24th COBEM - 2017

COBEM-2017-1907
COMPUTATIONAL STUDY OF THE WIND FLOW IN CONVERGENT-DIVERGENT NOZZLE

Francisco Carlos Lima de Sousa
Silmara Bispo do Santos
Rodrigo Sabino Pereira
KeteriPoliane Moraes de Oliveira
Universidade Federal de Mato Grosso, Instituto de Ciências Agrárias e Tecnológicas, Rondonópolis, Brazil
e-mail: syllmara@yahoo.com.br
e-mail: sabinorsp@gmail.com
e-mail: keteri@gmail.com

Abstract.*The search for renewable energy sources has been growing in the last decades due to the concern with energetic demand and environmental impact caused by fossil and nuclear energy source. The wind energy has been proved as a promising renewable source. It has been widelly studied for its inexhaustible source of energy and no greenhouse gases emission. The present study aimed at evaluating the different nozzle geometries convergent-divergent for increasing wind velocity at the wind turbines entrance.   For this study realization, computational fluid dynamics (CFD) technique was used . This technique allow us evaluating several parameters wich influence the increase in speed, as convergent-divergent angles and lenght relation at entrance and exit with the nozzle throat diameter. ANSYS Fluent was used for simulation and prediction of fluid flow velocity and pressure. The simulation was setup as a 2-D stationary, steady state, ideal gas analysis. The turbulence model used was SST k-ω method. The simulations were performed for free stream velocity of 4.5 m/s and for different shapes of convergent-divergent nozzle.The research indicated that one of the geometries among all the assessed nozzle, promoted a higher wind aceleration in the inner region of the nozzle. This geometry has an entry angle of 60°, throat length and diameter ratio of 1/8, being the divergent section an angle of 45º and length ratio of 1. In such conditions, the relation of increase in speed was 1.98.*

Keywords:*Nozzle, wind energy, CFD*

1. INTRODUCTION

The wind energy sector is booming worldwide and it is taking its first steps in Brazil, where the energy potential is almost twice the power currently installed in the country. For the wind energy could be considered technically useable at a height of 50 m, it is necessary that the minimum wind speed is from 7 to 8 m/s (Grubb and Meier, 1993).

Facing a climate change scenario where the increase in energy demand occurs simultaneously with the water resources reduction for hydroelectric generation, the feasibility of systems for micro wind generation in regions with lower wind speed should be better evaluated.

Adaptations to conventional horizontal turbines have been proposed over the last few years. The aim is increasing wind speed that reach the turbine and thus make possible to implement wind power generation systems in regions with low wind speed. (Foreman et al. (1978); Dick, (1986); Ohya et al. (2008); Wang et al. (2008); Takahashi et al. (2012); Mansour e Meskinkhoda, (2014); Jafari e Kosashi, (2014); Ahsan et al. (2015)).

The increase in flow velocity would be resulted from a low pressure zone created at the back of the diffuser. As the wind energy in an open air flow is proportional to the cube of wind speed, small increase in wind speed can mean large increase in the turbine shaft power. It means, if the wind speed is the double, it can reach eight times in wind power(Ahsan et al. 2015).

According to Dick (1986), the increase of a wind turbine power is result of power of two fundamental mechanisms: increase of mass flow and turbulent mixing with external flow.

Ohya et al. (2008) obtained good results 2008 obtained good wind acceleration results using a diffuser-type system with a flange located on the external exit session of the diffuser which contributes to the creation of a large region of flow separation at the diffuser downstream. The region of flow separation generated around the flange is a low pressure region that acelerate airflow in the internal diffusor area compared to a system with a flangeless diffuser. The

experimental reseach conducted by these systems has shown that a wind turbine coupled to a diffusor generates more energy when comparated to a same turbine without this device, with a power coefficient four times higher.(Abe et al. 2005).

