ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL STUDY OF FLOW AROUND DIFFUSERS WITH DIFFERENT GEOMETRY USING CFD APPLIED TO HYDROKINETICS TURBINES DESIGN

Léo Daiki Shinomiya
Faculdade de Engenharia Mecânica, Instituto de Tecnologia, Universidade Federal do Pará, Rua Augusto Corrêa, 01, CEP.: 66075-110, Guamá, Belém, Pará. , Fone: (91) 3201-7962
leo.shinomiya@itec.ufpa.br

Déborah Aline Tavares Dias do Rio Vaz
Programa de Pós-GraduaçãoemEngenharia de RecursosNaturais da Amazônia, Universidade Federal do Pará, Rua Augusto Corrêa, 01, CEP.: 66075-110, Guamá, Belém, Pará, Fone: (91) 3201-7962
deborahvaz@ufpa.br

Amanda Maria Bizzinotto Ferreira
Departamento de Enegenharia Mecânica, Instituto de Ciências Tecnológicas e Exatas, Universidade Federal do Triângulo Mineiro, Avenida Doutor Randolfo Borges Júnior, 1250, CEP 38064-200, Uberaba, Minas Gerais, Fone: (34)3318-5600
amandabizinotto@gmail.com

Taygoara Felamingo de Oliveira
Departamento de Engenharia Mecânica, Faculdade de Tecnologia, Universidade de Brasília, Campus Darcy Ribeiro, Asa Norte, CEP: 70910-900 Brasília, Distrito Federal, Fone: (61)3107-5503
taygoara@unb.br

José Gustavo Coelho
Departamento de Enegenharia Mecânica, Instituto de Ciências Tecnológicas e Exatas, Universidade Federal do Triângulo Mineiro, Avenida Doutor Randolfo Borges Júnior, 1250, CEP 38064-200, Uberaba, Minas Gerais, Fone: (34)3318-5600
josegustavo@icte.uftm.edu.br

André Luiz Amarante Mesquita
Faculdade de Engenharia Mecânica, Instituto de Tecnologia, Universidade Federal do Pará, Rua Augusto Corrêa, 01, CEP.: 66075-110, Guamá, Belém, Pará. , Fone: (91) 3201-7962
andream@ufpa.br

Jerson Rogério Pinheiro Vaz
Faculdade de Engenharia Mecânica, Instituto de Tecnologia, Universidade Federal do Pará, Rua Augusto Corrêa, 01, CEP.: 66075-110, Guamá, Belém, Pará. , Fone: (91) 3201-7962
jerson@ufpa.br

Abstract. *The use of diffuser on the hydrokinetic turbines improves its efficiency, exceeding the Betz limit (59.26%). The diffusers are technologies which has the function of causing an effect of increase in flow velocity that arrives on the turbine blades. This effect is caused by acceleration of the fluid particles due to the pressure drop downstream of the diffuser. In this paper, one describes a numerical study of the flow around three different geometries diffusers applied to the horizontal axis hydrokinetic turbine design. We evaluate the behavior of the diffuser velocity speed-up ratio in order of ranking the efficiencies of three different geometries. The numerical study is performed using computational fluid dynamic (CFD). The numerical model is validated using experimental data available in the literature. The results are applied to the case of a hydrokinetic turbine and the horizontal axis shows were satisfactory.*

Keywords: *Diffusers, Hydrokinetics turbines, CFD.*

1. INTRODUCTION

The study of the flow around the diffuser has great importance, since, when applied to wind and hydrokinetic turbines, the power coefficient can exceed the Betz limit increasing the power generated (Abe et al, 2004, Hansen, 2008, Ohya and Karasudani, 2010). The diffuser is an innovation that has piqued the interest of many researchers for possible use in generating energy more efficiently, due to the considerable increase in power extracted from the kinetic energy due to the motion of the fluid. The use of diffusers on the hydrokinetic turbines is a more efficient way of extracting energy, since even a small increase in mass flow within the diffuser leads to a proportional increase in energy production (Rio Vaz et al. 2011).

