COMPUTATIONAL MODELING OF THE AIR-FLOW IN AN OSCILLATING WATER COLUMN SYSTEM

Mateus das Neves Gomes, mateusufpel.gomes@gmail.com
Claudio Rodrigues Olinto, claudioolinto@furg.br
Liércio André Isoldi, liercioisoldi@bol.com.br
Jeferson Avila Souza, jasouza1974@gmail.com
Luiz Alberto Oliveira Rocha, laorocha@gmail.com
Universidade Federal do Rio Grande - Programa de Pós Graduação em Modelagem Computacional (PPGMC) - Rio Grande - RS

Abstract. *Several alternatives for electric power production have been studied in the last decades. Because of the huge energetic resources stored in the oceans in the form of wave — about 2TW — value that is compared to the annual rate of electric power used in the earth, the conversion of the wave's energy of the oceans in electric power comes up important as one of these alternatives. One of the ways to make that conversion is through the oscillating water column (OWC) system: the wave enters into the hydro-pneumatic chamber (resembling a cave with entry below the waterline) and the up-and-down movement of water column inside the chamber makes air flow to and from the atmosphere, driving an air turbine. The turbine is symmetric and is driven indifferently in which direction the air flows. This paper presents the computational modeling of the air flow in a oscillating water column chamber using two different methodologies: in one of them it is considered just the chamber, varying the velocity in its entrance according to the wave's equation, considering just the air, and a new one considering the chamber put into a wave's tank, so it takes in account the complete interaction between water and air into the chamber. In this method, to consider the water and air it is used the multiphase model volume of fluid (VOF). It was simulated the same geometric compound of an oscillating water column system with a vertically placed tower, in order to compare these two different numerical models. It is noted that the dimensions of the tested chamber are in laboratory scale and the proposed model was used to simulate a 2D case. It was used GAMBIT® software for geometry creation and mesh generation, while FLUENT® package was employed for solving the conservation equations and analysis of the results.*

Keywords: *Wave Energy, Oscillating Water Column, Numerical Simulation, Numerical Wave Tank, Volume of Fluid.*

1. INTRODUCTION

Would it be possible to imagine the life on earth if there was not electric energy? Would the man have aimed any technological evolution?

The imposed challenges by need of fixing politics that assure their sustainable development are particularly pertinent in the energy's domain. More and more we are faced to request finding in renewable energies, a real and reliable alternative for conventional ways of producing electric energy, most of them responsible for hard treats to environment. The obligation fixed by law and by the Quioto protocol just reinforces this need. The oceans has a great energetic potential that can be used in a significant way to supply the growing needs of energy in the world (Cruz and Sarmento, 2004).

Brazil, in special, has a large potential to transform the ocean energy in electric energy, so its huge coast could be explored.

Energy in oceans encloses ocean thermal energy, tidal energy, wave energy and energy of marine currents. Great efforts have been made during the last decades, especially in the field of wave energy for conversion purpose to useful energy. Wave energy is derived from the winds as they blow across the oceans, and this energy transfer provides a convenient and natural concentration of wind energy in the water near the free surface. The energy fluxes occurring in deep water sea waves can be very large. The power in a wave is proportional to the square of the amplitude and to the period of the motion. Therefore, long period (~7–10 *s*), large amplitude (~2 *m*) waves have energy fluxes commonly averaging between 40 and 70 *kW* per m width of oncoming wave. Nearer the coastline the average energy intensity of a wave decreases due to interaction with the seabed. Energy dissipation in near shore areas can be compensated by natural phenomena such as refraction or reflection, leading to energy concentration ("hot spots") (Clément *et al.*, 2002).

According to Cruz and Sarmento (2004) there are several technologies used to convert the sea wave energy, which can be classified as follow: devices on-shore, near-shore, and off-shore. Another way to classify the devices of wave's energy conversion is related to its physical principle of working: Oscillating Water Column, Surging devices, and Overtopping devices.

Conde and Gato (2008) developed a 3D numerical simulation of the air flow in an oscillating water column (OWC) device considering only the air behavior, varying its velocity on the chamber's entrance according to the ocean wave's equation. Marjani *et al*. (2008) accomplished a study quite similar, where only the air behavior in OWC device was considered, however an improvement in the air entrance was applied (a dynamic mesh in the inlet region). Liu *et al.* (2008b), using the Volume of Fluid (VOF) method, performed the numerical simulation of a wave tank, comparing the generated wave with the analytical solution. Besides, an OWC device was modeled, where the water column variation into the chamber in accordance with the incident wave length was analyzed

This work presents, firstly, a numerical simulation of the problem proposed by Conde and Gato (2008), in order to know and validate this methodology. However, only qualitative analyses could be realized because we don't have the exact dimensions of the OWC device. The VOF methodology used by Liu *et al.* (2008b) was also validated.

