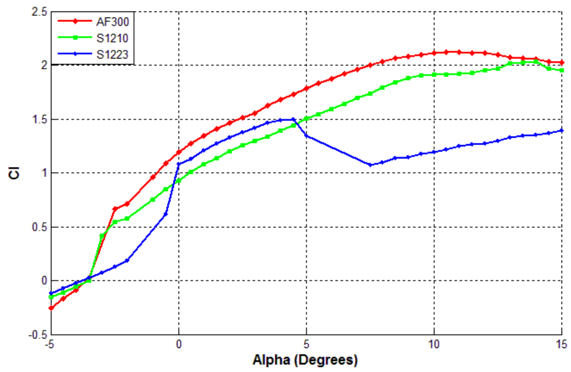

Fig. 5 C_L vs Alpha (α°) comparison between AF300, S1210 and S1223 airfoil

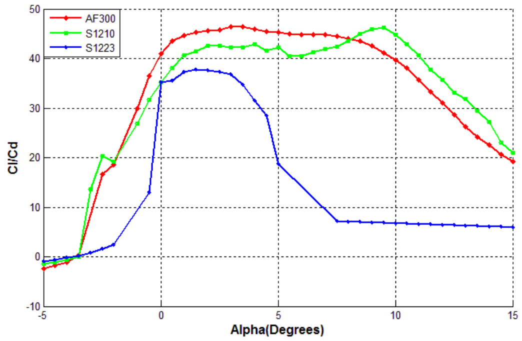

Fig. 6 C_L/C_D vs Alpha (α°) comparison between AF300, S1210 and S1223 airfoil

a result, in this section the comparison of the airfoil and diffuser performance are discussed. The results were obtained using QBlade® software and the C_L and C_D data were analyzed to compare the aerodynamic performance of the three selected airfoils. Afterwards, the three diffuser designs are evaluated through fully three dimensional simulations.

3.1 Airfoil performance analysis

An aerodynamic performance comparison was performed between the S1210, S1223 and the AF300 airfoils for a low Reynolds equal to 72,500. The performance comparison was developed using the software QBlade, which implements the panel method used in Xfoil.

Fig. 5 shows the C_L coefficient of the three airfoils with respect a range of α (0° to 15°). It is possible to observe AF300 airfoil maintains a higher C_L coefficient than the S1210 and S1223. As the C_L is directly related to the pressure distribution around the airfoil, it makes sense to assume that a diffuser built after the shape of an airfoil with the low pressure side as the interior of the diffuser will provoke a low pressure zone inside of the diffuser as the air stream flow over the diffuser. This, in turn, will increase the air mass flow through the wind turbine and thus augment the power output (Aranake *et al*. 2015). As the aerodynamic drag is a force that opposes to the airflow it is desirable to minimize it. In this way, achieving the highest lift to drag

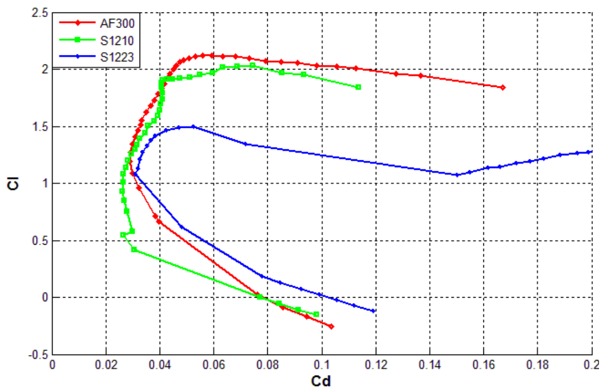

Fig. 7 C_L vs C_D, comparison between AF300, S1210 and S1223 airfoil

ratio will result in a better power augmentation ratio.

Fig. 6 reveals a comparison of the aerodynamic efficiency between the three studied airfoils with respect of the C_L/C_D ratio under a range of angles of α; meanwhile Fig. 7 plots the C_L vs C_D respectively. According to these Figures, it can be evidently seen that the AF300 shows a better aerodynamic performance compared with the other two airfoils, which infers that its implementation in the design of new diffusers improves performance and increases the efficiency percentage of diffuser-augmented wind turbine (DAWT) systems.

