DEVELOPMENT OF A NUMERICAL MODEL FOR THE STUDY OF AN OSCILLATING WATER COLUMN DEVICE CONSIDERING AN IMPULSE TURBINE

A. L. dos Santos[a], L. A. Isoldi[a], L. A. O. Rocha[b], M. N. Gomes[c], R. S. Viera[d], and E. D. dos Santos[a]

[a]Universidade Federal do Rio Grande
Escola de Engenharia
Av. Itália, Km 8, CEP 96.201-090
Rio Grande, Rio Grande do Sul, Brasil
amandalopesq@hotmail.com
liercioisoldi@furg.br
elizaldosantos@furg.br
[b]Universidade do Vale dos Sinos
laorocha@gmail.com
[c]Instituto Federal do Paraná
mateus.gomes@ifpr.edu.br
[d]Universidade Federal do Rio Grande do Sul
Departamento de Engenharia Mecânica
Rua Sarmento Leite, 425, CEP: 90050-170
Porto Alegre, RS, Brasil
rodrigospotorno@yahoo.com.br

Received: March 21, 2019
Revised: March 26, 2019
Accepted: April 22, 2019

ABSTRACT

The present work brings a numerical study of an energy conversion device which takes energy from the waves through an oscillating water column (OWC), considering an impulse turbine with rotation in the chimney region through the implementation of a movable mesh model. More precisely, a turbulent, transient and incompressible air flow is numerically simulated in a two-dimensional domain, which mimics an OWC device chamber. The objectives are the verification of the numerical model with movable mesh of the impulse turbine in the free domain from the comparison with the literature and, later, the study of the impulse turbine inserted in the geometry of the OWC device. In order to perform the numerical simulation on the generated domains, the Finite Volume Method (FVM) is used to solve the mass and momentum conservation equations. For the closure of the turbulence, the URANS (Unsteady Reynolds Averaged Navier-Stokes) model k-ω SST is used. To verify the numerical model employed, drag coefficients, lift, torque and power are obtained and compared with studies in the literature. The simulations are performed considering a flow with a Reynolds number of $Re_D = 867{,}000$, air as the working fluid and a tip speed ratio of $\lambda = 2$. For the verification case, coefficients similar to those previously predicted in the literature were obtained. For the case where the OWC device was inserted it was possible to observe an intensification of the field of velocities in the turbine region, which led to an augmentation in the magnitude of all coefficients investigated (drag, lift, torque and power). For the case studied with the tip velocity ratio $\lambda = 2$, results indicated that power coefficient was augmented, indicating that the insertion of the turbine in a closed enclosure can benefit the energy conversion in an OWC device.

Keywords: oscillating water column; impulse turbine; numerical study; turbulent flow

NOMENCLATURE

A	Area, m^2
A	Distance, m
c	Chord length, m
Cd	Drag coefficient
Cl	Lift coefficient
Cp	Power coefficient
Ct	Torque coefficient
D	Diameter, m
e	Thickness, m
H	Height, m
h	Chimney height, m
k	Turbulent kinetic energy, m^2/s^2
L	Length, m
l	Chimney length, m
OWC	Oscillating Water Column
p	Pressure, Pa
Pr_t	Turbulent Prandtl number
Re_D	Reynolds number
s	Overlap, m
t	Time domain, s
u	Velocity in x direction, m/s
v	Velocity in y direction, m/s
V	Air flow velocity, m/s
V_o	Undisturbed wind velocity, m/s
x	spatial coordinate, m
y	spatial coordinate, m
w	Angular velocity, rad/s
$\overline{(\ )}$	Time-averaged variables

Greek symbols

α_t	Turbulent diffusivity, m^2/s
β	Thermal expansion coefficient, K^{-1}
λ	Tip speed ratio

μ_t Turbulent viscosity, kg/ms
μ Dynamic viscosity, kg/ms
μ_t Turbulent dynamic viscosity, kg/ms
υ Kinematic viscosity, m²/s
ρ Fluid density, kg/m³
ω Specific dissipation rate, 1/s