After that, a numerical modeling for the air behavior in an OWC device was developed. The computational domain is composed of water and air into a wave tank, and the Volume of Fluid (VOF) method is applied. To validate this approach, an OWC chamber was studied (2D model), and the results compared with the ones presented by Conde and Gato (2008). The ability to obtain an accurate behavior of the air flow in the OWC device is an important characteristic for the correct definition of the turbine type that will be employed in the energy conversion.

2. MODELING THE PROBLEM

In order to present a better modeling problem, it is showed some conceptions that are very important to understand the paper.

2.1. OWC system

The OWC device comprises a partly submerged concrete or steel structure, open below the water surface, inside which air is trapped above the water free surface. The oscillating motion of the internal free surface produced by the incident waves makes the air to flow through a turbine that drives an electric generator. The Wells axial-flow turbine has the advantage of not requiring rectifying valves. It has been used in almost all prototypes (Pontes, 2003). Figure 1 shows a schematic representation of the OWC operation.

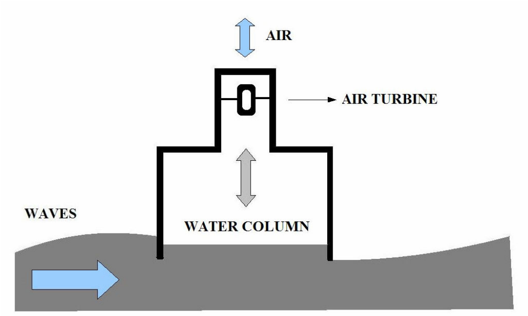

Figure 1. Oscillating water column system.

2.2. Wave maker

The tank has a beater which produces waves working similar to a piston. At the end of the tank, there is an angle to absorb the produced waves, which does the function of a beach. To control the waves, it is necessary to know the period and the height of the generated wave. So, with the knowledge of these two variables, and using a transference function between wave and beater, it is possible to generate the desired wave, as told by Dean and Dalrymple (1991):

$$\frac{H}{S} = \frac{2\left(\cosh\left(2kh\right)-1\right)}{\sinh\left(2kh\right)+2kh} \tag{1}$$

where: H is the height of wave*,* S is the displacement of piston, h the depth and k the wave number.

This way the beater displacement needed to generate the wave with its requested features is defined by (Liu et al., 2008b):

$$x\left(t\right) = \frac{S_0}{2}\left(1 - e^{-\frac{5t}{2T}}\right)\sin\left(\omega\, t\right) \tag{2}$$

where: S_0 is the maximum displacement of the wave maker, T is the period of the incident wave, and $\omega = 2\pi/t$.

The velocity of the mobile surface (the beater) can be calculated by deriving Eq. (2) and given by:

$$v(t)=\left(\frac{S_0}{2}\right)\left[\left(1-e^{-\frac{5t}{2T}}\right)\omega\cos(\omega t)+\frac{5}{2T}e^{-\frac{5t}{2T}}\sin(\omega t)\right] \tag{3}$$

2.3. The chamber geometry

The geometry of the OWC chamber is presented in Fig. 2.

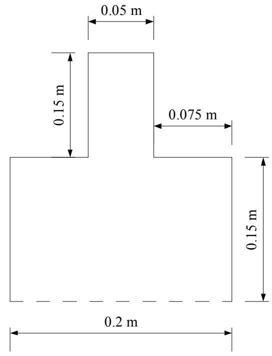

Figure 2. Oscillating water column chamber.

It should be noticed that the dimensions are in laboratory scale. Figure 3 shows the oscillating water column in the wave tank. This geometry becomes possible to consider the complete behavior of the water and the air at the entrance of the chamber.

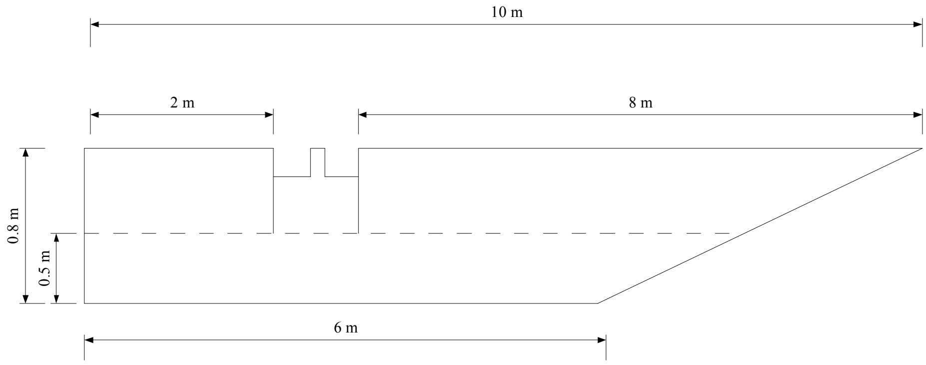

Figure 3. Wave tank

This tank is filled up with 0.5 m height of water. Above this level, it is considered air.