3.2 Diffuser performance comparison through fully three dimensional simulations

Wind turbine-diffuser simulations were performed for the three different diffusers using the actuator disc model. With the purpose of comparing qualitatively the flow close to the diffuser is shown in Fig. 8 for each one of the wind turbine-diffuser systems. For the three cases, flow separation can be observed in the trailing edge of the diffuser profiles. However, for the AF300 diffuser, the flow separation zone is weaker than for the S1210 and S1230 diffuser. Then, it can be inferred that less energy is loosed due to viscous effects in the wake behind of the diffuser and more available energy can be harnessed by the wind turbine.

In Fig. 9, the pressure distribution at the surface of the AF300 diffuser and the streamlines of the flow are shown. It is well-known that the energy available in the wind is proportional to the cubic wind speed, therefore, the analysis if the DAWT system can be done in terms of how the velocity changes due to the presence of the diffuser. As it was mentioned, the suction side of the airfoil corresponds to the inner surface of the diffuser, due to this configuration the flow is accelerated the flow inside the diffuser. This effect can be qualitatively appreciated by plotting the streamlines of the flow from which can be observed that the velocity close the diffuser increases as a consequence of the ducted effect promoted by the diffuser-augmented wind turbine (DAWT) system.

In order to establish a reference case, a CFD simulation of the actuator disk without diffuser was implemented. Fig.

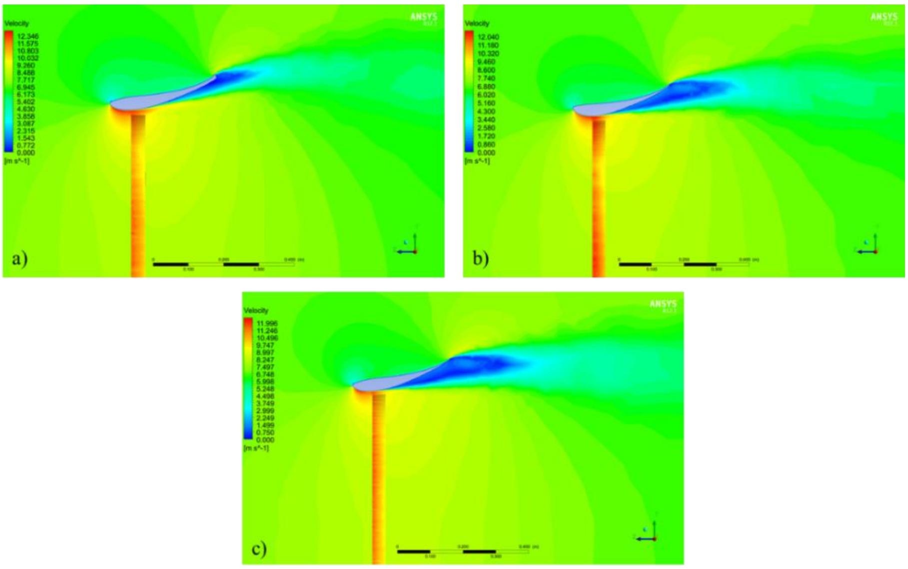

Fig. 8 The isocontours of a plane of the velocity field close to the actuator disk, a) AF300 diffuser, b) S1223 diffuser and c) S1210 diffuser, U_∞=7 m/s

