COB-2019-0802
A NEW DIFFUSER TECHNOLOGY APPLIED TO HYDROKINETIC TURBINES

Marilza S. Viana
Pedro O. C. da Silva
Pedro E. S. Barbosa
Luiz F. V. Ferry
Jerson R. P. Vaz
Federal University of Para -Av. Augusto Correa, nº 1 -Belém, PA, 66075-900, Brazil
marilza182_@hotmail.com, pedro.0106@gmail.com, ped.barbosa97@gmail.com, luiz.ferry@live.com, jerson@ufpa.br

Abstract. *Diffuser technology has been strongly studied worldwide, mainly because such a technology is able to improve hydrokinetic turbine performance. Research on diffuser geometry has been made in order to assess the impact of efficiency and thrust of diffuser on turbines. These parameters are important because they are able to change turbine power coefficient, strongly contributing to a better kinetic energy extraction from the flow. Hence, this work concentrates on a novel diffuser geometry, which is able to reach an efficiency of 98 %. The new geometry has a double flange, whose objective is to increase the suction region at the diffuser downstream. The flow around the new geometry is simulated through computational fluid dynamics, using $\kappa - \omega$ shear-stress transport turbulence model. An extended Blade Element Momentum approach is used to evaluate turbine performance, in which the effect of efficiency and thrust of the diffuser is employed. The double-diffuser shows relevant changing on the flow velocity and pressure fields, enhancing the turbine performance.*

Keywords: *Diffuser technology, Hydrokinetic turbine, Renewable energy*

1. INTRODUCTION

Diffuser-augmented hydrokinetic turbines have been used as alternative technologies for clean and low-cost energy generation. These turbines are capable of utilizing relative small river streams without the need of building dam. In the last decades, several work have been made on new diffuser geometries in the literature. However, only a few has been developed on the study of effects of diffuser efficiency and thrust.

Initially, several researchers have studied the effects of diffusers for wind energy generation. Among them, Abe & Ohya (2004) have investigated the characteristics of the flow field around a flanged diffuser, highlighting the importance of the load coefficient in the performance of the diffuser, as well as of the opening angle. Barbosa et al (2015) developed a mathematical formulation to describe the velocity profile inside diffusers that are applied to wind turbines.

Now turning the attention to the effects of diffuser applied to hydrokinetic turbines, Riglin et al (2014) experimented with conical non-flanged diffusers in water, observing a velocity ratio of 1.45. Silva et al (2017) studied two configurations, the flanged conical diffuser and flanged lens diffuser, using a novel approach that takes the diffuser efficiency into account in the blade element momentum theory. It was observed that for the flanged conical diffuser, the turbine experienced a 53 % increase in the power output. Gaden & Bibeu (2010) investigated the effects of the diffuser in hydro turbines, showing that this configuration produces 3.1 times more power than the conventional turbine without a diffuser. This paper focused on studying the benefits of a new diffuser design, that has two flanges instead of the regular one-flanged diffuser. A CFD simulation was carried out to compare the velocity ratio and the efficiency of both designs

2. NUMERICAL METHOD

2.1 Double-Diffuser Technology

The double-diffuser proposed in this work is based on the flanged diffuser developed by Abe and Ohya (2004), which performed a numerical investigation applied to small wind turbines under 1.5 kW. Their work showed that the performance of a flanged diffuser depends on two important subjects: the loading coefficient, and the opening angle as it affects the boundary layer separation at the inner diffuser wall. The results of the numerical investigations were validated with corresponding experimental data from Ohya et. al. (2008). The flanged diffuser comprises typically two parts: the conical one, and a flange at the diffuser outlet, as illustrated in Fig. 1. At small opening angle, the conical part is able to

maintain the maximum diffuser velocity ratio, γ, closest to the inlet, benefiting the turbine positioning, since rotor far from the inlet can expose the turbine to a region where the boundary layer is detached. Experimental investigations on the effect of opening angle of three different conical diffusers (5°, 22.5°, and 40°) are made by Barbosa et al (2015). In their work, increasing the opening angle, γ tends to move from the inlet to the diffuser outlet, increasing also the flow separation on the internal diffuser wall. According to them, the best diffuser geometry is that of 5°. Classically, conical diffusers need to be designed at low opening angles, where its efficiency is higher.
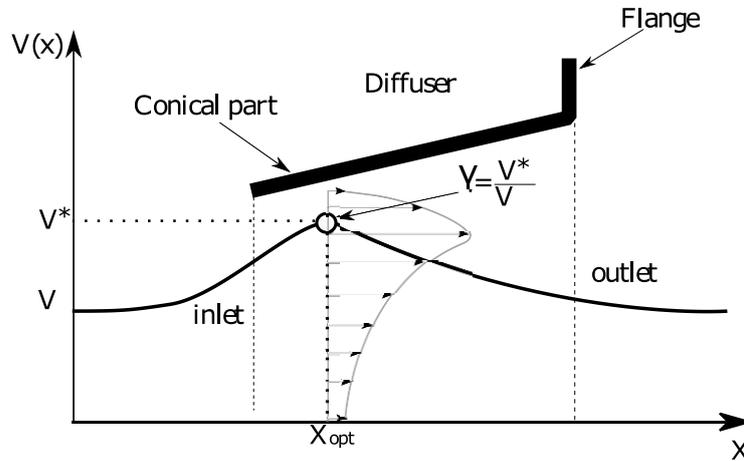

Figure 1. Simplified illustration of the flow velocity profile in a diffuser technology.