3. NUMERICAL MODEL

The 2D proposed numerical model, used in the numerical simulations, consists basically in the solution of the mass conservation equation, given by

$$\frac{\partial\rho}{\partial t}+\nabla\left(\rho\vec{v}\right)=0 \tag{4}$$

and in solution of the Navier-Stokes equation, given by Eq. (5):

$$\frac{\partial}{\partial t}\left(\rho\vec{v}\right)+\nabla\left(\rho\vec{v}\vec{v}\right)=-\nabla p+\nabla\left(\bar{\bar{\tau}}\right)+\rho\vec{g}+\vec{F} \tag{5}$$

where: p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor, $\rho\vec{g}$ is the gravitational force and $\vec{F}$ are the external forces. The stress tensor is given by Eq. (6):

$$\overline{\overline{\tau}} = \mu\left[\left(\nabla \vec{v} + \nabla \vec{v}^{T}\right) - \frac{2}{3}\nabla \vec{v}\, I\right] \tag{6}$$

where: μ is the viscosity and I is the unit tensor. The second term on the right side is the effect of the volume expanding.

In the present solution the VOF model was used to solve the two-phase problem (water and air). The VOF formulation relies on the fact that two or more fluids (or phases) are not interpenetrating. For each additional phase that you add to the model, a variable is introduced: the volume fraction of the phase in the computational cell. In each control volume, the volume fractions of all phases sum to unity. The fields for all variables and properties are shared by the phases and represent volume-averaged values, as long as the volume fraction of each of the phases is known at each location. Thus the variables and properties in any given cell are either purely representative of one of the phases, or representative of a mixture of the phases, depending on the volume fraction values. In other words, if the fluid's volume fraction in the cell is denoted as α_q, then the following three conditions are possible:

- $\alpha_q = 0$: the cell is empty (of the qth fluid);
- $\alpha_q = 1$: the cell is full (of the qth fluid);
- $0 < \alpha_q < 1$: the cell contains the interface between the qth fluid and one or more other fluids.

Based on the local value of α_q, the appropriate properties and variables will be assigned to each control volume within the domain.

The tracking of the interface (s) between the phases is a accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases. For the qth phase, this equation has the following form:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}\left(\alpha_q \rho_q\right) + \nabla\left(\alpha_q \rho_q \vec{v}_q\right) = S_{\alpha_q} + \sum_{p=1}^{n}\left(\dot{m}_{pq} - \dot{m}_{qp}\right)\right] \tag{7}$$

where: $\dot{m}_{qp}$ is the mass transfer from phase q to phase p and $\dot{m}_{pq}$ is the mass transfer from phase p to phase q. By default the source term on the right-hand side of Eq. (7), S_{α_q}, is zero, but you can specify a constant or user-defined mass source for each phase. The volume fraction equation will not be solved for the primary phase because the primary-phase volume fraction will be computed based on the following constraint:

$$\sum_{q=1}^{n} \alpha_q = 1 \tag{8}$$

The properties appearing in the transport equations are determined by the presence of the component phases in each control volume. In a two-phase system, for example, if the phases are represented by the subscripts 1 and 2, and if the volume fraction of the second of these is being tracked, the density in each cell is given by Eq. (9):

$$\rho = \alpha_2 \rho_2 + \left(1 - \alpha_2\right)\rho_1 \tag{9}$$

In general, for an n-phase system, the volume-fraction-averaged density takes the following form:

$$\rho = \sum \alpha_q \rho_q \tag{10}$$

All other properties (e.g., viscosity) are computed in this manner.

A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum equation which is given by Eq.(5) is dependent on the volume fractions of all phases through the properties ρ and μ.

One limitation of the shared-fields approximation is that in cases where large velocity differences exist between the phases, the accuracy of the velocities computed near the interface can be adversely affected (FLUENT 6.2 users' guide).

To solve this model it is used the Fluent® package. Fluent® is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. Fluent® is written in the C computer language and makes full use of the flexibility and power offered by the language (FLUENT 6.2 users' guide). The discretization of equations is obtained by the Finite Volume Method (FVM). For geometry modeling and mesh generation, it was used the software Gambit®. Mesh independence was achieved using 37,500 quadrilateral elements for the chamber geometry and 83,902 quadrilateral and triangular elements for the tank geometry. In both chamber and tank geometries a time step equal to 0.001 s was used.