INTRODUCTION

Current life standards show a growing energy dependency and demand. The tendency is that the usage will grow even more and alongside the need to search for low cost and low environmental impact energetic solutions. One of the alternatives is the conversion of energy of ocean in electric one. That kind of energy is considered clean with high energetic density and abundantly distributed along the world, which might have a prominent place in the global energetic matrix. Amongst the energy types extracted from the oceans, one of the most studied is the wave energy. However, unlike what has been observed in other ways of conversion (for example, wind energy), there is no dominant technology for the conversion of waves energy (Falcão, 2010). Consequently, the development of modeling that represent the devices in a more adequate way is considered an important research subject.

The Oscillating Water Column (OWC) is one of the most studied wave energy converters. Figure 1 illustrates a domain with the main operational principle of this device. On this device, the kinetic energy from the air movement inside the chamber is converted into mechanical energy of turbine rotation and, afterwards, into electrical energy. The OWC devices consist in two partially submerged structures, which are open to the sea below waterline. The movement of the waves on the outside causes the movement of the water column inside the structure, so as to alternatively pressurize and depressurize the air inside the device, generating a pulsating flow passing through a turbine. Due to this flow condition, it becomes necessary to use a turbine which can be maintained in the direction of rotation, regardless of the flow direction (Gomes et al., 2018).

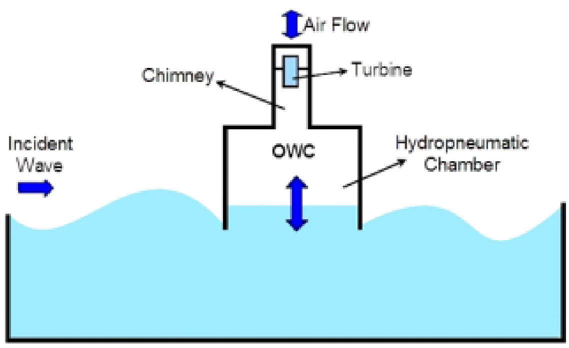

Figure 1. Illustration of the main operational principle of the OWC device.

The turbine is considered the most critical element in the composition of an energy conversion system in an OWC device. Since it is subject to operating conditions that are more demanding than in other applications, such as the direct contact with salinity of the sea and the highly variable flow over time. Recently, the impulse axial turbine has been considered as a promising alternative comparing to the Wells to equip OWC systems, given its higher range of efficient and stable operation and it's gifted with the capacity of dealing with high intensity waves, in conditions which the Wells turbine causes maintenance issues and noises (Suleman, 2011).

The development of computational modeling to simulate oscillating water column devices considering the rotating turbine still represents a big challenge, which is evident in the recent studies of Hashem et al. (2018) and Prasad et al. (2018). In the first study, a numerical study was developed simulating an idealized axial turbine with static blades in relation to the turbulent flow, obtaining theoretical recommendations regarding the strength of the turbine and the blade's profile type on the turbine performance. In the second study, a numerical study was developed considering the Savonius turbine immersed in wave tank. It was evaluated the effects of the immersion depth of turbine on the propeller power. However, to the knowledge of authors, there has not been observed the development of numerical models specifically for OWC devices.

Therefore, in this study a numerical simulation of an OWC device considering an impulse turbine is developed. Since numerical studies with the geometry model used considering a turbine was not found, the reproduction of simulations of impulse turbines in open area for verification of the computer model was necessary. Results were compared with the studies of Akwa (2010) and Silva (2010). The comparisons were made through mean values of drag, lift, torque and power coefficients.

PROBLEM DESCRIPTION

The problem consists of the air flow in the chamber and, as consequence, on the impulse turbine of the CAO device, considering a transient, incompressible, turbulent, two-dimensional regime with constant thermophysical properties. The modeling adopted is based on the solution of time-averaged Navier-Stokes equations. Figure 2 represents the computational domain with its main geometric variables. The length of the hydro pneumatic chamber (L) and height (H), the chimney outlet length (l) and its height (h) of the computational domain are given, respectively, by: L = 10 m, H = 6 m; l = 2 m and h = 6 m.