Another important part of the diffuser is the flange, which induces vortex formation behind it, producing pressure drop, accelerating the flow velocity throughout diffuser. The flange can increase γ about 4.7 % when compared to a geometry purely conic (Barbosa et al., 2015). In the present work, the double-diffuser is bio-inspired on the suction of fish gill slits. The water enters the fish's mouth due to its swimming motion, and it travels to the gills as the fish keeps going forward. For example, this breathing mechanism occurs in white and whale sharks. In a hydrokinetic turbine, it is vital to maintain a steady inflow of water, and this idea is the main motivation to conceive the double-diffuser proposed here. The new design has an extra diffuser layer with a special geometry, yielding a double-diffuser, as shown in Fig. 2a. The gap between flanges works like gill slit (Fig. 2b), allowing a smoother pathway for the water, increasing the vortex shedding at the diffuser outlet. In other words, the presence of a second flange, increases the formation of vortex in the wake region, causing pressure drop. This pressure drop is greater than what is observed in common diffuser designs. Therefore, it generates an increase in water flow through the diffuser inlet, raising up the velocity ratio, and diffuser efficiency. The diffuser efficiency, η_d, is defined as

$$\eta_d = \frac{p_2 - p_1}{\frac{\rho(V_1^2 - V_2^2)}{2}}$$

(1)

where p_1 and p_2 are static pressures at the inlet and at the diffuser outlet, respectively, while V_1 and V_2 are the corresponding velocities, and ρ is the flow density.
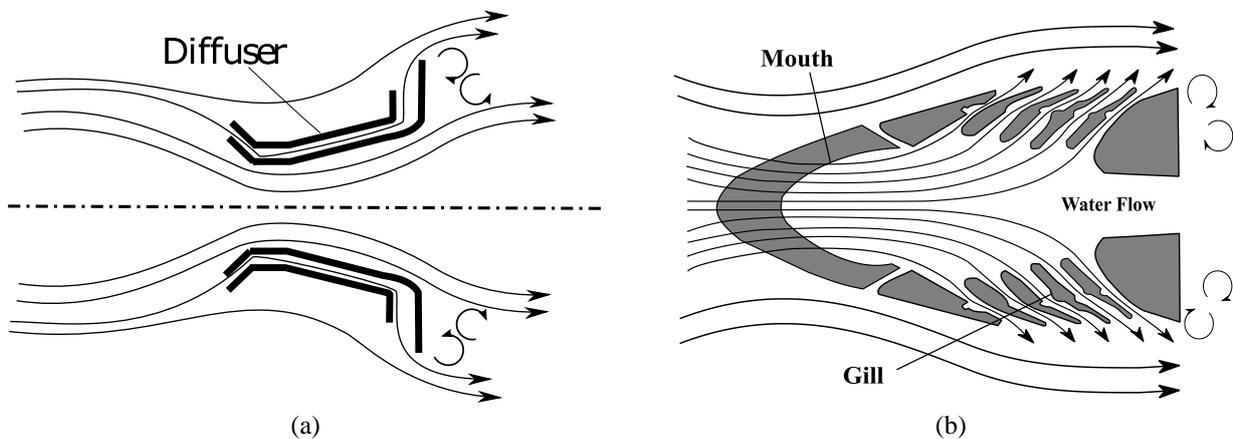

(a)   (b)

Figure 2. Illustration of water flow through (a) double-flanged diffuser, and (b) sharks gill slits.