Kosasih et al. (2012) investing three different diffuser shapes: straight diffuser, nozzle-diffuser combination, and diffuser-brim (brimmed diffuser) combination. Tests confirmed that placing micro wind turbine model inside a diffuser can substantially improve its performance. Straight diffuser improves the performance by 56% compared to the bare turbine whilst the nozzle-diffuser enhancement of 61% is slightly better than diffuser only.

The increase in speed depends on the settings at the entrance and exit area of these devices. It is interesting the use of computational fluid dynamics to identify the features that allow a higher increase on the power of wind. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows (Garcia et al. 2014). The use of CFD technique allow predicting the phenomena occurrence in a flow with no immediate need of prototype construction, significantly reducing costs on the experimental stage .

Based on these observations, this study aims to evaluate the wind flow in convergent-divergent nozzle and estimate by computational simulation, the velocities in axial direction along the nozlle .

2. COMPUTATIONAL PROCEDURE

The simulations were performed using ANSYS Fluent 17.0. The present computational conditions and grid system around a convergent-divergent nozzle are shown in Fig. 1a and Fig. 1b, respectively. In this study, the flow was assumed to be an axisymmetric steady flow, with x and r being the stream wise and radial coordinates, respectively. The number of cells in this study was 69721 e 76689. The computational domain was created so the flow around the body is not influenced by the wall effects related to the boundary layer region. The free stream speed of the flow was defined as 4.5 m/s. The Reynolds number was defined as VD/v, here V is the axial velocity in m/s, D is the diameter of the inlet nozzle and v is the kinematic viscosity of air at 25 degrees. The numerical simulations used were based on equation solutions in wich the flow was taken as incomprehensible and stationary. The turbulence model used was SSTK-w SST. The solution algorithm is SIMPLE. A convergence criterion of $1x10^{-4}$ has been adopted.
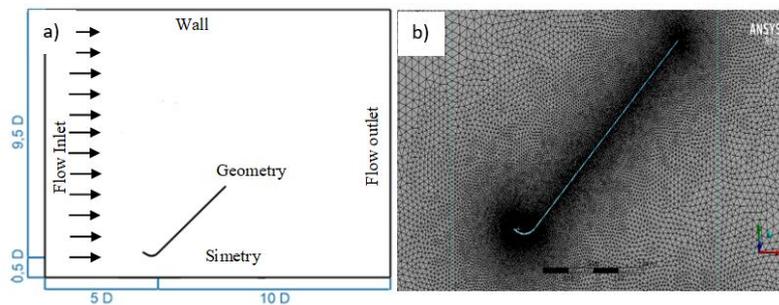

Figure 1. a) Computational design; b) Grid system and mesh

Initially, 24 cases were simulated and there was an opening angle variation at the entrance (α2), and the relation L 2/d (section length convergent/throat diameter). The combination of these parameters were evaluated and the linear momentum equation was used to estimate the theoretical wind power in each case. The figure 2 shows a general diagram of the studied nozzle and its variables. The effect of flange positioning was not studied in this stage.
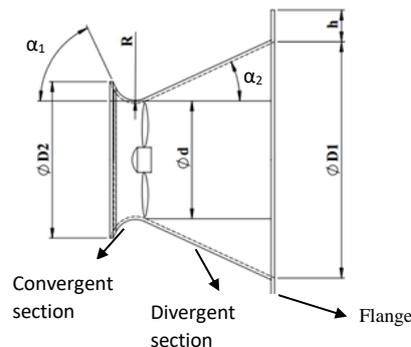

Figure 2 - Nozzle general model

Based on the numerical results by the simulations for each case, it was made a survey of the maximum speed values along the nozzle length.