The representation of the turbine rotor becomes necessary, which in this case is of the vertical axis type, composed of two blades separated from each other by 90° and of semicircular type, as shown in

Fig. 3.

In the study, the thickness of blade is considered e = 0.0072m and the rounded tips is arranged in a circumference of diameter D = 1.8 m. The distance between the blades (a) is zero (a = 0) and the overlap s is equal to s = 0.144 m, which represents 15% of the stipulated chord size, which in turn is 0.972 m.

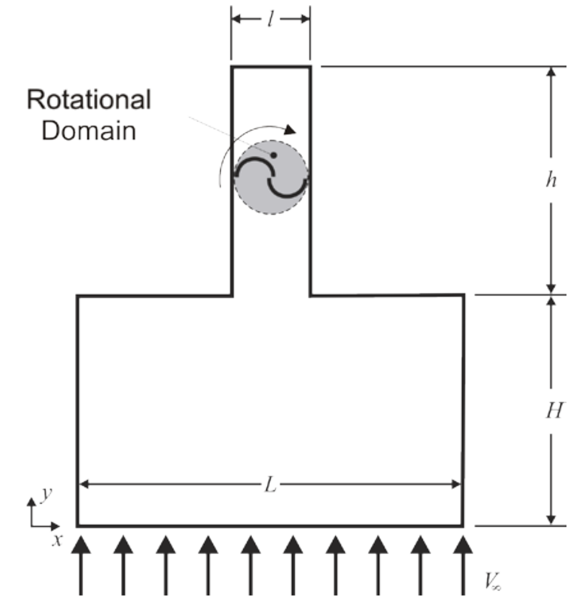

Figure 2. Schematic representation of the computational domain of the OWC device.

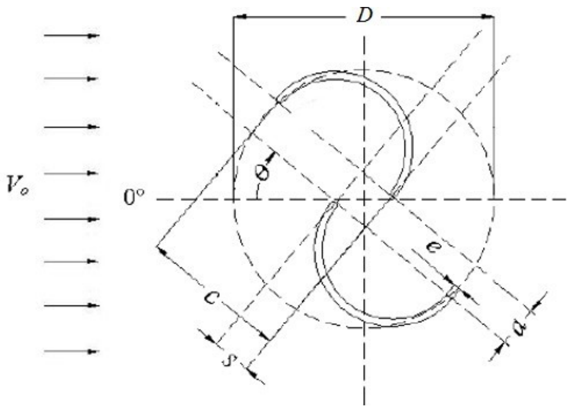

Figure 3. Schematic representation of a rotor of an impulse turbine. (Adapted from Akwa, 2010)

In the case studied, the flow is caused by the imposition of a constant velocity at the inlet of the domain (lower line) of V_0 = 1.4 m/s, so that Re_D = 867,000 is obtained in the chimney region of the OWC device. At the outlet of the domain (upper line) a null (atmospheric) pressure condition is imposed and on the lateral surfaces the non-slip and impermeability condition (for the OWC) is established. For a verification, a flow is studied on the free turbine, where the outer surfaces have a symmetry condition. A non-slip and impermeability boundary condition is imposed in the rotor surfaces. The rotor is surrounded by an interface boundary condition, which allows the specification of its angular velocity as w = 15.5 rad/s. For this case, once there is no restriction of the flow due to chamber and chimney, it is imposed a free stream velocity of V_0 = 7.0 m/s, leading to the same Reynolds number simulated in OWC case.