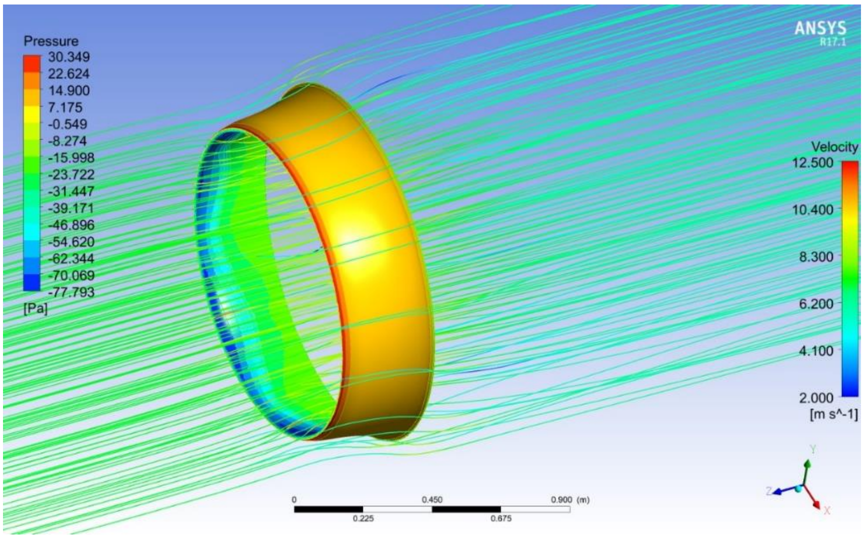

Fig. 9 Pressure distribution at the surface of the diffuser and streamlines of the flow for the AF300 diffuser (U_∞=7 m/s)

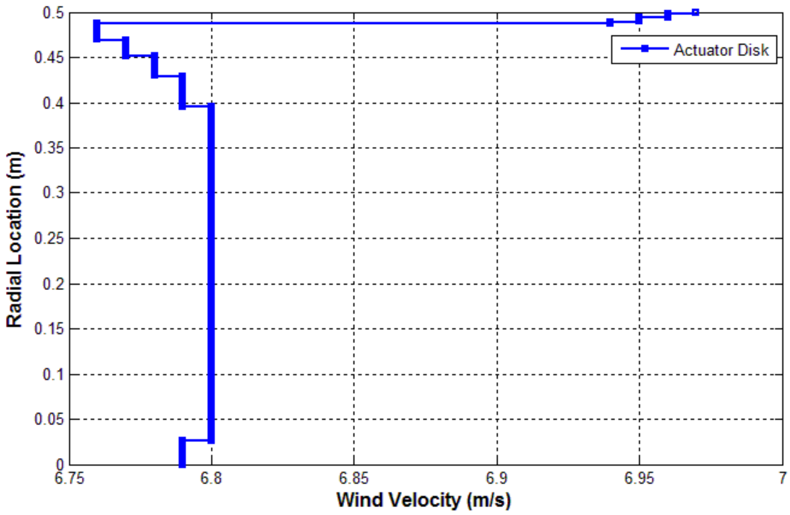

Fig. 10 Radial Location vs Wind Velocity. Actuator Disk without Diffuser (U_∞=7 m/s)

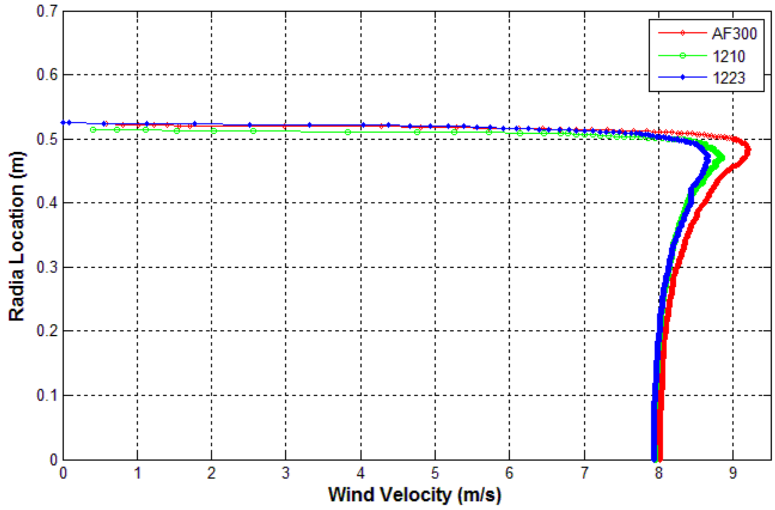

Fig. 11 Comparison between AF300, S1210, and S1223 wind velocity profile

Table 1 Kinetic energy ΔE_k comparison between AF300, S1210 and S1223

Diffuser Model	ΔE_k	%
AF300	7.1	49
S1210	6.2	41.8
S1223	6.1	41.1

10 describes a diagram where the *x* axis plots the wind velocity and the *y* axis represents the radial location of the actuator disk. According to the Figure is possible to obtain an average of 6.8 m/s through the vertical axis of the actuator disk.