2.2 Numerical Modeling

2.2.1 Geometrical Characteristics

The simulations are carried out with the purpose of evaluating the performance of the double-diffuser proposed here, as described in the last subsection. Figure 3a shows dimensions and shape of the flanged diffuser described by Abe & Ohya (2004). In this case, the dimensions are: L = 1.5D, h = 0.5D and φ = 4°. The double-diffuser design is obtained by offsetting the flanged diffuser by a distance d. The inlet is modified by including a horizontal and nozzle part. Then, a circular sector with radius R is attached at the end of the conical part. At the very edge of the circular sector, a flange with double the height as the original one is placed. The dimensions are as follows: L = 1.5D, a = 0.125D, b = 0.25D, h = 0.125D, H = 0.25D, R = 0.75D, d = 0.05D, G = 0.25D, φ = 4° and β = 15° It is importance to clarify that D in these specifications refers to the diameter of the outer diffuser. Figure 3b shows the new design.
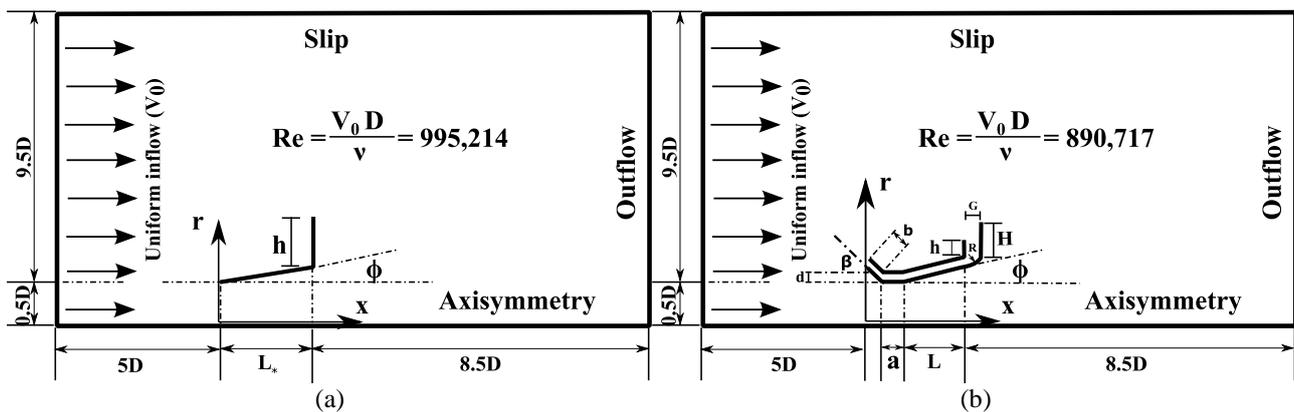

Figure 3. (a) Flanged diffuser. (b) Double-diffuser.

2.2.2 Computational Fluid Dynamics

It is well-known that turbulence phenomenon is characterized by chaotic motion of a fluid, where the molecules move along complex irregular paths. Due to a variety of length scales involved and time dependency, turbulence is one of the great unsolved scientific problems of the past century. The Navier-Stokes equations are a set of equations that mathematically describe the motion of viscous fluids, but when turbulence is involved, these equations become very difficult to solve numerically, demanding infeasible computational time and, therefore, rendering the study impossible for practical engineering applications (Wilcox, 1994). That is where the Reynolds-averaged Navier-Stokes equations come into play. They are time averaged equations that, along with turbulence models, make it possible to model turbulence in computational fluid dynamics applications.

In the present study, the chosen turbulence model is the Transition SST model. This model couples the SST k-ω transport equations with two other equations, one for intermittency and one for the transition onset criteria, using momentum-thickness Reynolds number. The computational domain is based on validations carried out by Abe & Ohya (2004). The dimensions of the domain are shown in Fig. 3, with the diffuser placed at 5D from the inlet surface. The Ansys FLUENT CFD package is used to simulate the flow. Since it is a 2D simulation, the domain is considered axisymmetric in a steady-state flow. The inlet boundary conditions are presented in Tab. 1. Zero gauge pressure is considered in the outflow boundary condition. The velocity is calculated in order to assure the requirement of Reynolds = 20000 made by Abe & Ohya (2004).

Table 1. Inlet Conditions

Condition	Value
Velocity [m/s]	1.46
Intermittency	1
Turbulent Intensity [%]	3
Turbulent Viscosity Ratio	10