4. NUMERICAL SIMULATION

From this moment the methodology proposed by Conde and Gato (2008) will be denominated as AIR methodology, and the procedure used by Liu *et al.* (2008b) and developed in this work will be named VOF methodology.

4.1. AIR methodology validation

Here, the 3D problem proposed by Conde and Gato (2008) was reproduced. The results obtained were presented in Fig. 4, where four points of the OWC interior are analyzed. These results are in agreement with those obtained by Conde and Gato (2008), demonstrating the correct utilization of this methodology.

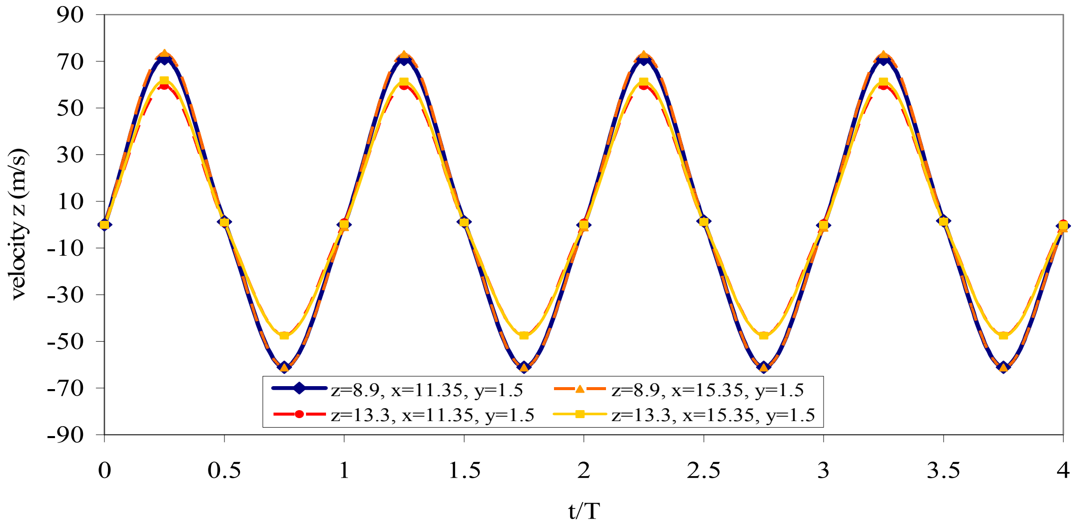

Figure 4. Variation of the velocity in the 3D case.

It is important to emphasize that only qualitative analyses could be realized because we do not have the exact dimensions of the OWC geometry used by Conde and Gato (2008).

4.2. VOF methodology validation

Initially, the generation of the numerical wave, using the VOF method, was validated. For this, the water free surface elevation in a particular region of the wave tank was compared with the analytical solution, which is (Dean and Dalrymple, 1991):

$$\eta(x,t) = \frac{H}{2}\cos(kx - \omega t) \tag{11}$$

where: H is the wave height, x is the position, t is the time variation, k and ω are, respectively, the wave number and the wave frequency, given by:

$$k = \frac{2\pi}{L} \tag{12}$$

$$\omega = \frac{2\pi}{T} \tag{13}$$

The wave tank length is 200 m, the water depth is 16 *m*, the wave amplitude is 0.5 *m* and the wave period is 6 *s* (Liu *et al.*, 2008b). In Fig. 5 the present results are compared with those obtained by Eq. (12), for the position $x = 20\,m$. An error of 0.5% was found, showing the capability of the present numerical model.