A quantitative comparison of the performance of the three diffuser models was evaluated by calculating the increment of the captured kinetic energy in comparison with that without a diffuser ΔE_k captured in each designed diffuser Eq. (2).

$$\Delta E_k = \frac{1}{2}\rho \int_0^r ({v_D}^2 - {v_{AD}}^2)\, dr \qquad (2)$$

where ρ is the air density, r is the radius of the diffuser, v_D is the wind velocity for each DAWT design and v_{AD} is the wind velocity for the actuator disk without diffuser.

Table 1 shows the values for ΔE_k for each diffuser design and the percentage of the augmentation effect of the diffuser in terms of ΔE_k. According with these results the new diffuser design based on the AF300 airfoil proposed in this work showed a higher increment in the energy captured by the DAWT system of the 49 % with respect to the actuator disk without diffuser.

As a final result, Table 1 also shows a significant increment in the percentage of the augmentation effect due to the presence of the AF300 diffuser by about 7% in comparison with the S1210 and S1230 diffusers.

4. Conclusions

- This work identified a promising blunt trailing edge airfoil for improving the performance of DAWT's. In a comparative study of the aerodynamic performance under low Reynolds number condition, the AF300 airfoil presented a better aerodynamic efficiency and a higher C_L coefficient compared to the S1210 and S1223 airfoils. These results aim to increment the augmentation diffuser effect if the AF300 airfoil is implemented.
- CFD simulations of the DAWT system were implemented using the actuator disk model to take into account the wind blockage effect of a wind turbine. The flow around the AF300 diffuser exhibits a smaller boundary layer separation with respect to the other two diffusers. This indicates less energy losses in the wake behind the AF300 diffuser and then, more energy is available to be harnessed by the wind turbine.
- From the quantitative point of view, it was found that the flow increment achieved due to the presence of the AF300 diffuser is larger than the increment due to the S1210 and S1230 diffusers. Table 1 showed an increment in the energy captured in the DAWT system of the 49 % with respect to the actuator disk without diffuser.
- This work demonstrated the potential of blunt trailing edge airfoils for diffuser augmented wind turbines as they clearly have performance advantages with respect to sharp trailing edge airfoils.
- The results presented in this work open a new line of investigation for exploring different geometries for blunt trailing edge airfoils specially designed for DAWT applications.
- The authors also noticed that there is a similarity between the trailing edge of the AF300 airfoil and the brim of the flanged diffuser designs, which work by creating vortex structures behind the flange. It could be interesting to study the power augmentation performance of blunt trailing edge airfoils with a flange to see if the combining the effects of both designs can be achieved an even larger power augmentation.
- As a future work, we suggest performed experimental wind tunnel tests. These additional experimental tests

will be relevant for additional research of the results presented in this paper.

Acknowledgments

The authors acknowledge the support of the Mexican Council for Science and Technology (CONACYT). S. Piedra would like to thank the Catedras program from CONACYT Mexico.

References

Aranake, A., Lakshminarayan, V. and Duraisamy, K. (2013), "Computational Analysis of Shrouded Wind Turbine Configurations", *The 51st AIAA Aerosp. Sci. Meet. Incl. New Horizons Forum Aerosp. Expo.*, 1-17. https://doi.org/10.2514/6.2013-1211.

Aranake, A.C. and Lakshminarayan, V. K. (2014), "Assessment of low-order theories for analysis and design of shrouded wind turbines using CFD", *J. Phys.: Conference Series*, **524**. https://doi.org/10.1088/1742-6596/524/1/012077.

Aranake, A., Lakshminarayan, V. and Duraisamy, K. (2015), "Computational analysis of shrouded wind turbine configurations using a 3-dimensional RANS solver", *Renew. Energy*, **75**, 818-832. https://doi.org/10.1016/j.renene.2014.10.049.

