Proceedings of the ENCIT 2012
14th Brazilian Congress of Thermal Sciences and Engineering
November 18-22, 2012, Rio de Janeiro, RJ, Brazil

EXPERIMENTAL AND NUMERICAL STUDY OF THE VELOCITY PROFILE IN A CONICAL DIFFUSER AIMING THE EFFICIENT DESIGN OF HORIZONTAL AXIS WIND TURBINE

Sávio Weslley Oliveira Figueiredo, savio.figueiredo@itec.ufpa.br
Arielly Assunção Pereira, arielly@ufpa.br
Erb Ferreira Lins, erb@ufpa.br
Marcelo de Oliveira Silva, mos@ufpa.br
Jerson Rogério Pinheiro Vaz, jerson@ufpa.br
Universidade Federal do Pará, Rua Augusto Corrêa, 01, Guamá, Belém, Pará, Brasil, CEP 66075-110

Abstract. *Diffusers are widely used in the horizontal axis wind turbines design, since the influence of the diffuser causes a considerable increase in the efficiency of the wind turbines. This increase is due to the low pressure region that is formed after the passage of flow through the diffuser, which increases the mass flow. At the point of maximum velocity, the rotor can extract more kinetic energy of the flow, increasing its efficiency. The diffuser geometry determines the point of maximum wind velocity, since its aspect ratio changes the shape of the velocity profile. This work presents an experimental and a numerical study of the flow around a conical diffuser, in order to obtain the velocity and pressure profiles along the axis of the diffuser. The computation of the velocity profile internally at the diffuser is done with the finite volume method and this data is compared with experimental data.*

Keywords: *diffuser, wind turbine, renewable energy*

1. INTRODUCTION

The application of conical diffusers employed in conventional piping and hydraulic systems, where the flow is internal, in general aims at transforming the kinetic energy of the fluid into pressure energy. However, the type of conical diffuser treated in this article causes a different effect on the flow. The presence of external flow causes an acceleration of the fluid particles towards the exit of the diffuser. This effect occurs due to the vortex wake in the outlet zone of the diffuser. Therefore, this device application in the horizontal axis wind turbines design becomes very interesting.

The diffuser creates a suction region behind its structure, increasing mass flow and the available kinetic energy. An interesting application is the use of diffusers in wind turbines, since in this case the Betz's limit (1926) is exceeded. In the case of free-flow turbines, the Betz limit is 59.26% (Rio Vaz, 2011). As reported in the work of Ohya and Karasudani (2010), the increment caused by diffuser may cause a wind turbine to achieve a power coefficient between 4 and 5 times greater than free-flow turbine. In the present work, we show a numerical-experimental study of a conical diffuser, aiming the efficient horizontal axis wind turbines design. It is evaluated the effect caused by the diffuser on the velocity profiles and the static pressure in diffuser internal flow. The numerical results were obtained using ANSYS FLUENT software and compared with experimental data obtained in this work.

2. PERFORMANCE EVALUATION OF A CONICAL DIFFUSER

Abe and Ohya (2004) computed the acceleration of the internal flow of a flanged diffuser using Computational Fluid Dynamics. They chose to make measurements along the longitudinal axis of the diffuser, spanning the regions upstream, through the inlet region to the region downstream of the diffuser. The choice of taking measurements only in the regions belonging to the longitudinal axis is justified by the fact that the results of the simulations showed that the streamlines are smooth inside the diffuser, except in the boundary layer. The results obtained there achieve an excellent approximation with experimental data, justifying the considerations regarding the variation in radial velocity. Thus, the distributions along the longitudinal axis are essential for the diffuser performance computation. Similarly, in the present work, the diffuser will be characterized on the basis of quantities (flow velocity and static pressure) measured in the longitudinal axis (to simplify the notation, "axis" will be used in this paper).

2.1 Experimental measurements of velocity profile and static pressure

In order to evaluate the performance of the diffuser, measurements were made for velocity and static pressure of the jet flow in a wind tunnel with section $310mm$ x $310mm$. We used a Pitot tube to measure data along the diffuser axis, Fig. (1). The aspect ratio (r) of the diffuser is approximately 0.756, Eq. (1). Measurements were made for different fan speed at rotations of $400rpm$, $500rpm$, $600rpm$, $700rpm$ and $800rpm$, which results in different flow velocities.

$$r = \frac{L}{d} \tag{1}$$

Where L is the length of the diffuser and d is the diameter of the inlet. Both dimensions are shown in Fig. (1).