ISSN 2176-5480

Oman et al. (1975), Foreman and Gilbert (1979) conducted an experimental study of wind turbines with diffusers which increase the speed ratio between the velocity at the rotor plane and the undisturbed flow velocity, it may be two or more times greater compared with wind turbine without diffusers. This fact leads to a considerable increase in turbine power coefficient, exceeding the Betz limit (1926), which is 59.26%. The Betz limit may be exceeded when the turbine is positioned in the diffuser, since the flow within the diffuser provides an increase in mass flow through the plane of the rotor due to the suction pressure caused by the diffuser (Rodrigues, 2007, Hansen, Sorensen, and Flay, 2000). Figure 1 illustrates the flow through a hydrokinetic turbine diffuser.

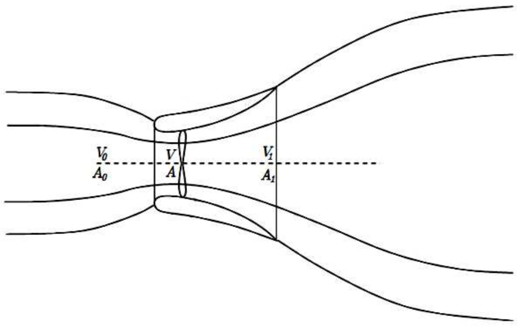


Figure 1. Simplified illustration of the velocities in the plane of the rotor and on the wake.

where V_0 is the free flow velocity, V is the axial velocity on the rotor plane and V_1 is the axial velocity on the diffuser outlet. Thus, in the present work, carried out a study using computational fluid dynamic (CFD) to assess the effect of the velocity speed-up ratio inside the diffuser on the efficiency of wind and hydrokinetic turbines. Simulations have the intention to develop a study in order to identify a geometry diffuser that has good hydrodynamic efficiency.

2. NUMERICAL AND COMPUTATIONAL CONDITIONS

The numerical method used corresponds to the finite volume method, using the software ANSYS - FLUENT. The present flow field is generally expressed by the continuity and the incompressible Reynolds-averaged Navier-Stokes equations (Abe and Ohaya, 2004). These equations can be written for the Newtonian fluid flow, such as:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$U_i \frac{\partial U_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left\{ v\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \overline{u_i u_j} \right\} + F_i \tag{2}$$

where $\overline{u_i u_j}$ denotes Reynolds-averaged value. In the equation (2), ρ, P, U_i, u_i and v, represents respectively, the density, the static pressure, the mean velocity, turbulent velocity and kinematic viscosity; F_i is the body-force term imposed for the representation of a load (Abe and Ohaya, 2004). The F_i in present study is zero because the analysis does not consider the rotor loads.

Computational conditions consist in an uniform structured mesh around each diffuser as shown in figure (2), this model has been implemented in each mesh geometry diffuser. In this work, the flow was considered to be a constant flow axisymmetric. The input velocity used in the simulation was 2.5 m/s. In this case, the fluid was considered as the water, temperature to 25 ^{o}C. The Reynolds number was calculated according to equation (6).

$$R_e = \frac{V_0 D}{v} \tag{3}$$

ISSN 2176-5480

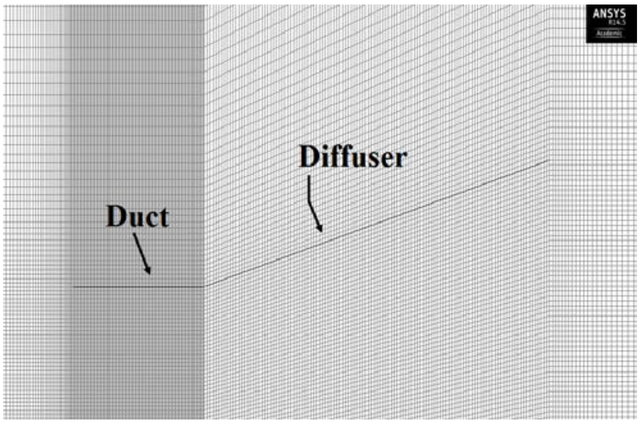


Figure 2. Mesh considered in the simulation.

2.1 Diffusers Geometry

To validate the model was used the geometry proposed by Abe and Ohya et al. (2004) (see Figure 3). Figures 4, 5, 6 and 7 show the geometries used in this study.