For comparative studies of impulse turbines and to obtain the power coefficient, it is of great importance the calculation of the dimensionless that relates the angular velocity of the rotor to the undisturbed wind speed, called the tip speed ratio, presented by:

$$\lambda = \frac{wR}{V_0} \tag{1}$$

where w is the angular velocity of the rotor [rad/s], R is the radius of the rotor [m] and V_o is the free stream velocity of the air flow [m/s]. In the present work, the magnitude of the tip speed ratio is defined as λ = 2. Concerning the thermophysical properties of air, a density of ρ = 1.18415 kg/m^3 and a dynamic viscosity of μ = 1,7894 × 10^{-5} kg/(ms) is considered.

MATHEMATICAL AND NUMERICAL MODELING

For the analysis of incompressible and transient flows in a two dimensional domain, it is solved the time-averaged conservation equations of mass and momentum in x and y directions, given respectively by (Pope, 2000):

$$\frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} = 0 \tag{2}$$

$$\frac{\partial\left(\rho\bar{u}\right)}{\partial t} + \frac{\partial\left(\rho\overline{uu}\right)}{\partial x} + \frac{\partial\left(\rho\overline{vu}\right)}{\partial y} = -\frac{\partial \bar{p}}{\partial x} + \left(\mu + \mu_t\right)\left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2}\right) \tag{3}$$

$$\frac{\partial\left(\rho\bar{v}\right)}{\partial t} + \frac{\partial\left(\rho\overline{uv}\right)}{\partial x} + \frac{\partial\left(\rho\overline{vv}\right)}{\partial y} = -\frac{\partial \bar{p}}{\partial y} + \left(\mu + \mu_t\right)\left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2}\right) \tag{4}$$

where x represents the cartesian spatial coordinate in the direction of the x-axis [m]; u is the velocity component in the direction of the x-axis [m/s], y represents the spatial cartesian coordinate in the y-axis direction [m] and v is the velocity component in the y direction [m/s], p is the pressure [N/m²] and the overlay bar (-) indicates the average operator.

The turbulence model adopted for modeling the

Reynolds tensions in the present work is the turbulence model k - ω SST of Menter (2003). This closure modeling was previously employed in the work of Akwa (2010) leading to results close to those obtained experimentally in the literature.

The k-ω model aims to correlate the Reynolds tensions with the mean field strain rate by means of a turbulent viscosity. In this way, the turbulent viscosity is given by:

$$\mu_t = \frac{\bar{\rho}\alpha_1 k}{\max(\alpha_1\omega, SF_2)} \tag{5}$$

Equation (5) is then introduced into the k - ω model to generate the new turbulent kinetic energy equation given by:

$$\frac{\partial k}{\partial t} + \frac{\partial(\tilde{u}_j k)}{\partial x_i} = \tilde{P}_k - \frac{k^{\frac{3}{2}}}{L_T} + \frac{\partial}{\partial x_i}\left[(\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_i}\right] \tag{6}$$

And the equation of the specific dissipation rate is given by:

$$\frac{\partial \omega}{\partial t} + \frac{\partial(u_i\omega)}{\partial x_i} = \left(\frac{\alpha}{\mu_t}\right)\tilde{P}_k - \beta\omega^2 + \frac{\partial}{\partial x_i}\left[(\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right] + 2(1 - F_1)\frac{\sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{7}$$

where: k is the turbulent kinetic energy [m²/s²], it is a limiting function that prevents the generation of turbulence in regions of stagnation, ω is the specific dissipation rate, μ_t is turbulent viscosity, $\beta = 0.09$, $\alpha_1 = 5/9$, $\beta_1 = 3/40$, $\sigma_k = 0.85$, $\sigma_\omega = 0.5$, $\sigma_2 = 0.44$, $\beta_2 = 0.0828$, $\sigma_2 = 1$, $\sigma_{\omega 2} = 0.856$ and F_1 and F_2 are functions of combining variables and constants defined by:

$$F_1 = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^*\omega y'}, \frac{500\nu}{y^2\omega}\right), \frac{4\rho\sigma_{\omega 2}k}{CD_{k\omega}y^2}\right]\right\}^4\right\} \tag{8}$$

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y'}\frac{500\nu}{y^2\omega}\right)\right]^2\right] \tag{9}$$

The time-averaged conservation equations of mass and momentum as well as those of the adopted turbulent closure model are solved with the Finite Volume Method, more precisely with CFD code FLUENT 14.0 (Versteeg and Malalasekera, 2007; FLUENT, 2011). The simulations were performed using computers with an Intel® Core ™ i7 5820K @ 3.30 GHz six-core processor and 16.0 Gb of RAM. To reduce the simulation time, the parallel processing technique was adopted.