The maximum velocities obtained were organized, also the distances in which that ones occurred along the nozzle length, so that an estimate of the theoretical wind potential could be obtained for each geometry. The theoretical wid power was calculated for two circunstances: a bare turbine and a turbine with nozzle by the E.q.1 and Eq. 2, respectively. The calculated powers were related to comparison with other reports in the literature.

$$P_b = \frac{1}{2}\rho A V_\infty^3 \tag{1}$$

$$P_n = \frac{1}{2}\rho A V_2^3 \tag{2}$$

Where, P_b is theoretical wind power (Watts) for turbine without nozzle; P_n is theoretical windpower (Watts) for turbine with nozzle; ρ is specific air mass (1,2 kg/m^3) and V is wind velocity at the rotor entrance (m/s); V_∞ is upstream wind velocity (m/s); A is the flow section area. The indices 1 e 2 indicate the section at the rotor entrance for abare turbine and turbine with nozzle.

Considering incompressible flow where the density is constant and a rotor of area A, the power ratio is:

$$\frac{P_n}{P_b} = \left(\frac{V_2}{V_1}\right)^3 \tag{3}$$

3. RESULTS AND DISCUSSION

Soon after the processing step the speed values and pressure were reported by Fluent in graphic form and text files so it was possible evaluate numerical results and the behavior of these properties for different geometries of the nozzle.

In table 1 the evaluated cases in this study are presented, so that the results obtained for maximum speed in the axial direction inside the nozzle and the position where these levels of speed were reached. It would indicate the position where the turbine could be installed for a better use of the wind power. In these positions, in evaluating the gradient speeds in radial direction, we could verify that there is little variation in levels of speed from the central axis up to the radial distance of 1R approximately, where R is the radio of the throat nozzle.

Table 1 - The convergent-divergent nozzle configurations tested by computational simulation and the effects of these configurations on the wind flow speed.

Cases	Concentrator		Diffuser		U_{max}	U_{max}/U_∞	Dintance X of the Throat	Pn/Pb
	L1/D	θ1	L2/D	θ2	(m/s)		(m)	
Case 1	1/2	30°	1	4°	4.63	1.03	0.26	1.09
Case 2	1/2	60°	1	4°	6.39	1.42	0.33	2.86
Case 3	1/8	30°	1	4°	4.98	1.11	0.25	1.37
Case 4	1/8	60°	1	4°	5.01	1.11	0.28	1.37
Case 5	1/2	30°	1.5	4°	4.77	1.06	0.28	1.19
Case 6	1/2	60°	1.5	4°	5.43	1.21	0.50	1.77
Case 7	1/8	30°	1.5	4°	3.10	0.69	1.00	0.33
Case 8	1/8	60°	1.5	4°	1.64	0.36	1.00	0.05
Case 9	1/2	30°	1	15°	6.02	1.34	0.46	2.41
Case 10	1/2	60°	1	15°	6.16	1.36	0.23	2.52
Case 11	1/8	30°	1	15°	5.09	1.13	0.20	1.44
Case 12	1/8	60°	1	15°	6.13	1.36	0.21	2.52
Case 13	1/2	30°	1.5	15°	6.38	1.42	0.33	2.86
Case 14	1/2	60°	1.5	15°	7.77	1.73	0.67	5.18
Case 15	1/8	30°	1.5	15°	6.02	1.34	0.18	2.41
Case 16	1/8	60°	1.5	15°	7.49	1.66	0.30	4.57
Case 17	1/2	30°	1	45°	5.75	1.28	1.00	2.10
Case 18	1/2	60°	1	45°	5.00	1.11	1.00	1.37
Case 19	1/8	30°	1	45°	6.83	1.52	0.63	3.51

Case 20	1/8	60°	1	45°	8.90	1.98	0.42	7.76
Case 21	1/2	30°	1.5	45°	6.91	1.53	0.60	3.58
Case 22	1/2	60°	1.5	45°	6.19	1.37	1.00	2.57
Case 23	1/8	30°	1.5	45°	7.79	1.73	1.00	5.18
Case 24	1/8	60°	1.5	45°	7.80	1.73	0.68	5.18