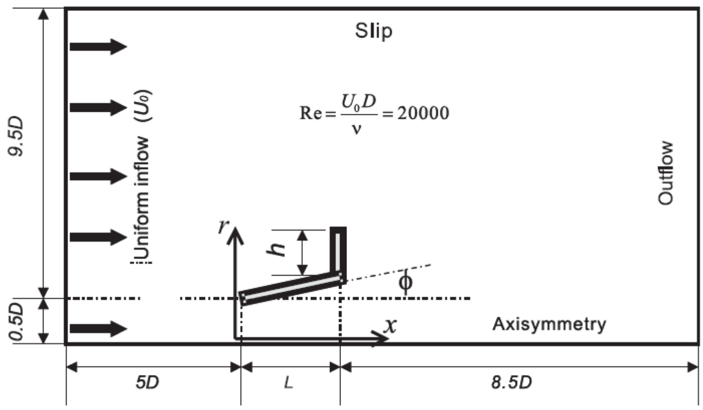


Figure 3. Geometry and computational conditions (Abe e Ohya, 2004) for validation.

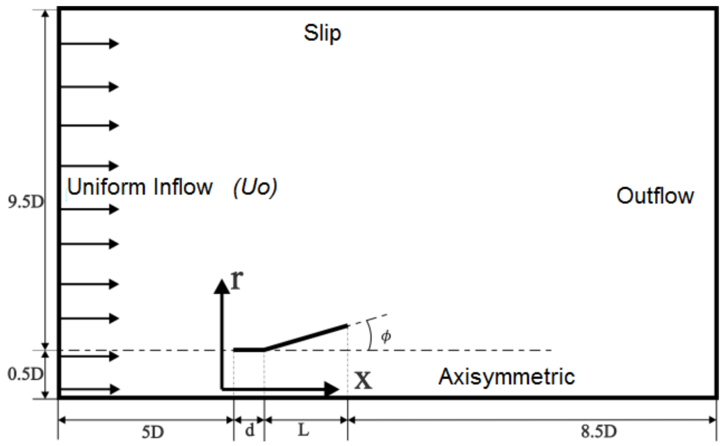


Figure 4. Geometry and computational conditions - Geometry 1.

ISSN 2176-5480

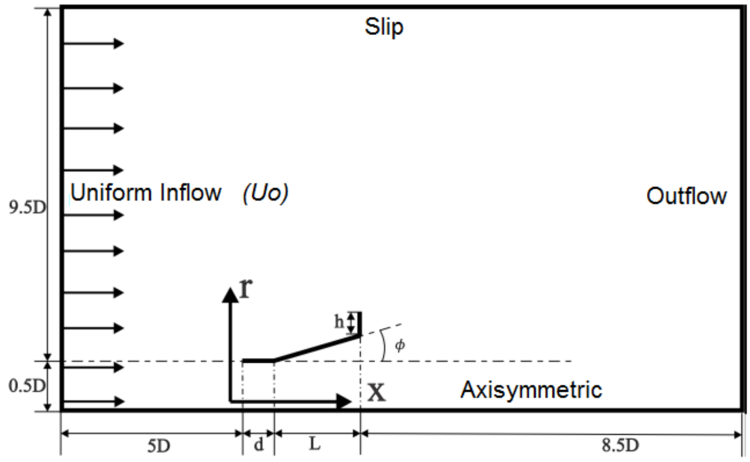


Figure 5. Geometry and computational conditions - Geometry 2.

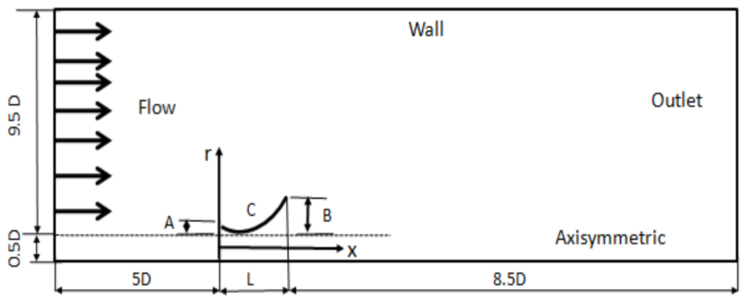


Figure 6. Geometry and computational conditions (Ohya and Karasudani, 2010) - Geometry 3.