The solver used in this study is the Pressure-Based. A 2nd Order Upwind advection scheme is used. Moreover, the SIMPLE pressure-velocity coupling is defined. The solutions are considered converged when the residuals of mass conservation equations, momentum, turbulent kinetic energy and specific dissipation rate reached values less than 10^{-5}. A time step of $\Delta t = 1.75 \times 10^{-3}$ s was adopted and the final simulation time was $t_f = 1.75$ s.

RESULTS AND DISCUSSION

Initially, it is studied a turbulent air flow over an impulse turbine inserted in a free domain, reproducing the study developed by Akwa (2010). This case will be hereafter called verification case. The same fluid dynamic conditions (initial and boundary conditions) are applied and drag, lift, torque and power coefficients are measured and compared with those achieved in Akwa (2010) with the purpose to verify the present numerical model. Afterwards, drag, lift, torque and power coefficients are achieved for an enclosed domain which mimics an Oscillating Water Column (OWC) device. The parameters obtained for the simulation with free domain are used as reference to observe the influence of enclosed domain over the fluid dynamic behavior of turbulent air flow.

For all simulations the same materials and discretization of computational domain discretization (mainly in the region of turbine) were used. The conditions and parameters imposed on the drainage are presented in Tab. 1.

Table 1. Conditions imposed on simulations.

Air inlet speed (Free domain)	$V_0 = 7.0$ m/s
Air inlet speed (OWC domain)	$V_0 = 1.4$ m/s
Rotor diameter	D = 1.8 m
Reynolds number	$Re_D = 867{,}000$
Pressure outlet	p = 101,325 Pa
Ingress turbulence intensity	IT = 1.0 %
Characteristic length of the turbulence	0.01 m

RESULTS FOR VERIFICATION CASE

In the simulations with the free domain turbine, the domain has dimensions of H = 12D and L = 26D. These large dimensions are defined in order to avoid the influence of boundary conditions in the turbine region, allowing a most suitable measure of time and spatial averaged coefficients.

In order to identify whether the phenomenology of the problem is suitably represented, Fig. 4 represents the velocity field in different instants of time. It can be observed an incidence of air flow on the surface of the blades of the turbine, generation of vortex streets behind the turbines and regions of low velocities in the concave part of turbine blades (which indicates high pressure in this region). In general, the velocity fields obtained with the present model are in agreement with those predicted in literature.

The fluctuating behavior caused by the vortex streets causes the coefficients of drag, lift and torque to also assume such behavior. In this way, it is necessary to obtain an average RMS (Root Mean Square) of the coefficients. For the present work a range of Δt = 1.0 s was used to obtain the RMS coefficients. In addition, a spatial average process is performed on the turbine blades to obtain the coefficients studied.