To validate the present modelling, the CFD results obtained by Abe & Ohya (2004) are employed. For this, the level of mesh refinement has to be appropriate to avoid incoherent solutions. Therefore, the idea is to generate a mesh that is fine enough so that any further refinement does not change the final result. Another important factor is that such a refinement has to be done locally, in regions where high property gradients are observed. In the domain, the refinement is mainly focused at boundary layers, due to the fact that the turbulence model requires y^+ less than 1. To achieve this, first nodes closer to the diffuser wall are placed at $\Delta y = 10^{-6}$ m. Regarding the geometrical specifications, it is an unstructured mesh composed mainly of triangular elements, with quadrilaterals near the diffuser wall. Figure 4a illustrates the mesh around the double-diffuser, while Fig. 4b shows the details of the mesh near the surface.
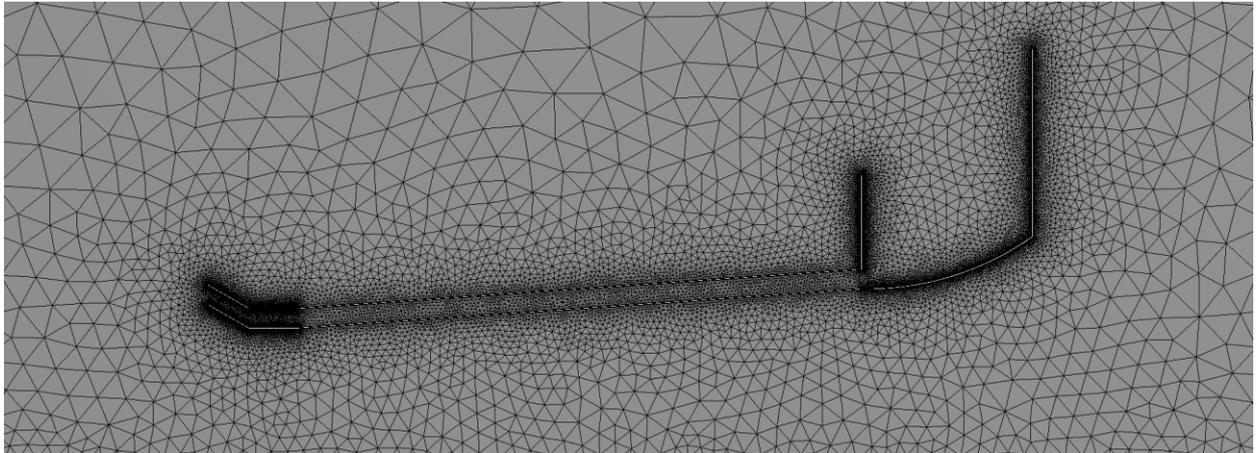
(a)
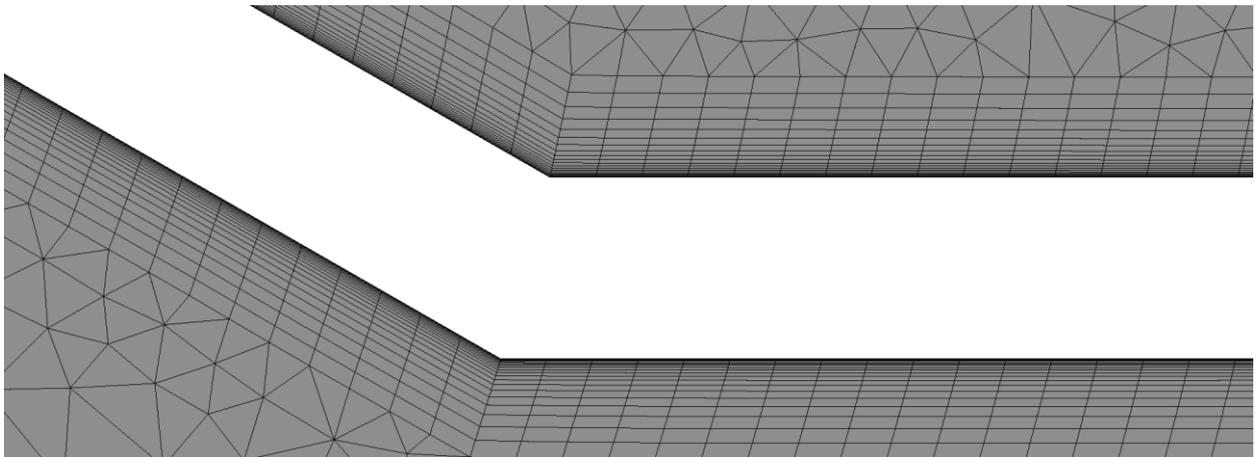
(b)

Figure 4. (a) Mesh around the diffuser. (b) Details of the mesh near the surface.

2.2.3 Validation

The on-axis stream wise velocity CFD results of Abe & Ohya (2004) for diffusers with a ratio h/D of 0.5 are reproduced here. Figure 5 shows the simulated results compared with that obtained. It can be seen that there is a reasonable trend in velocity variation, with the flow characteristics inside the diffuser also being well captured. To perform a grid dependency study, three meshes with different level of refinement are tested. Meshes B and C are capable of capturing the trend of the velocity profile, in which the results did not present any further differences. Therefore, the chosen mesh is mesh B, due to minimal computational cost. Table 2 shows additional information about the three meshes.
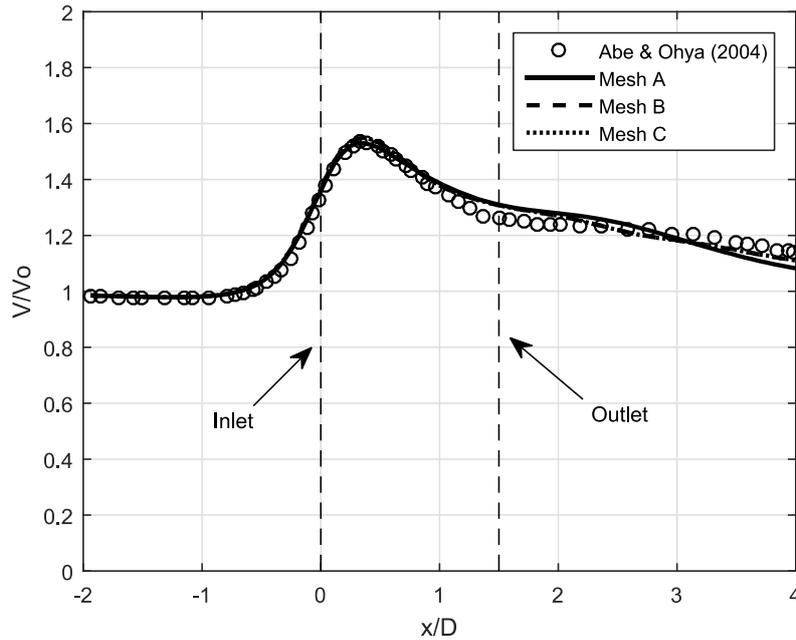
Figure 5. Results obtained with the domain