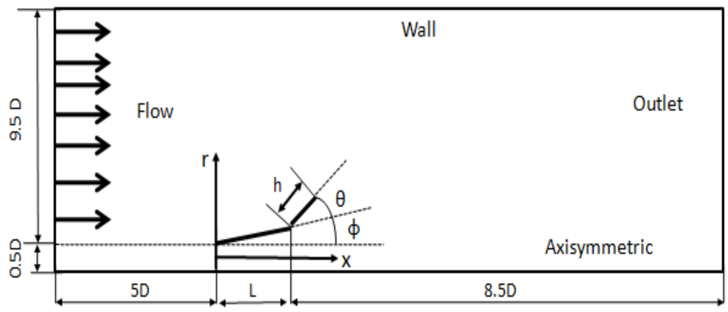


Figure 7. Geometry and computational conditions, variant of the Abe e Ohya – Geometry 4.

where L, D, ϕ, H and d denote the length of the diffuser, diameter at the diffuser inlet, the opening angle ($\varphi = 15$ °) of the diffuser flange height and length of the duct. x and r are the coordinates of the model.

3 RESULTS AND DISCUSSION

In this study the CFD computation is carried out using the ANSYS software with the *SST* (Shear Stress Transport) turbulence model. In order to validate the CFD simulation, the experimental data available in the literature was used Abe and Ohya (2004). Figure 8 presents the results obtained. The good agreement between numerical and experimental results permits to validate the CFD model.

ISSN 2176-5480

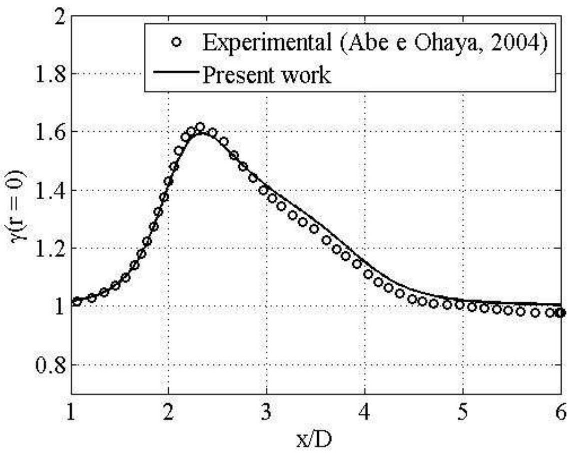


Figure 8. Validation of the simulation.

Figure 9 show that the presence of the diffuser influences the flow, causing an increase in the velocity inside the diffuser, this effect occurs at all geometries investigated in this work. This increase occurs due to the shape of the diffuser geometry. The flange increases the effect of movement behind the diffuser causing a drop in static pressure and consequently increasing the speed within the diffuser.

The geometry 4 presents the highest speed-up ratio. However this geometry present an aspect ratio *L/D* = 1.5. This fact is not suitable for large diameter rotors. For example, for a rotor diameter of 5 m would result in a length of the diffuser approximately 7.5 m, considering that the rotor diameter is approximately equal to the diameter of the diffuser inlet. This would lead to a system with high costs in construction.

Therefore, the geometry 2 is the most appropriate for use in hydrokinetic turbines, since it possesses an aspect ratio less (*L/D* = 0.35). With this geometry the length of the diffuser with the same rotor would be 1.75 m, making the system more inexpensive. The implementation of the duct, in the geometry 1, is interesting, because the flow velocity is better distributed in this region, making it easier to position the rotor, and can have lower drag on the diffuser. It is noteworthy that, in this work the goal is to present a simplified numerical study on speedup internally in different geometries diffusers in order to evaluate geometries with good performance and application possibilities in the design of wind and hydrokinetic turbines, is not to describe in detail the effect of drag on the geometries, nor the effects caused by different turbulence models in the literature. Figure 10 shows the velocity field for the simulated geometries.