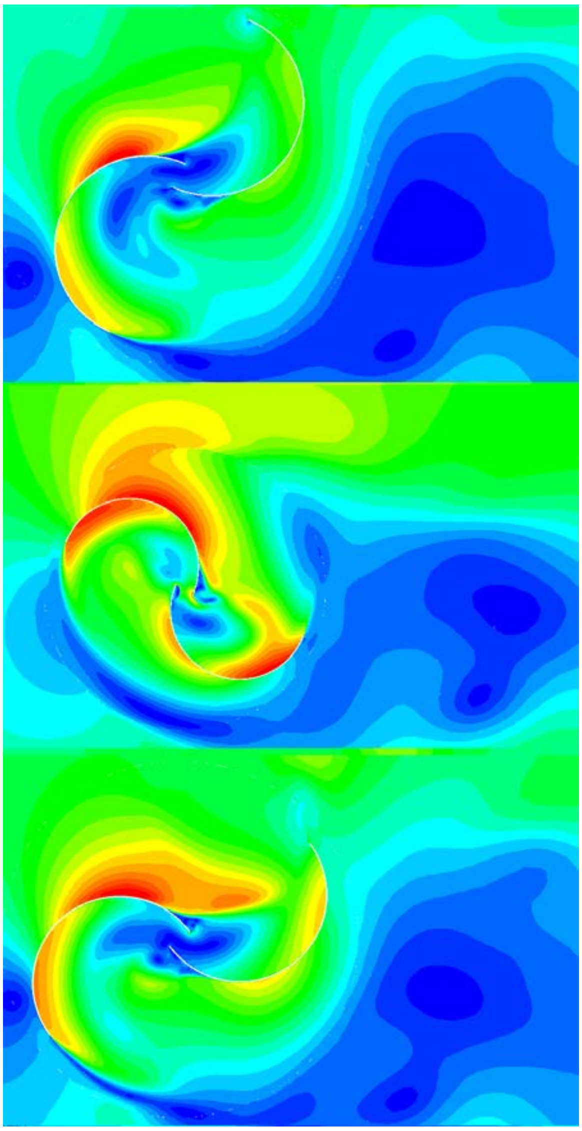

Figure 4. Field of velocities in the flow with free turbine in three different instants of time.

Figure 5 presents a comparison of the coefficients of drag (Cd), lift (Ct), torque (Ct) and power (Cp) obtained in the present work and the numerical results of Akwa (2010). As can be observed, there is a behavior very similar to the values found by Akwa (2010) and those obtained in the present study. It is worth mentioning that the coefficients of torque (Ct) and power (Cp) had very low magnitudes. In the present model positive coefficients were obtained, which indicates that the turbine is receiving energy from the flow, and in the literature results negative values were obtained, which indicates that the turbine is supplying energy for the flow (turbo-machine behavior). This type of magnitude is expected for this magnitude of tip speed ratio, λ = 2.0. Despite some differences, it is possible to affirm that the numerical model employed had adequate behavior for the prediction of the coefficients and the phenomenology of the proposed problem.

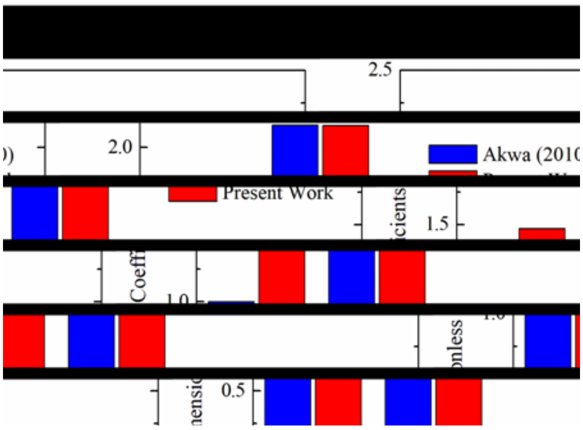

Figure 5. Comparison between the coefficients of drag (Cd), lift (Cl), torque (Ct) and power (Cp) obtained in this work and those presented by Akwa (2010).

RESULTS FOR THE SIMULATION IN THE OWC DEVICE

After verification of the numerical model, simulations of the OWC device with a coupled impulse turbine are performed and the velocity and pressure fields obtained for the same configurations already mentioned above. In the results presented by Fig. 6, a large increase in the magnitude of the velocity field in the turbine region is observed, caused by the restriction of the flow from the chamber to the chimney.