Table 2. Mesh Information

	Mesh A	Mesh B	Mesh C
Total Nodes	14315	18474	72048
y_{max}^+	0.5095311	0.5315897	0.5240282
y_{avg}^+	0.04723233	0.04701996	0.04657459
Element Size [m]	0.1	0.03	0.01
Avg Skewness	0.11043	0.096397	0.059121
Avg Orthogonal Quality	0.93707	0.94437	0.96447

3. RESULTS AND DISCUSSION

3.1 Diffuser Performance Analysis

For the performance analysis, several R/D ratios are simulated to assess the effect of the circular section radius. The results are shown in Fig. 6 along with the result obtained by Abe & Ohya (2004). It can be seen that the ratio R/D has significant impact on the performance of the diffuser. By increasing the ratio R/D, both the length of the diffuser and area ratio increase, and the graphs indicate that there is a direct correlation between the changes in these geometrical features and the velocity ratio generated by the diffuser. The pressure profiles show that the higher the area ratio, the lower the exit pressure. This is supported by Jafari & Kosasih (2014) and Hansen et al. (2000). The velocity ratio for both the R/D = 1 and R/D = 0.75 configurations are very close, which leads to the ratio R/D = 0.75 being chosen for further analysis due to the fact that it makes for a more compact diffuser. The maximum velocity ratio that can be generated is 1.9, against 1.51 generated by the single-flanged design studied by Abe & Ohya (2004). Fig. 7 shows that the pressure drop has a significant increase as well. To understand what caused this behavior, we will analyse the pressure contours, as well as the streamline plots.
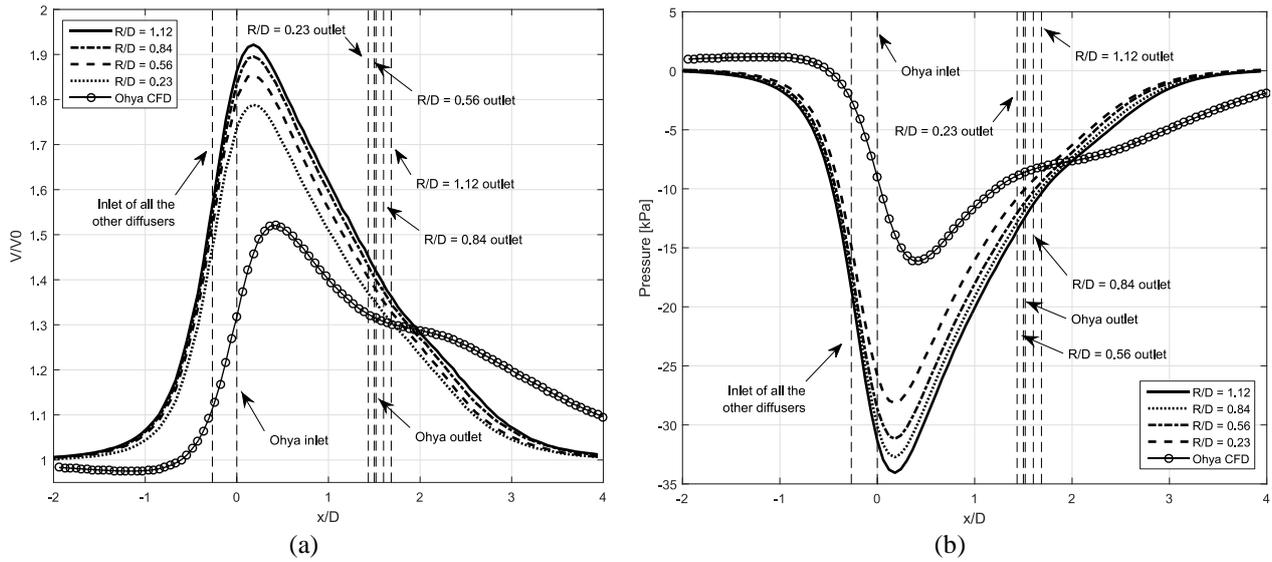
Figure 6. (a) Velocity profiles of the designs (b) Pressure profile of the designs