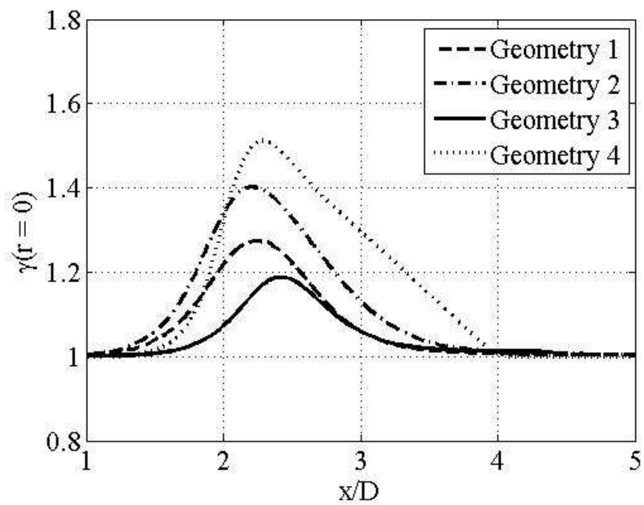
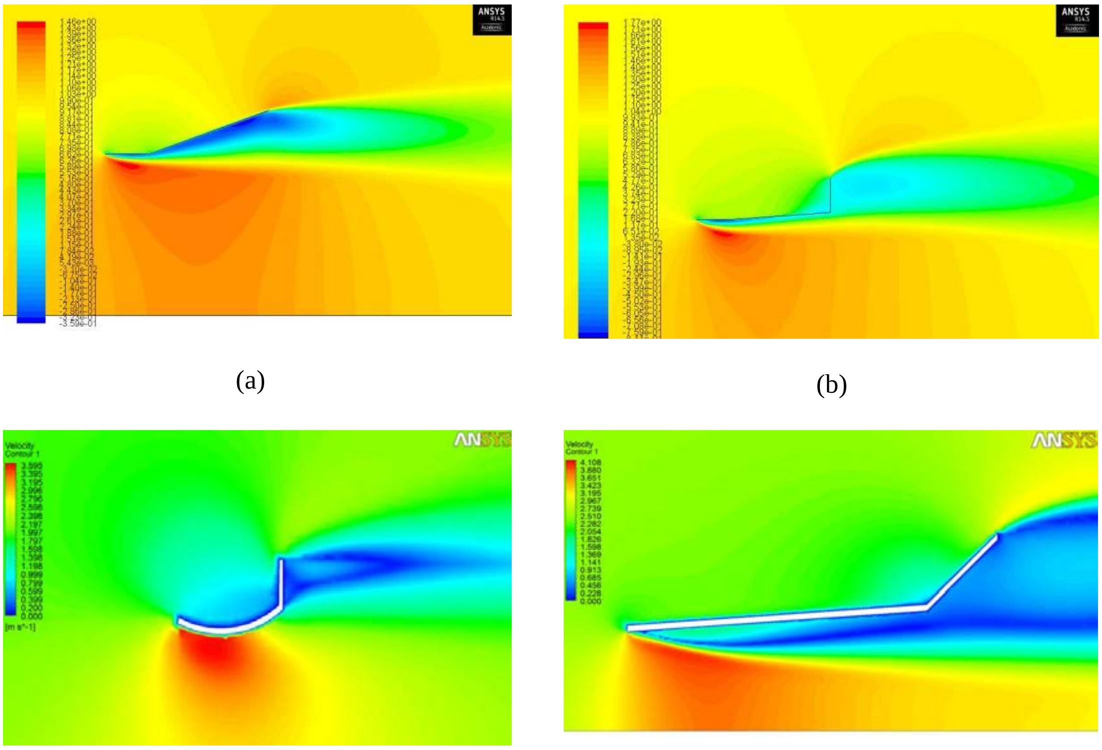


Figure 9. Velocity Speed-up ratio on the axis of symmetry for the simulated geometries.



(a) (b) (c) (d)

Figure 10. (a) Velocity field for the geometry 1. (b) Velocity field for the geometry 2. (c) Velocity field for the geometry 3. (d) Velocity field for the geometry 4.