It is also possible to observe in Fig. 7 that the pressure field has similar behavior to that found in other works developed with the impulse turbine model, presenting a zone of higher pressure in the vicinity of the blades and one of lower pressure after the fluid passage in them. The difference in pressure between rotor blades can also be observed, according to Silva Júnior (2010). Such a difference between blade surfaces generates pressure drag, contributing to the driving force of the device.

The interference caused by the walls of the domain in the flow velocities also reflected in a considerable increase in the coefficients of drag, lift, torque and power. The values for the coefficients are presented in Tab. 2 and compared with those obtained in the case of verification. It should be noted that the value for the power coefficient was obtained by multiplying between λ and the torque coefficient.

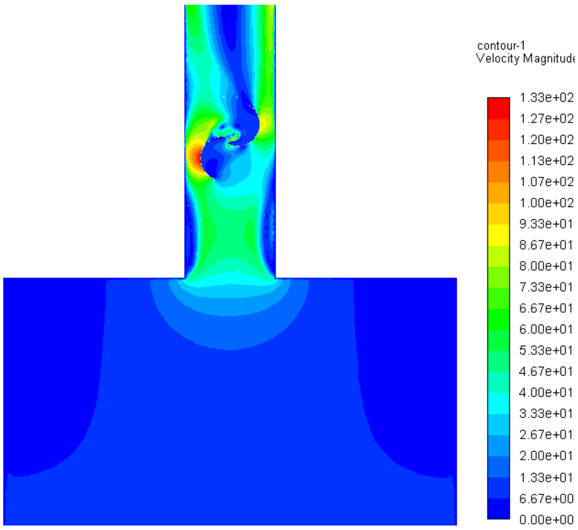

Figure 6. Field of velocities in the OWC device with the coupled impulse turbine.

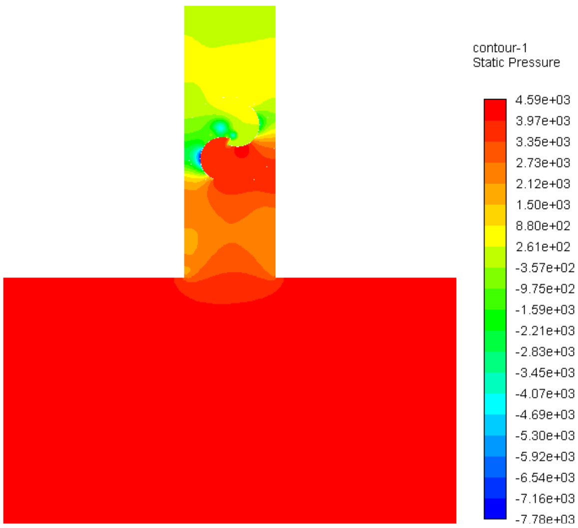

Figure 7. Pressure field in the OWC device with the coupled impulse turbine.

As can be observed, the geometric change and the simulation in an enclosed domain influenced considerably the coefficients for the same proposed rotor. The concentration of fluid flow in the chimney enclosure causes an intensification of pressure difference between the concave and convex sides leading, mainly in the driven blade, which leads to an increase of torque and power measured in the turbine. It may also be observed that the geometry of the device causes a blocking effect, so that the undisturbed velocity of the wind (V_0) arriving in the vicinity of the blades of the rotor is not equal to the speed prescribed at the entrance of the domain, causing consequently an increase in the tip speed ratio value (λ).

Table 2. Drag coefficients (Cd), lift (Cl), torque (Ct) and power (Cp) values for the CAO device and free turbine.

	OWC	Free turbine
Drag coefficient (Cd)	11.4900	1.4715
Lift coefficient (Cl)	-4.7400	2.1669
Torque coefficient (Ct)	1.4300	0.0400
Power Coefficient (Cp)	2.8700	0.0806

CONCLUSIONS

In this study, a numerical simulation was developed to simulate turbulent flows that mimics the chamber region of an oscillating water column device that converts wave energy in electrical one. In the first step of the study, the verification of a model simulation for an impulse turbine was conducted according to the results obtained in Akwa (2010), using the Finite Volume Method. For closure of turbulence, it was employed the k – ω SST model. Flows with a Reynolds number of Re_D = 867.000 and a tip speed ratio of λ = 2 were considered. Two case studies were performed, the first one to verify the numerical model simulating turbulent flow over impulse turbine in a free domain and the second one a domain similar to that found in the oscillating water column device.