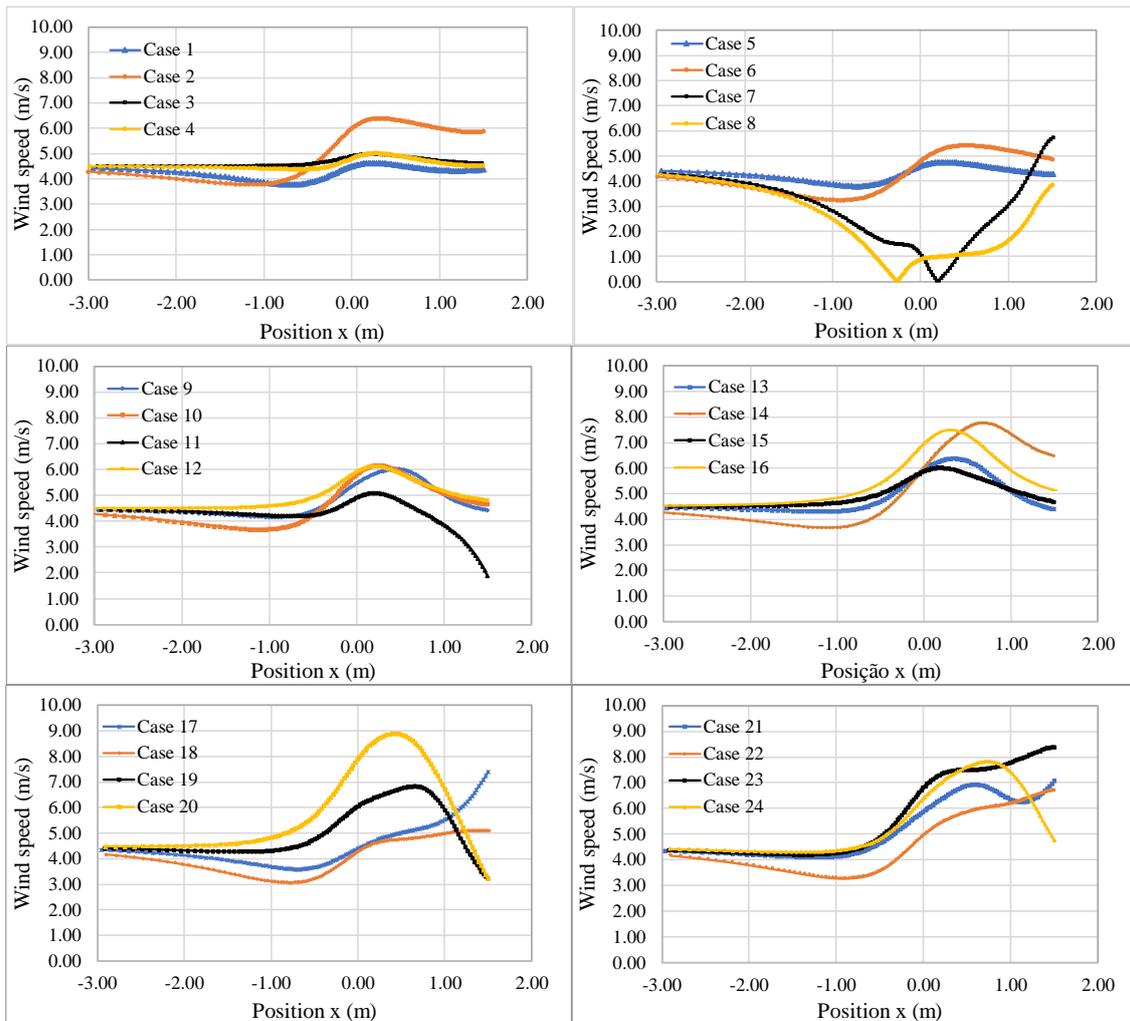In figure 3 are presented the numerical results of air flow velocity along the nozzle central axis.