Figure 11 shows the velocity speed-up ratio behavior as a function of radial position within the diffuser. This result shows that the incident flow velocity on the turbine varies with the radial position and must be taken into account in the design of an efficient of the horizontal axis turbine. It is noted that the geometry 2 presents the greatest velocity magnitude. This aspect results in a higher efficiency of the turbine.

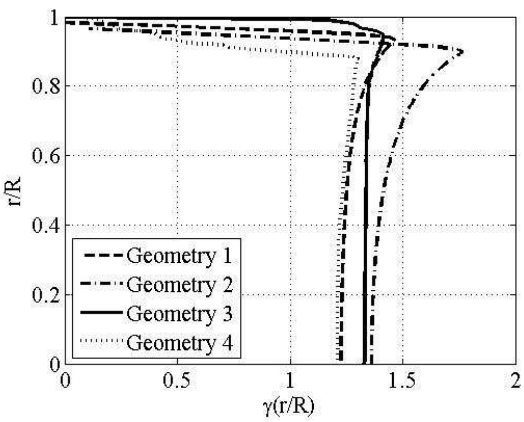


Figure 11. Velocity Speed-up ratio as a function of radial position for all geometries.

The effect of the diffuser on the flow velocity (shown in figure 9) results in the velocity speed-up ratios of speeds shown in Table 1. Classically, the curve of the free speed decreases to 2/3 to pass through an ideal turbine (Glauert, 1935). The values in Table 1 show that the speed in the rotor plane with diffuser can increase the speed to 1.5 times than a turbine without diffuser. This fact shows that the suction region at the diffuser outlet induced increases the mass flow in the plane of the rotor, resulting in an extrapolation of the Betz limit (as described in Rio Vaz *et al*. 2011)

ISSN 2176-5480

Table 1. Maximum velocity speed-up ratio for each diffuser geometry.

Diffusers geometries	γ_{max}
Geometry 1	1.27
Geometry 2	1.40
Geometry 3	1.20
Geometry 4	1.50

The use of the flange provides a notable increase in the power coefficient of a turbine positioned internally of the diffuser. Fig (12) shows the results for a ideal turbine. For the theoretical efficiency was used the model described by Rio Vaz et al. (2011), where the power coefficient is defined by:

$$Cp = \gamma_{\max} C_T \left(\frac{1+\sqrt{1-C_T}}{2} \right) \tag{7}$$

where γ is the maximum velocity speed-up ratio, and C_T is the thrust coefficient.

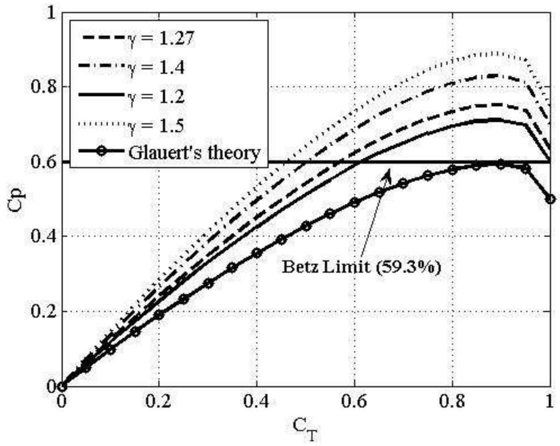


Figure 12. Theoretical power coefficient under diffuser effect.

It is observed that the result for the geometry diffuser 2 has a better efficiency, i.e, it has a considerable increase in the theoretical power extracted of the kinetic energy by the fluid motion, as shown in Fig. (12).

4 CONCLUSIONS

The numerical simulations confirm that the diffuser can increase the efficiency of a horizontal axis turbine. The results presented in figure (12) shows that the diffusers improve efficiency compared to conventional turbines, without the use of diffusers and confirmed in the simulations in this work and experimental studies made by Abe and Ohya (2004), however, still there is a great need to improve the knowledge about the geometry to be used to produce more energy in wind and hydrokinetic turbines. In this work, the diffuser geometry 2 proves to be the shape that can generate power more efficiently compared to other diffusers geometries studied, these results are preliminary, however, using a theoretical comparison shows an increase in the efficiency of wind turbine and hydrokinetic with diffuser.