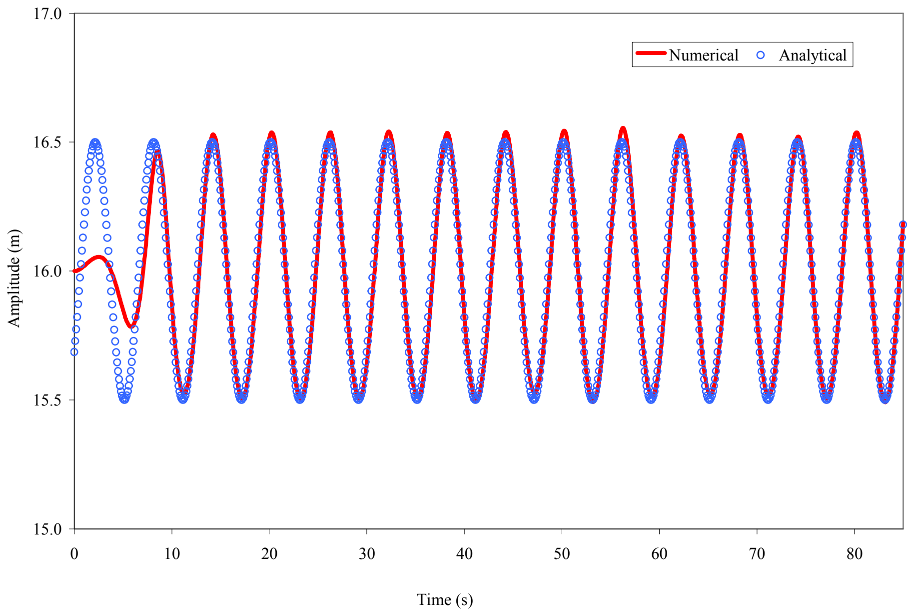

Figure 5. Time series of elevation at the position of $x = 20\,m$.

After that, another validation of the VOF methodology was carried out, in which the water column variation into the OWC chamber, in accordance with the incident wave length, was numerically simulated and the results were compared with those obtained by Liu *et al.* (2008b). The dimensionless a/a_0 against the dimensionless λ/L_f is showed in Fig. 6, where a/a_0 is the relative wave amplitude variation (being a the wave amplitude into the OWC chamber and a_0 is the incident wave amplitude) and λ/L_f is the wave length ratio (being λ the incident wave length and L_f is the chamber width). For this simulation was adopted $a_0 = 0.5\,m$ and $L_f = 6\,m$. It can be seen that the curve generated in this work follows the same trend obtained by the reference.

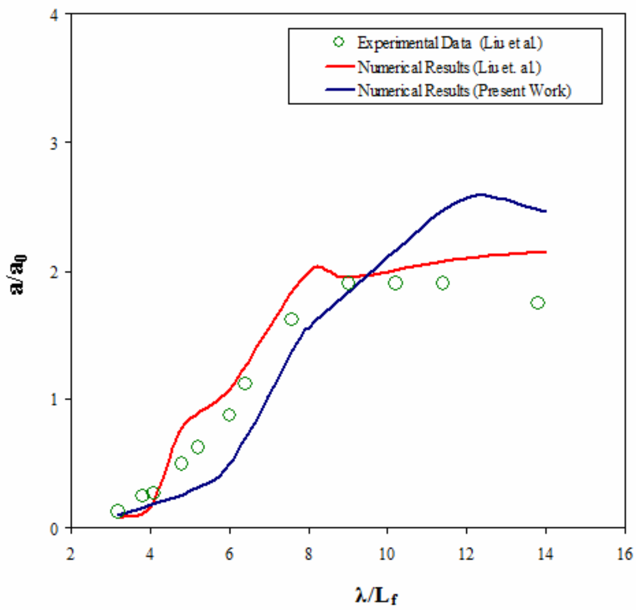

Figure 6. Relative wave amplitude distribution with the wave length ratio variation.

Finally, in Fig. 7, the comparison between analytical solution and numerical solution, in the position $x = 2\,m$, for the wave that will be used in the case study is showed. This numerical wave has a height of 0.14 *m* and was generated in the wave tank previously described (see Fig. 3).

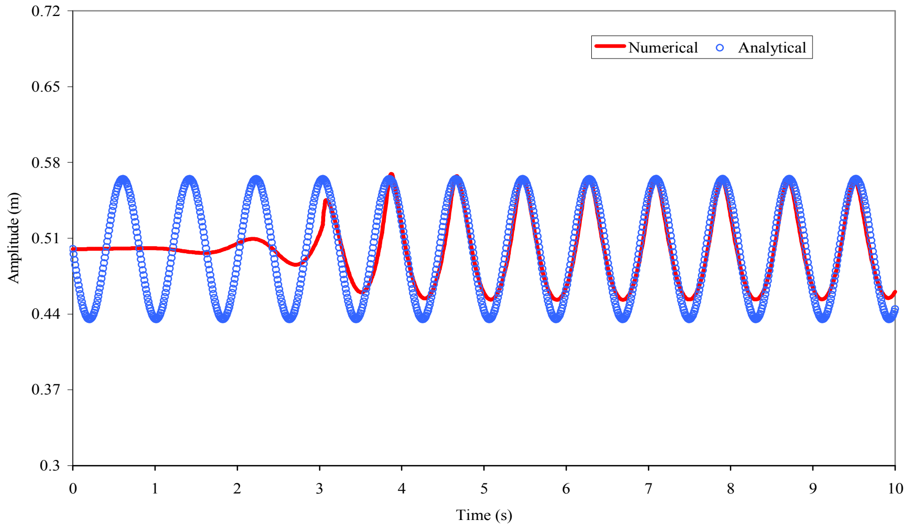

Figure 7. Time series of elevation at the position of $x = 2\,m$.