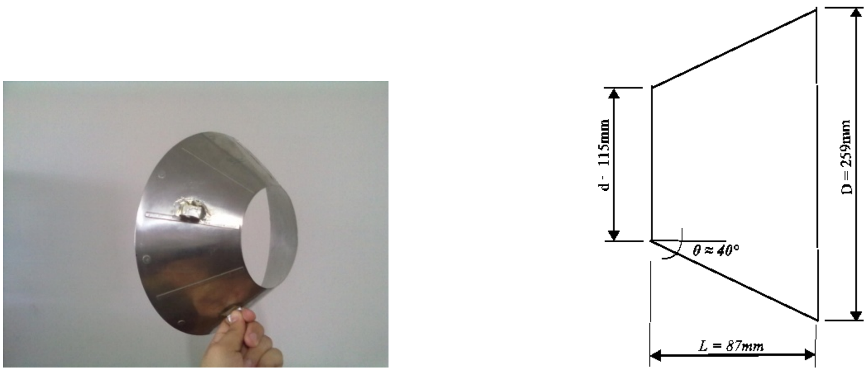

Figure 1. Diffuser and its dimensions and used in the experiment

The augmentation factor K and the pressure coefficient C_p were the quantities obtained from the collected data. The augmentation factor is given by Eq. (2).

$$K = \frac{U_i}{U_0} \tag{2}$$

where U_i is the velocity measured at a point i on the axis and U_0 is the free stream velocity.

The augmentation factor is the increase (or decrease) in flow velocity caused by the influence of the diffuser. This comparison is valid since a turbine without diffuser extracts energy from the air flow with the velocity U_0. Therefore, this parameter is used to compare the power extracted by a free-flow turbine with a turbine equipped with diffuser, since this power depends directly of the velocity. The power available in the flow is given by Eq. (3)

$$P = \frac{1}{2}\rho A U^3 \tag{3}$$

It can be noticed that this power is dependent on the third power of wind velocity. Therefore, any increase in value of the velocity also increases significantly this value. In Eq. (3) where ρ is the fluid density, A is the cross sectional area of the control volume considered and U is the stream velocity. The variation of the pressure coefficient C_p, given by Eq. (4), is important for indicating the pressure behavior at a point.

$$C_p = \frac{P_i - P_0}{\frac{1}{2}\rho U_0^2} \tag{4}$$

Where P_i is the static pressure on the axis, measured at the point i and P_0 is the free stream static pressure.

2.2 Numerical simulations

We considered the five experiments where in the wind tunnel velocity flow were 3.86, 5.06, 6.24, 7.39 and $8.51 m/s$, measured in experimental procedure. ANSYS-FLUENT software was employed to compute the flow around the diffuser. The grid considered in the study presents 51725 nodes (Fig. 2) and flow was treated as axisymmetric and steady-state. The turbulent model adopted was $k - \omega/SST$. The fluid inlet condition was set to 3% of free-flow intensity.