5 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 of Wind Engineering Industrial Aerodynamics, v. 93, p. 951-970.

Abe, K., Ohya, Y, 2004 "An investigation of flow fields around flanged diffusers using CFD", Journal of Wind Engineering Industrial Aerodynamics, v. 92, p. 315-330.

Bardina, J.E., Huang, P.G., Coakley, T.J. 1997, "Turbulence Modeling Validation, Testing, and Development", NASA Technical Memorandum 110446.

Betz, A., 1926 ,"Wind Energie und ihre Ausnutzung durch Windmuehlen".

ISSN 2176-5480

Brasil-Junior, A. C. P., Salomon, L. R. B., Els, R. V., Ferreira, W. O., 2006."A New Conception of Hydrokinetic Turbine of Isolated Communities in Amazon", IV Congresso Nacional de Engenharia Mecânica, Recife, Pernambuco, Brasil.

Foreman, K. M. and Gilbert, B.L., 1979, "Technical Development of the Diffuser Augmented Wind Turbine (DAWT)" Concept, Wind Energy Innovative Systems Conf. Proc., Colorado Spring, Colorado, USA, pp 121-134,

Freire, A. P. S., Menut, P. P. M. e Su, J., 2002, "Turbulência", Coleção Cadernos de Turbulência, *ABCM - Associação Brasileira de Ciência Mecânicas*, Vol. 1, Rio de Janeiro, Brasil, pp. 191-224.

Glauert, H., 1935, *AirPlane Propellers, in Aerodynamic Theory.* Ed. W. F. Durand, Spring Verlag.

Hansen, M. O. L., Sorensen, N. N. and Flay, R. G. J., 2000, "Effect of placing a diffuser around a wind turbine", Wind Energy, Vol 3.

Hansen, M., 2008, *Aerodynamics of Wind Turbines*, 2nd Edition, Earthscan.

Harlow, H. & Nakayama, P.I., 1968, "Transport of turbulent energy decay rate", Rep. no LA- 3854 Los Alamos, cit. Jones & Launder (1972).

Jones, W. P., and Launder, B. E. 1972, "The Prediction of Laminarization with a Two-Equation Model of Turbulence", *International Journal of Heat and Mass Transfer*, Vol. 15.

Launder, B. E., and Sharma, B. I. 1974, "Application of the Energy Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc", *Letters in Heat and Mass Transfer*, Vol. 1, no. 2.

Launder, B. E., Spalding, D. B.1974, "The numerical computation of turbulent flows". *Computer Methods in Applied Mechanics and Engineering*, 3, n. 2.

Ohya, Y. and Karasudani, T., 2010, "A shrouded wind turbine generating high output power with wind-lens technology", *Energies*, v. 3, pp. 634-649.

Oman, R. A., Foreman, K. M. and Gilbert, B.L., 1975, "A Progress Report on the Diffuser Augmented Wind Turbine", Proc., 3rd Biennal Conference and Workshop on Wind Energy Conversion Systems, Washington, D. C., USA.

Rio Vaz, D. A. T. D., 2011, "Projeto de Rotores Hidrocinéticos de Eixo Horizontal Adaptados às Condições dos Rios Amazônicos", *M. Sc. thesis*, Universidade Federal do Pará, Brasil, pp. 63-73.

Rodrigues, A. P. de S. P., 2007, "Parametrização e Simulação Numérica da Turbina Hidrocinética – Otimização Via Algoritmos Genéticos", Dissertação de Mestrado em Ciências Mecânicas, Universidade de Brasília, Brasília.

Fraenkel, P.L., 2007. Marine current turbines: pioneering the development of marine kinetic energy converters. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 221(2), 159-169.

Batten, W., Bajaj A., Molland A and Chaplin J., 2008. "The prediction of the hydrodynamic performance of marine current turbines". *Renewable Energy*, 33(5), 1085-1096.

Abe and Ohya, 2004. A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology. *Journal of Wind Engineering and Industrial Aerodynamics*. 92 (2004) 315–330

Yuji Ohya and Takashi Karasudani, 2010. "A Shrouded Wind Turbine Generating High Output Power with Wind-lens Technology", *Energies*, 634–649

6 RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.