It is important to emphasize that an error of 6% was encountered, showing a good agreement between the numerical and the analytical solution.

4.3. Case study

An oscillating water column with one vertical tower was studied (Fig. 2). A 2D numerical simulation in FLUENT® package was performed, employing the AIR and the VOF methodologies. The principal difference between these methodologies is how the air behavior is considered. In the AIR methodology, the vertical component of velocity (boundary condition: velocity inlet) of the air in the OWC chamber is defined by a User Defined Function (UDF), given by:

$$v(t) = \frac{0.14\,\pi}{T}\cos\left(\frac{2\,\pi\,t}{T}\right) \tag{14}$$

where: t is the time and T is the wave period. The atmospheric pressure is the boundary condition in the outlet region.

In the VOF methodology a more realistic behavior of the water-air interaction in the OWC device is obtained. For this, it is necessary to consider a computational domain composed by an OWC chamber inside of a wave tank (Fig. 3). The waves are generated using piston type wave-maker, in accordance with Eq. (3). A beach is introduced in the end of the tank to reduce the effect of reflected wave. Again, the outlet region is defined as atmospheric pressure. The configurations adopted for the problem solution in FLUENT® software are shown in Tab. 1.

Table 1. FLUENT® configurations.

Equation	Solution method
Pressure-velocity coupling	Piso
Pressure	Presto
Momentum	First order upwind
Volume fraction	Geo-reconstruct
Turbulence	$k-\varepsilon$

More details about the solution methods mentioned in Tab. 1 are found in FLUENT 6.2 user's guide (2005) and Maliska (2004).

The equation that accounts for the position of the wave in both methods is given by Eq. (15):

$$s(t) = 0.07\sin\left(\frac{2\pi t}{T}\right) \tag{15}$$

where: t is the time [s] and T is the wave period [s^{-1}]. The employed wave's characteristics for comparing the two methodologies are: wave period of 0.8 s and wave amplitude of 0.07 m.

In Fig. 8 the results obtained for the vertical velocity in two interior points of the OWC chamber – (0.1, 0.1) and (0.1, 0.2) – generated with the AIR methodology and the VOF methodology are presented.

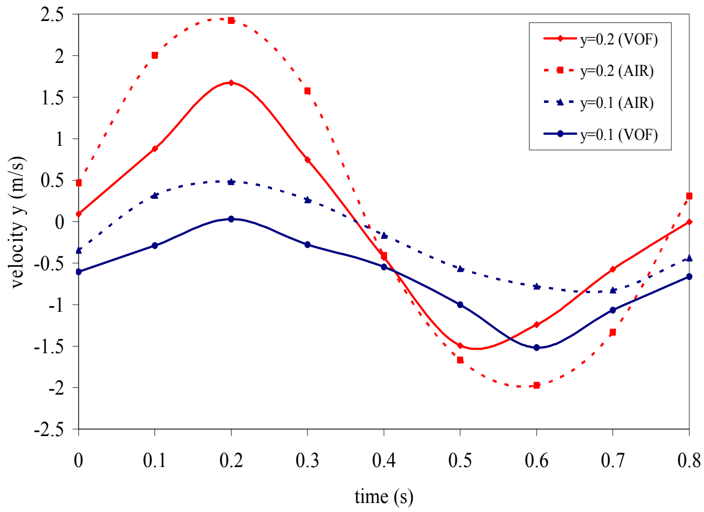

Figure 8. Vertical velocity comparison in OWC chamber.

Analyzing Fig. 8, differences between the results generated by the two methodologies can be observed. This already expected behavior is due to the fact that the AIR methodology is a simplification, considering a constant value for the vertical velocity inlet in the OWC chamber, while the VOF methodology represents the real incidence of the wave into the OWC chamber. In Fig. 9 and Fig 10 are presented the velocity fields into the OWC chamber for the AIR methodology and VOF methodology, respectively.