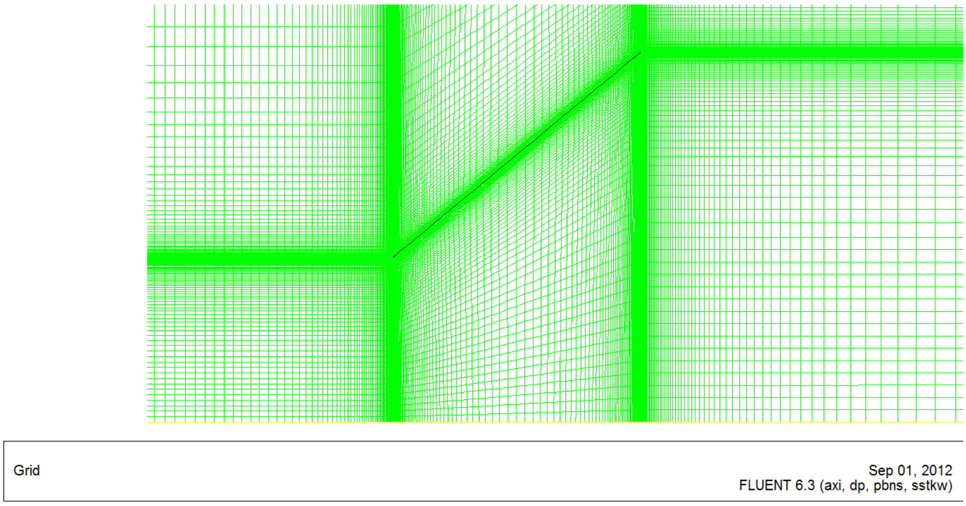

Figure 2. Grid used in the simulation.

3. RESULTS AND DISCUSSIONS

Figure (3) shows the velocity field (in m/s) around the diffuser. It is observed that velocity of the flow increases near the outlet due to the suction caused by the diffuser. Another important aspect is the formation of vortex contributes to the increased mass flow within the diffuser, according to Hansen et al. (2000), Abe and Ohya (2004), and Ohya Karasudani (2010).

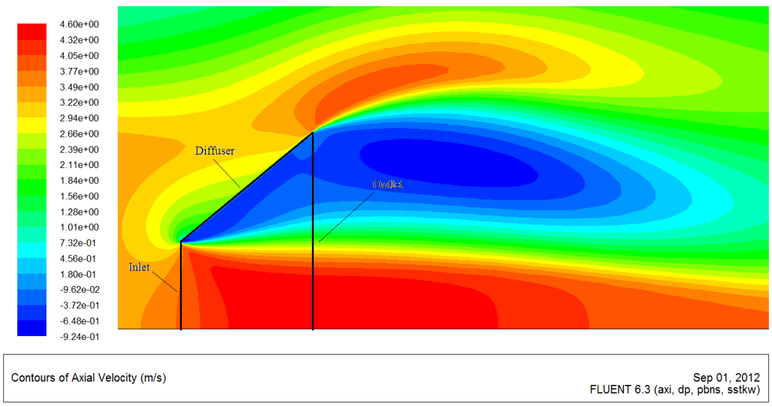

Figure 3. Velocity field around of the diffuser.

The profiles of flow velocity and static pressure (experimental and numerical) are shown in Figs. (4) and (5), for a velocity of $3.86 m/s$ in the wind tunnel. Only the results for velocity $3.86 m/s$ are shown, because the other results are very similar.

In Fig (4), the maximum augmentation factor (K) is approximately 1.22 and occurs at $340 mm$ point. Note that this point is outside the limits of the diffuser geometry from $220 mm$ to $310 mm$ (represented by the vertical dashed lines). This is due to the fact that the opening angle of the diffuser is $40°$. For smaller angles (around the $5°$) the point of maximum augmentation factor is located near the inlet (Abe and Ohya, 2004, Ohya and Karasudani, 2010). The installation of a

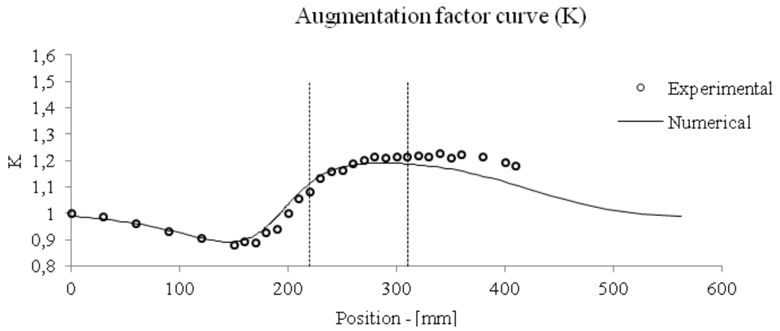

Figure 4. Comparison between data from experimental and numerical augmentation factor