Figure 3 - Air speed in different sections in flow direction obtained by CFC

3.1 Effect of the divergent section geometry

According to Figure 3 and table 2, in all the evaluated cases the nozzle promoted an increase in speed of the flow in any internal section, , except in cases 7 and 8. Keeping the divergence angle in 4° and the ratio L2/d in the nozzle diverging section of 1, the most expressive increasein speed occured for the case 2, where the concentrator had an opening angleof 60° and L1/d = 1/2. For the cases from 5 to 8 we can notice that, independently of the concentrator geometry, the increase of length ratio in divergent section for 1.5 did not contributesthe increase in speed even mantaining the angle in 4°. Although, in the cases 7 and 8, the speeds decreased in most of the nozzle internal section.

In almost all cases from 9 to 16 where the divergent angle section was increased to 15°, the maximum speeds along the symmetry axis of the nozzle were higher to speed levels obtained in previous cases where the angle was of 4°. This behavior was more marked in cases where the divergent section ratio is 1.5, showing that for the angle of 15° in divergent section of the nozzle, the length increase favoured the increase in flow speed.

Cases from 17 to 24 include the nozzle where the divergency angle was of 45°. We could notice that in most of these cases the increase in speed was still more significant when compared to previous cases. The cases 17, 18 and 22 were the only that promoted lower increase in flow speed if compared to cases with divergent angle of 4°. Except for

the case 20, we could notice that keeping the divergence angle in 45°, a higher increase in speed occur inside the nozzle with a length ratio increase of the divergent section of 1 for 1.5.

Garcia et al. (2014),developed studies aiming optimize the power generation conditions. They used nozzle in marine turbines. These authors prioritized in their studies, evaluating the ducts design and geometries effect on increasing efficiency by using computational simulation These authors obtained speed ratio of 2.55 for a Flow-lens diffuser type geometry with input angle of 35°, input legth of 10mm, a duct with length of 850 mm (where the turbine would be placed) and a length divergent section of 4000 mm and angle of 5°. The authors did not report the used diameter, so it was difficult to compare the obtained results in this work.

3.2 Effect of the convergent section geometry

For evaluating the effect of the concentrator angle on the wind flow speed inside the nozzle, the speed ratios shown on the table 2 and the speed along the symmetry axis of the nozzle were compared, defining the other parameters.

From this analysis, we could notice that if keeping the length ratio of a convergent section L1/d = 1/2, a higher increase in flow speed occurs by using angle of 60º at the input when compared to an angle of 30º. However this behavior was virified only in cases that the divergent angle section were maintained in 4° and 15º. For the cases where the divergent angle section was of 45°, the behavior was reverse. So, the angle of 30° at the input contributed most for the increase in speed. In evaluating the effect of the concentrator angle for the cases where the length ratio was of 1/8, it was verified that the same behavior is not present and the concentrator angle separately, has a lower effect on the flow speed when compared to other factors for L1/d in the range of 1/8.

In evaluating the effect of length ratio L1/d on the speeds, we could notice that in most cases where the divergent section was built with angle of 4 and 15°, the use of ratio L 1/d = 1/2 was more likely for increasing the speed inside the nozzle. For the cases with divergence angle of 45º, this length also was reverse, so that the cases with L1/d = 1/8 promoted higher increase in speeds.

Anzai et al. (2004) confirmed the geometry effect of convergent section on the increase of power and verified a certain curvature at the inlet section and increasing the diameter of 680 mm to 880 mm the power coefficient increase of 0.42 to 0.57. At the test performed without the section, the measured coefficient was approximately 0.3%coefficient 0.3%. These authors reported that the outlet section diameter also has a significant effect on the power coefficient .i Shikha et al. (2005), also verified the effects on length and input angle at the divergent section on the wind increase in speed.length and section input angle effects on the increase in speed of the wind. For the setupused, the maximum increase in wind speed was achieved whenthe concentrator length, was 55 cm after testing the lengthsranging from 25cm to 80cm. Increasing concentrator lengthresulted in increasedfrictional losses, but the incidence angle, or inlet angle, decreased resultingin increased component of wind velocity parallel to the concentrator wall. It is therefore important to optimize concentrator length and inlet angle since they depend on each other while influencing concentrator frictional losses.