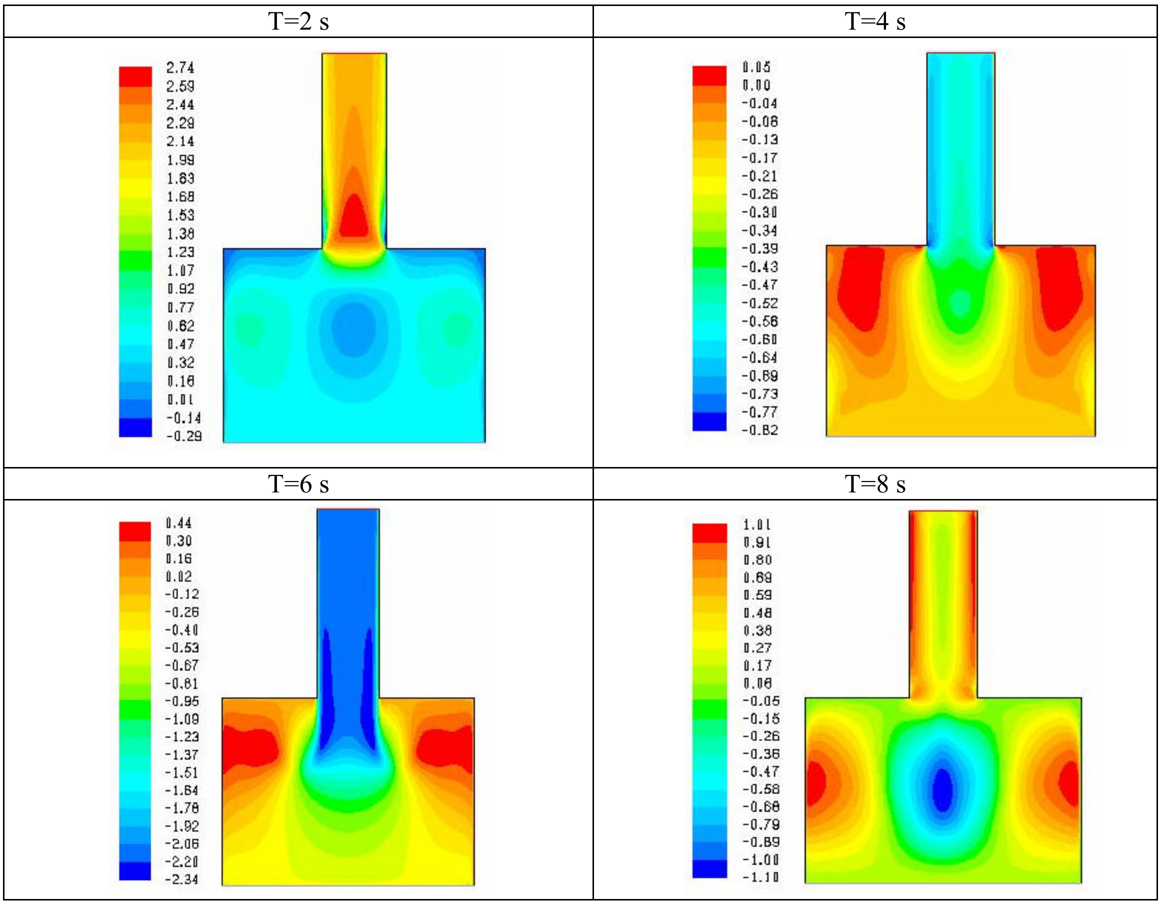

Figure 9. Velocity field of the AIR methodology (m/s)