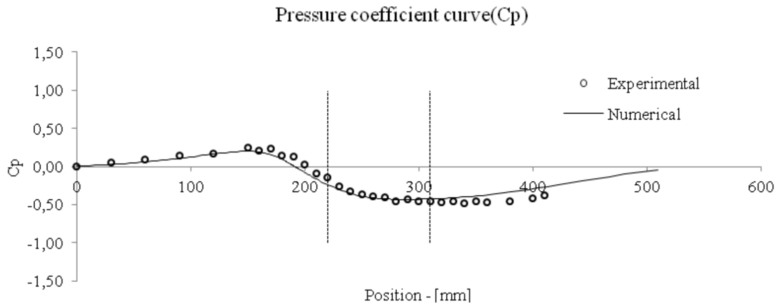

Figure 5. Comparison between data from experimental and numerical pressure coefficient

wind turbine must be made at the point of maximum augmentation factor (K), where the diffuser improves the energy potential at this point. However, due to the turbulent flow which arrives on the rotor is recommended placing the turbine at the outlet of the diffuser. The ideal position for the rotor is inside the diffuser and near the inlet (Fig. 3). The relative error between the experimental results and numerical at the point where K is around 5%. If a turbine is not equipped with the diffuser, the maximum avaliable power (P) in the fluid would be only $P_0 = \frac{1}{2}\rho A U_0^3$ whereas a turbine equipped with a diffuser would extract an output $P_K = \frac{1}{2}\rho A U_K^3$.

In Fig. (5), the values of the pressure coefficient (C_p) at the inlet region are greater than internal and outlet regions. The variation of the pressure coefficient confirms the presence of the suction phenomena. This observation demonstrates that the diffusers applied to wind turbines are different from conventional diffusers (where the value of C_p tends to increase in the direction of flow).

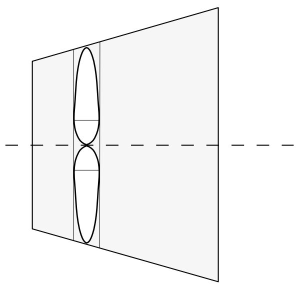

Figure 6. Ideal position of a turbine in a diffuser.

4. CONCLUSIONS

In this paper, a brief comparison of the fluid flow velocity in the internal region of a diffuser is done. The comparison between experimental and numerical results shows that the simulations are reliable for determining the behavior of the velocity profile and static pressure in diffusers. Due to the great similarity with the experimental results, the simulation in finite volume can be used to make predictions about the behavior of diffusers with similar geometries to study with other values of aspect ratio and opening angle. However, few adjustments are necessary in the $k - \omega/SST$ model parameters to further enhance the proximity between the numerical and experimental results.

The velocity profile is an important information in the study of a diffuser, as can indicate where one turbine should be placed inside the diffuser. As the external flow influences directly the internal flow in a diffuser, some studies aimed at observing these flow characteristics and the influence of the external geometry of the diffuser flow are necessary. The vortices generated at the outlet of the diffuser are important to induce low pressure in this region.

5. ACKNOWLEDGEMENTS

The authors thank to Pró-Reitoria de Pesquisa e Pós-Graduação – PROPESP (EDITAL PARD 04/2011), to Instituto Nacional de Ciência e Tecnologia de Energias Renováveis e Eficiência Energética da Amazônia - INCT – EREEA and to Conselho Nacional de Desenvolvimento Científico e Tecnológico – CNPq, for the granting of financial support.

6. REFERENCES

Abe, K., Ohya,Y., 2004, “Na Investigation of Flow Fields Around Flanged Diffusers using CFD”, Journal of Wind Engineering and Industrial Aerodynamics, Vol. 92, pp.315-330.

Betz, A., 1926, “Wind und ihreAusnutzungdurch Windmuehlen”.

Hansen, M. O. L; Sorensen, N. N. and Flay, R. G. J. ‘Effect of placing a diffuser around a wind turbine’, Wind Energy v. 3, pp. 207–213, 2000.

Ohya,Y., Karasudani T., 2010, “A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology”, Journal Energies, No 3, pp.634-649.

Rio Vaz, D. A. T. D. R., 2011, “Projeto de Rotores Hidrocinéticos de Eixo Horizontal Adaptados às Condições dos Rios Amazônicos”, Trabalho de conclusão de curso (Dissertação de mestrado em Engenharia Mecânica) - Faculdade de Engenharia Mecânica, Universidade Federal do Pará, Belém, Pará, Brasil, 116p.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.