Chaker, R., Kardous, M., Aloui, F. and Nasrallah, S.B. (2013), "Open angle effects on the aerodynamic performances of a flanged Diffuser Augmented Wind Turbine (DAWT)", *Conférence Internationale des Energies Renouvelables (CIER'13) Sousse*, Tunisie.

D'Angelo, M.R. (1989), *The effects of trailing edge bluntness on airfoil performance as calculated approximately by a viscid-inviscid vortex panel method*, Master Dissertation, Massachussetts Institute of technology.

El-Zahaby Aly, M., Kabeel, A.E., Elsayed, S.S. and Obiaa, M.F. (2017), "CFD analysis of flow fields for shrouded wind turbine's diffuser model with different flange angles", *Alexandria Eng. J.*, **56**, 171-179 https://doi.org/10.1016/j.aej.2016.08.036.

Hartwanger, D. and Horvat, A. (2008), "3D Modelling of a Wind Turbine Using CFD", *In Proceedings of NAFEMS Conference*; NAFEMS: Cheltenham, U.K.

Jafari, S.A. and Kosasih, B. (2014), "Flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations", *Jnl. Wind Eng. Ind. Aerod.*, **125**, 102-110. https://doi.org/10.1016/j.jweia.2013.12.001.

Javaherchi-Mozafari, A.T. (2010), *Numerical Modeling of Tidal Turbines: Methodology Development and Potential Physical Environmental Effects*, Master Dissertation, University of Washington.

Liu, Y. and Yoshida, S., (2015), "An extension of the generalized actuator disc theory for aerodynamic analysis of the diffuser-augmented wind turbines", *Energy* **93** 1852-1859 https://doi.org/10.1016/j.energy.2015.09.114.

Menter, F.R. (2009), "Review of the shear-stress transport turbulence model experience from an industrial perspective", *Int J. Comput. Fluid D.,* **23**, 305-316 https://doi.org/10.1080/10618560902773387.

Ohya, Y. and Karasudani, T. (2010), "A shrouded wind turbine generating high output power with wind-lens technology", *Energies*, **3** (4) 634-649. https://doi.org/10.3390/en3040634.

Salgado, V., Troya, C., Moreno, G. and Molina, J. (2016), "Airfoil selection methodology for small wind turbines", *Int. J. Renew. Energy Res.*, **6**(4), 1410. https://www.ijrer.org/ijrer/index.php/ijrer/article/view/4642/pdf.

Selig, M.S. (2017), *UIUC Airfoil Data Site", UIUC Applied Aerodynamics Group*. http://m-selig.ae.illinois.edu/index.html.

Singh, R.K., Ahmed, M.R., Zullah, M.A. and Lee, Y.H. (2012), "Design of a low reynolds number airfoil for small horizontal axis wind turbines", *Renew. Energy*, **42**, 66-76. https://doi/org/10.3390/en3040634.

Sorribes-Palmer, F., Sanz-Andres, A., Ayuso, L., Sant, R. and Franchini, S. (2017), "Mixed CFD-1D wind turbine diffuser design optimization", *Renew. Energy* **105** 386-399 https://doi.org/10.1016/j.renene.2016.12.065.

Spalart, P.R. and Venkatakrishnan, V. (2016), "On the role and challenges of CFD in the aerospace industry", *Aeronaut. J.*, **120**, 209-232, https://doi.org/10.1017/aer.2015.10.

van Bussel, G.J.W. (2007), "The science of making more torque from wind: Diffuser experiments and theory revisited", *J. Phys. Conf. Ser.*, **75**(1-12), 12010. https://doi:10.1088/1742-6596/75/1/012010.

Mayda, E.A., Van Dam, C.P., Chao, D.D. and Berg, D.E. (2008), *Computational design and analysis of flatback airfoil wind tunnel experiment* (No. SAND2008-1782). Sandia National Laboratories.

Vaz J.R.P. and Wood, D.H. (2018), "Effect of the diffuser efficiency on wind turbine performance", *Renew. Energy,* **126** 969-977. https://doi.org/10.1016/j.renene.2018.04.013.

TK