As shown through the streamlines in Fig. 7, the flange creates a substantial flow separation in both cases, but in the new design the affected region seems to be smaller, due to the outer flange being smaller than the flange of the other configuration. Since the flow acceleration produced by the double-flanged diffuser is greater than the value previously obtained with the single-flanged design, the streamline analysis indicates that another factor is playing a significant role in the pressure drop that is present in the wake region due the flow separation, besides the h/D ratio. This is a very low pressure region, and it generates a pressure gradient between the diffuser inlet and outlet. Due to this gradient, the flow accelerates inside the diffuser, thus increasing the velocity ratio.
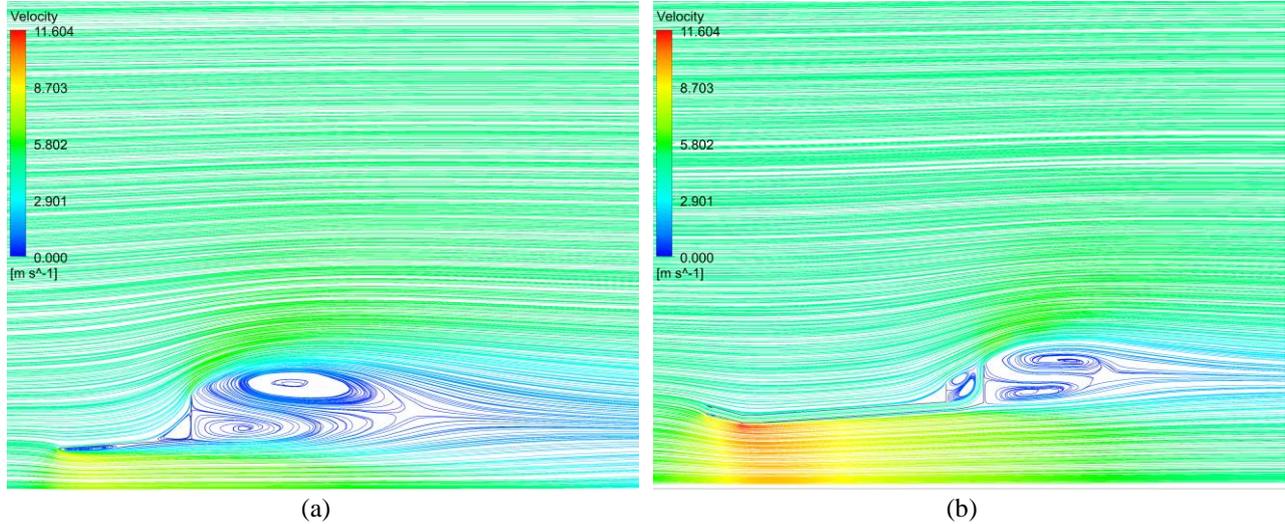

Figure 7. (a) Velocity contour for the single-flanged diffuser. (b) Velocity contour for the double-flanged diffuser.