3.3 Selected models

Based on these analyses and presented results, we can say that the combination of factors L1/d = 1/8, α1 = 60°, L2/d = 1 and α2 = 45° promoted a higher increase in speed of the wind. This increase is much higher if observed from other geometries. The speed gradients gerated with the simulation of this case are shown at figure 4.
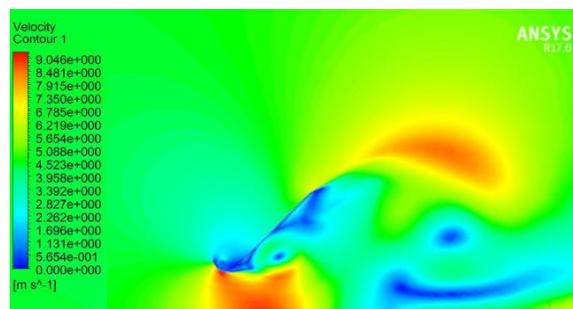

Figure 4 –Velocity contours of flow nozzle of case 20.

For these model, with the level of increase in speed obtained (1.98) it was possible obtain a theoretical value for the power ratio of 7.76. It indicates that despite the losses for system inefficiency the power generated by using the nozzle may be in a range of 7.76 times higher than a generated power in conventional system without a nozzle.

Although we use a simplification for obtaining the power ratio, other researchers such as Gilbert and Foreman (1983) by experimental procedure have found that the wind turbine extracted 4.25 times the extracted power by the same turbine without a diffusor.

Abe et al. (2005) andOhya et al. (2008) in numerical and experimental analysis observed that the power to the turbines by using the diffusor is aproximately 4 times higher for speed ratio of 1.5 and 1.6 approximately. The increase in power observed by using a turbine in nozzle is due to, beyond other possible factors, the increase in mass flow caused by reduction of pressure at the flow outlet. It is relevant to mention that many authors did not evaluated the combined effect of different geometries at the inlet section of the nozzle with the geometries of the diffusion section. It may explain partially a higher level of increase in speed and so the the power ratio observed in this study. Ohia et al.(2010), related that for a diffusor with inlet, it is possible obtain speeds ratio of aproximately 1.3 to 2 for L/d of the diffuser ranging from 1.5 to 4.5. However these authors did not analysed the settings effects at the inlet area. As discussed before, the geometry of the inlet section (convergent section) had a significant effect on the speed ratios.

Chaker et al., (2012) shown by computational simulation and by experiments that when the diffuser is analyzed with the turbine, a reduction in the speed ratio of about 20% is observed.

The application of this correction would cause a reduction in our speed ratio to be in reality approximately 1.58. To confirm these hypotheses, further simulation studies and experiments should be performed taking into account the effect of the rotor within the nozzle.