Results indicated that, although some difference found in the drag coefficients, the coefficients obtained in the present work were close to the ones obtained by Akwa (2010), which verify the simulation that was utilized in this work. When the same turbine was inserted in a domain that simulates the OWC device chamber, a considerable raise was observed on the intensity of the flow in the chimney region, mainly around the turbine. As a consequence, there was a significant raise on the magnitude of the drag coefficients, lift, torque and power in comparison to the free turbine case (which was not under influence of the side walls).

Therefore, the results obtained in this study contributed to improve the knowledge of the main operational principle of an impulse turbine inside an OWC device. In future works it is intended to study the influence of tip speed ratio in main coefficients. Moreover, it is intended to impose a more realistic wave velocities in the lower surface of the domain which simulates the OWC, in order to observe the influence of pulsating flow over the fluid dynamic behavior and available power of device.

ACKNOWLEDGEMENTS

E. D. dos Santos, L. A. Isoldi and L. A. O. Rocha thank CNPq (Brasília, DF, Brasil) for research grants (Processes: 306024/2017-9, 306012/2017-0, 307847/2015-2).

REFERENCES

Akwa, J. V., 2010, Análise Aerodinâmica de Turbinas Eólicas Savonius empregando Dinâmica dos Fluídos Computacional, Master Thesis, Universidade Federal do Rio Grande do Sul, Porto Alegre, Brasil. (*in Portuguese*).

Falcão, A. F. de O., 2010, Wave Energy Utilization: a Review of the Technologies, Renewable and Sustainable Energy Review, Vol. 14, pp. 899- 918.

FLUENT, 2011, *User's Guide*, ANSYS Inc.

Gomes, M. N., Lorenzini, G., Rocha, L. A. O., Dos Santos, E. D., and Isoldi, L. A., 2018, Constructal Design Applied to the Geometric Evaluation of an Oscillating Water Column Wave Energy Converter Considering Different Real Scale Wave Periods, Journal of Engineering Thermophysics, Vol. 27, pp. 173-190.

Hashem, I., Abdel Hameed, H. S., and Mohamed, M. H., 2018, An Axial Turbine in an Innovative Oscillating Water Column (OWC) Device for Sea-Wave Energy Conversion, Ocean Engineering, Vol. 164, pp. 536-562.

Menter, F. R., Kuntz, M., and Langtry, R., 2003, Ten Years of Industrial Experience with the SST Turbulence Model, Turbulence, Heat and Mass Transfer 4, Begell House Inc., pp.625-632.

Pope, S. B., 2000, *Turbulent Flows*, Cambridge University Press, Cambridge, UK.

Prasad, D. D., Ahmed, M. R., and Lee, Y-H., 2018, Studies on the Performance of Savonius Rotors in a Numerical Wave Tank, Ocean Engineering, Vol. 158, pp. 29-37.

Silva Júnior, G. A., 2010, Análise da Influência de Diferentes Razões de Sobreposição no Desempenho de Turbinas Eólicas Savonius empregando Dinâmica dos Fluidos Computacional, Course Completion Work in Mechanical Engineering, UFRGS, Porto Alegre, RS. (*in Portuguese*).

Suleman, Z., and Khaleeq, H. B., 2011, Comparative Study of Power Take-Off Units of OWC Based Wave Energy Power Plants, Technical Journal, University of Engineering and Technology Taxila, pp. 110-120.

Versteeg, H. K., and Malalasekera, W., 2007, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Pearson, England.

Wilcox, D. C., 2002, *Turbulence Modeling for CFD*, 2nd Edition, DCW Industries, La Canada.