The zoomed-in streamline plots shown in Fig. 8 give a deeper understanding of why the double-flanged design is capable of producing such increase in the flow velocity. The distance between the inner and outer diffusers allowed water to flow in. When the flow reached the region between the flanges, some recirculation began to occur. Two vortex were generated and interacted both with the free stream flow and with each other. This interaction directly interfered with the level of turbulence of the incoming flow, making up for the seemingly smaller flow separation. Consequently, the pressure drop also increased. Figure 9 illustrates the difference between the wake region in terms of pressure.
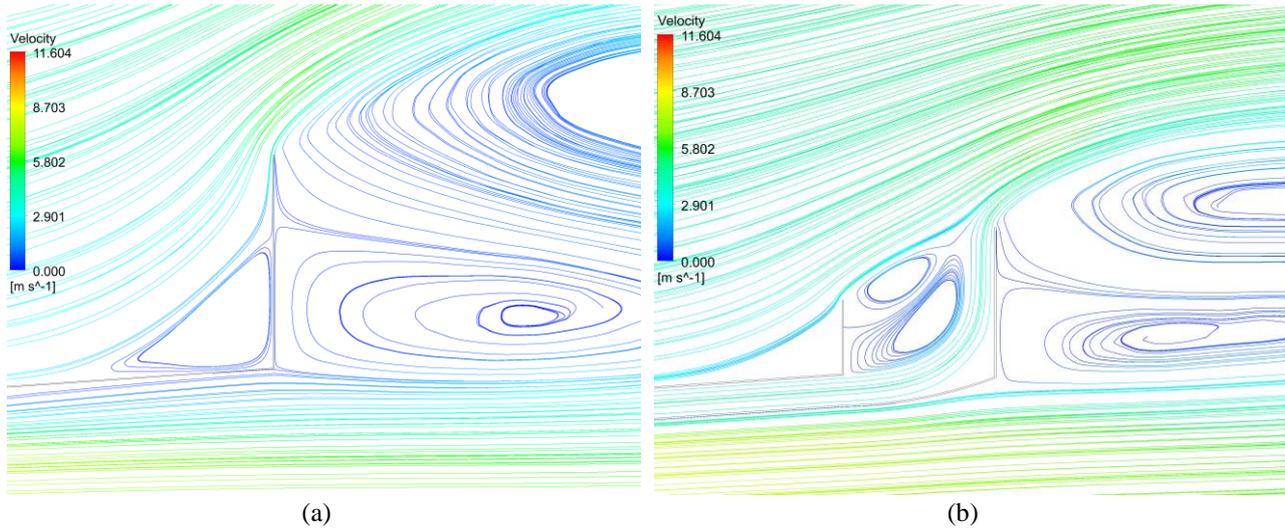
Figure 8. (a) Details of the vortex in the single-flanged diffuser. (b) Details of the vortex in the double-flanged diffuser.
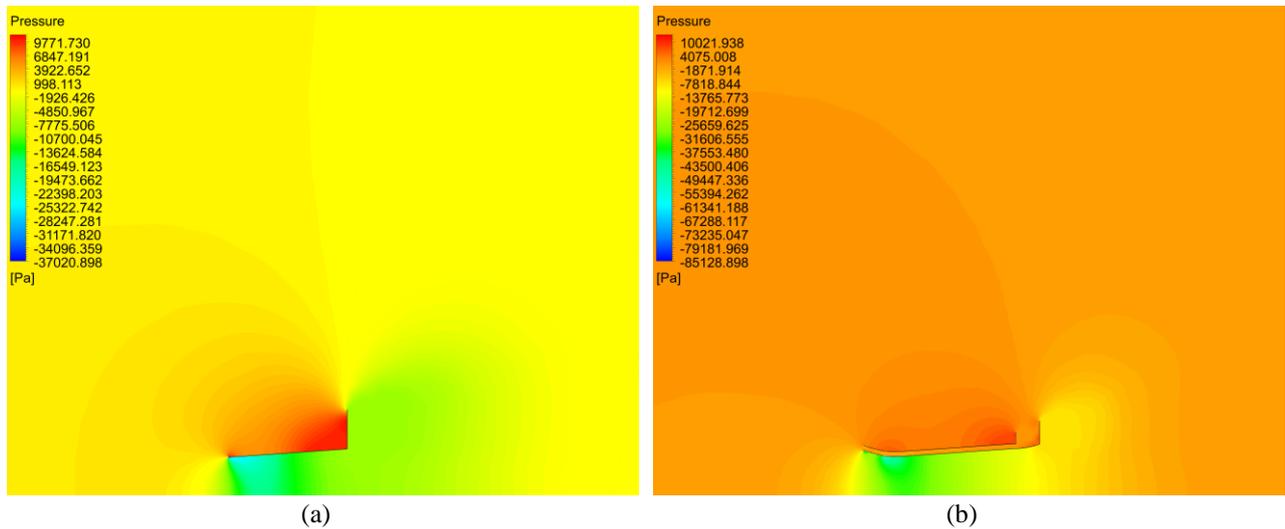

Figure 9: (a) Pressure contour for the single-flanged diffuser. (b) Pressure contour for the double-flanged diffuser.

The diffuser efficiencies of both designs were calculated, using the formula provided by Vaz & Wood (2018). The velocities $\bar{V}_2$ and $\bar{V}_3$ were calculated using a mass flow average, while $\bar{p}_2$ and $\bar{p}_3$ were calculated using an area average. The specific mass is 998.2 kg/m³.

$$\eta_d = \frac{\bar{p}_3 - \bar{p}_2}{\frac{1}{2}\rho(\bar{V}_2^2 - \bar{V}_3^2)} \tag{2}$$

Table 3 shows the values obtained. As it can be seen, the double-flanged design exhibits greater efficiency, which correlates directly to the increase in the pressure drop and in the velocity ratio. The thrust and thrust coefficient are also displayed.

Table 3. Diffuser Characteristics

Diffuser	η_d	T_d [N]	C_{Td}
Ohya	0.803203	2178.121	0.1745639
Double-flanged	0.977627	802.09182	0.064283055