Other geometries as in cases 14, 16, 19, 21, 23 and 24 (shown in Figure 5) may be interesting for further studies. Variables such as the use of flanges and other accessories to increase speed or decrease drag on the structure should also be evaluated.
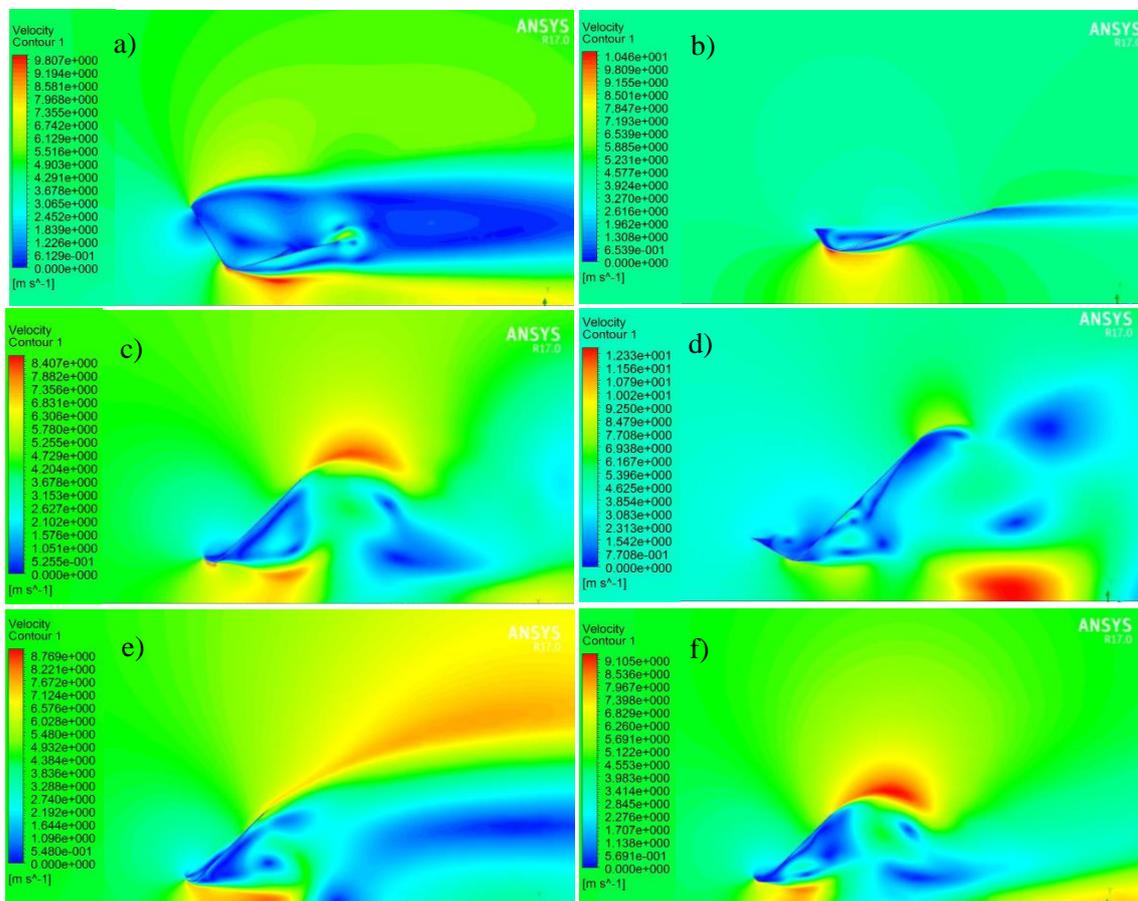

Figure 5 – Velocity contours for flow in nozzle: a) 14, b) 16, c) 19, d) 21, e) 23 e f) 24.

4. CONCLUSIONS

The present work has shown that one of the geometries among all the nozzles evaluated, promoted higher acceleration of the wind in the nozzle. This geometry has an angle of 60 ° and a length ratio of 1/8 at the nozzle inlet, and anangle of 45 ° and a length ratio of 1 at the outlet of the nozzle.

Under these conditions the velocity increase ratio was 1.98 in a large part of the internal area, a very interesting result for future discussions on wind generation for low speed regions.

To obtain the results of this work it was necessary to use a smoothing in the region of encounter between the convergent section and the divergent section (throat of the nozzle), which was standardized for all the geometries used.

The theoretical power ratio was 7.76, indicating that using the methodology described in this computational study, the theoretical power of a turbine enveloped by a nozzle with the configurations described above is 7.76 times higher when compared to the power of the same turbine without nozzle.

The numerical method using fluent was interesting for the study of wind flow in nozzles, even without including the turbine in the computational analysis, because it allows the understanding of the physical phenomena involved with the geometry change. The use of computational simulation allowed the reduction in the demand for financial and labor resources with experiments that will be performed only for optimized cases.

For the next studies, other geometries will be evaluated around the optimum case obtained in this work, aiming also the smoothing of the external surfaces for a minimization of drag forces and positive lift.

The effect of the wind speedstream on the efficiency of each proposed geometry still needs to be evaluated in order to establish for each one the limitations of application.

5. ACKNOWLEDGEMENTS

6. REFERENCES

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, Journal Wind Engineering and Industrial Aerodynamics., Vol. 93, p. 951-970.

Ahsan, M.D., Bishnoi., Ramniwas., Singh., Sanjeev K.R., 2015. CFD Analysis of Wind Concentrator. International Journal of Mechanical and Industrial Technology., Vol. 2, n.2, p: 51-78.

Anzai, A., Nemoto, Y., Ushiyama, I., 2004. Wind tunnel analysis of concentrators for augmented wind turbines., Wind Engineering., Vol.28, p. 605–13.

Gilbert, B.L. and Foreman, K.M., "Experiments with a diffuser augmented model wind turbine," Journal of Energy Resources Technology, Transactions of the ASME., Vol. 105, no. 1, p. 46–53.

García, E., Pizá, R., Benavides, X., Quiles, E., Correcher, A., Morant. F., 2014.Mechanical Augmentation Channel Design forTurbine Current Generators. Advances in Mechanical Engineering., Vol. 1, 12p.

Chaker, R., Kardous, M., Aloui, F., Nasrallah, S.B., 2012. "Relationship between Open Angle and Aerodynamic Performaces of a DAWT", The Fourth International Renewable Energy Congress, December 20-22, – Sousse, Tunisia.

Dick, E., 1986. "Power Limits for Wind Energy Concentrator Systems". Wind Energy., Vol. 10, n. 2, p. 98-115.

Foreman, K.M., Gilbert, B., Oman, R.A. 1978. Diffuser Augmentation of Wind Turbines. Solar Energy., Vol. 20, p. 305-311.

GRUBB, M.J. and MEYER, N., I., 1993. Análise sobre a implantação de aerogeradores. 18 Fev. 2015 <http://www.sbpcnet.org.br/livro/58ra/SENIOR/RESUMOS/resumo_3032.html>.

Jafari, S., and Kosasih, B., 2014. "Flow analysis of shrouded small wind turbine with a simple frustum diffuser with computational fluid dynamics simulations", Journal of Wind Engineering and Industrial Aerodynamics., Vol. 125,p. 102-110.

Kosasih, P.B., Bryce1, N.,Tondelli, A., Beazley,A. 2012.Experimental Study of the Performance of Bare and Nozzle – Diffuser ShroudedMicro Wind Turbine under axial and non-axial Inflow Condition. 18th Australasian Fluid Mechanics Conference, December 3-7, - Launceston, Australia.

Mansour, K., and Meskinkhoda, P., 2014. "Computational analysis of flow fields around flanged diffusers", Journal of Wind Engineering and Industrial Aerodynamics., Vol. 124, p. 109-120.

Ohya, Y., and Karasudani, T., 2010. "A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology", Energies., Vol. 3, p. 634–649.

Ohya, Y., Karasudani, T., SakuraI, A., Abe, K.., Inoue, M., 2008. "Development of a shrouded wind turbine with a flanged diffuser", Journal of Wind Engineering and Industrial Aerodynamic, Vol. 96, p.524–539.

Shikha, S., Bhatti, TS., Kothari, D.P., 2005. "Air concentrating nozzles: a promising option for wind turbines". International Journal of Energy Technology and Policy., Vol. 3, p. 394–412.

Takahashi, S., Hata, Y., Ohya, Y., Karasudani, T., Uchida, T., 2012. "Behavior of the blade tip vortices of a wind turbine equipped with a brimmed-diffuser shroud", Energies., Vol. 5, n. 12, p. 5229-5242.

Wang, F., Bai, L., Fletcher, J., Whiteford, J. Cullen, D., 2008. "The methodology for aerodynamic study on a small domestic wind turbine with scoop", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96, n.1, p.1-24.

7. RESPONSIBILITY NOTICE

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