3.2 Application on a Hydrokinetic Turbine

It is well known that the classical BEM theory does not consider losses, so that a turbine is modeled in free flows, without restriction. For DAHTs, the classical formulations need to be modified, as the diffuser behaves as a duct. Therefore, according to Vaz & Wood (2018), to model a diffuser with losses, the turbine is taken as if in a duct, in which the presence of losses are taken into account. Recently, Silva et al. (2018), demonstrated a very interesting BEM model extended to DAHTs. Their approach account on diffuser efficiency, area ratio, and thrust. Therefore, to evaluate the impact of the double-flanged diffuser proposed here, the horizontal axis hydrokinetic turbine designed by Silva et al. (2018) as shown in Fig. 10 is employed.
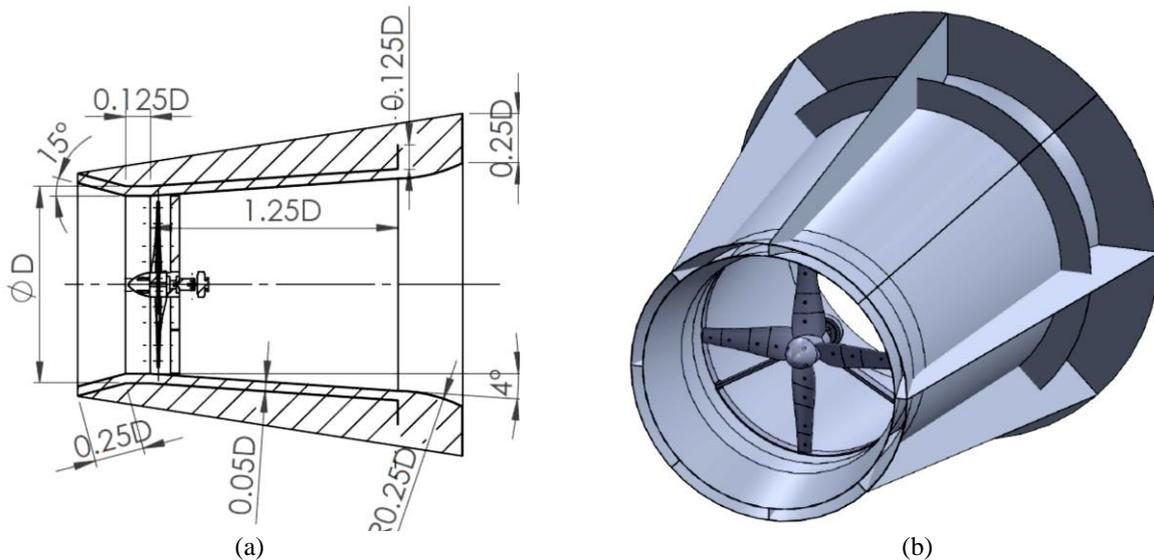

(a)   (b)

Figure 10. (a) Cross sectional view (b) Turbine inside the diffuser.

4. CONCLUSIONS

The main conclusions derived from this study are:
• The double-flanged design is remarkably different from conventional designs. The results obtained after the computational simulations also showed that it is capable of providing a greater velocity ratio.
• The efficiency of the double-flanged diffuser is also greater than that of the single-flanged design, as expected. Due to its geometry, the double-flanged design also has more drag.
• The extra flange increased the low-pressure region in the diffuser wake, which has direct correlation to the increase in the velocity ratio.

5. ACKNOWLEDGEMENTS

6. REFERENCES

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

Barbosa, D. L. M., Vaz, J. R. P., Figueiredo, S.W.O., Silva, M. O., Lins, E. F. and Mesquita, A. L.A., 2015. "An Investigation of a Mathematical Model for the Internal Velocity Profile of Conical Diffusers Applied to DAWTs". *Annals of the Brazilian Academy of Sciences*, Vol. 87, pp. 1133-1148.

Fluent version 12.0, ANSYS FLUENT Theory Guide, ANSYS, Inc.

Gaden, D. L.F. and Bibeau, E. L., 2010. "A numerical investigation into the effect of diffusers on the performance of hydro kinetic turbines using a validated momentum source turbine model". *Renewable Energy*, Vol. 35, pp. 1152-1158.

Hansen, M.O.L., Sorensen, N.N., Flay, R.G.J., 2000. "Effect of placing a diffuser around a wind turbine." *Wind Energy* Vol.3, pp. 207-213

Jafari, S.A.H., 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, pp. 102-110.

Ohya, Y., Karasudani, T., Sakurai, A., Abe, K. and Inoue, M., 2008. "Development of a shrouded wind turbine with a flanged diffuser." *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96, No. 5, pp. 524–539.

Riglin J., Schleicher, W. C. and Oztekin, A., 2014. "Diffuser Optimization For a Micro-Hydrokinetic Turbine", *Proceedings of the ASME 2014 International Mechanical Engineering Congress and Exposition.*

Silva, P. A. S. F., Rio Vaz, D. A. T. D., Britto, V., Oliveira, T. F., Vaz, J. R.P. and Brasil Junior, A.C.P., 2018. "A new approach for the design of diffuser augmented hydro turbines using the blade element momentum". *Energy Conversion and Management*, Vol. 165, pp. 801-814.

Vaz, J. R. P. and Wood, D. H., 2018. "Effect of the diffuser efficiency on wind turbine performance". *Renewable Energy*, Vol. 126, pp. 969-977.

Wilcox, D. C., 1993-1994. *Turbulence Modeling for CFD*, DCW Industries, Inc.

7. RESPONSIBILITY NOTICE

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