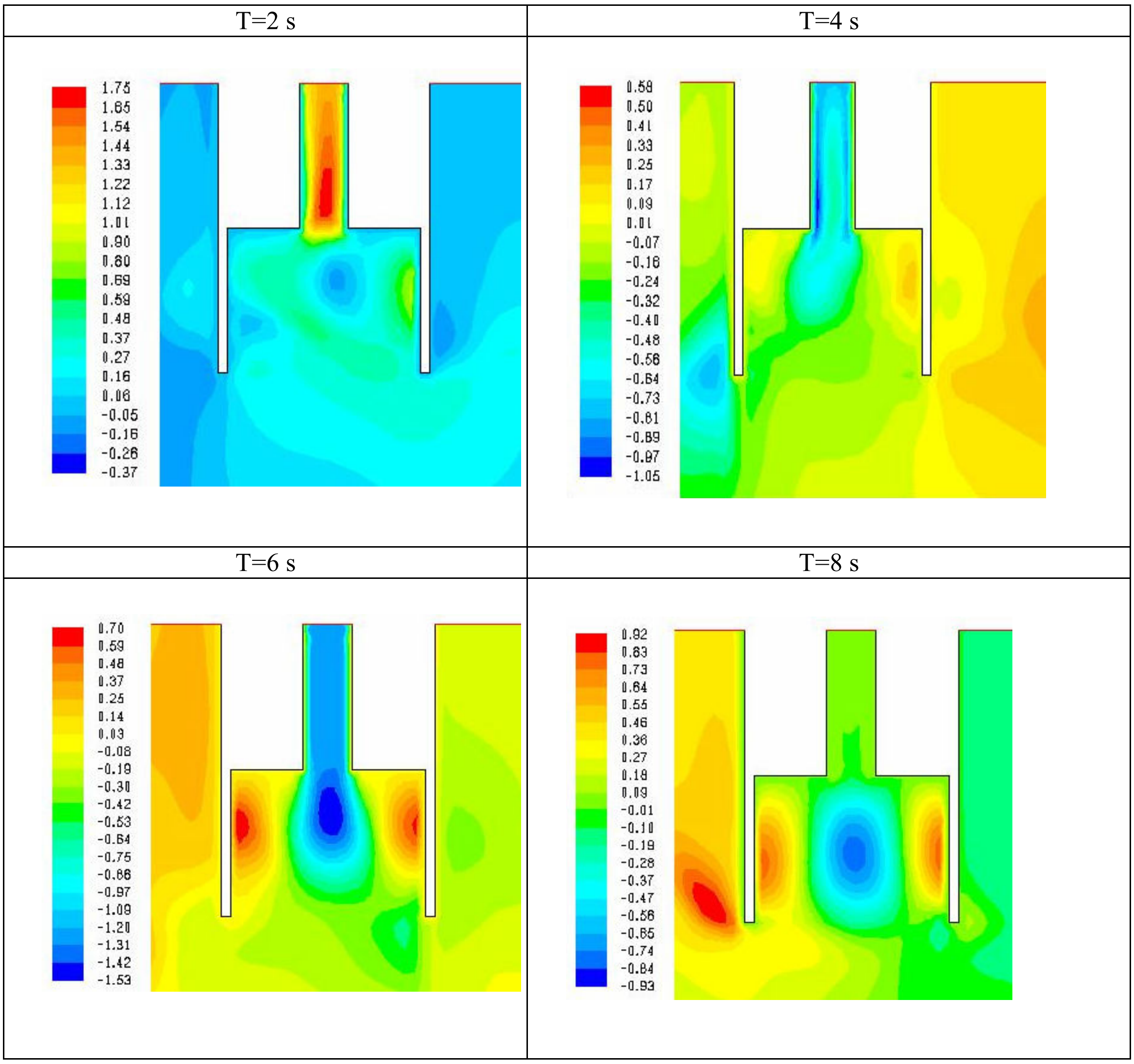

Figure 10. Velocity field of the VOF methodology (m/s)

The AIR methodology (Fig. 9) presents a symmetrical velocity distribution, on the vertical axis, of the air into the chamber, because the entire inlet has the same velocity defined by a cosine function.

In the VOF methodology (Fig. 10), the wave passes under the chamber and the water level presents a non-uniform elevation. Therefore, the air velocity distribution is not symmetric. This methodology allows the simulation of different chambers geometries for several wave lengths, therefore one can optimize the air velocity in the turbine region.

6. CONCLUSION

The accurate behavior of the air flow in the OWC device is an important characteristic for the correct definition of the turbine that will be employed in the energy conversion.

This work proposes a methodology for the numerical simulation of the air behavior in an OWC device where the VOF method is used to represent the water-air interaction into the chamber.

A comparison with the methodology proposed by Conde and Gato (2008) – which considers only the air entrance in the OWC device – was realized demonstrating the validity of the present work.

The principal advantage of this methodology is the way that the air entrance occurs into the OWC chamber, allowing a more realistic air flow behavior. However, this approach uses a more complex computational domain and increases the simulation time. This methodology is also important to optimize the chamber geometry and adapt it to the incident wave.

7. ACKNOWLEDGEMENTS

The authors wish to thank CAPES for the financial support.

8. REFERENCES

Clément, A. et. al, 2002, "Wave energy in Europe: current status and perspectives", Renewable and Sustainable Energy Reviews, Vol. 6, pp. 405–431.

Conde, J.M.P., Gato, L.M.C.,2008, "Numerical study of the air-flow in an oscillating water column wave energy converter". Renewable Energy, Vol. 33, pp. 2637-2644.

Cruz, J.M.B.P., Sarmento, A.J.N.A.,2004 "Energia das Ondas: Introdução aos Aspectos Tecnológicos, Económicos e Ambientais", Ed. Instituto do Ambiente, Amadora, Portugal, 61 p.

Dean, R.G., Dalrymple, R.A., 1991, "Water wave mechanics for engineers end scientists", Vol.2, World scientific, Singapore, 353 P.

Fluent Inc. GAMBIT 2.2, 2004, modelling guide.

Fluent Inc. GAMBIT 2.2, 2004 users' guide.

Fluent Inc. FLUENT 6.2, 2005, users' guide.

Liu, et. al, 2008, "Numerical Prediction for Overtopping Performance of OWEC", Journal of the Korean Society for Marine Environmental Engineering, Vol. 11, No.1, pp. 35-41.

Liu, et. al, 2008, "Aplication of Numerical Wave Tank to OWC air chamber for wave energy conversion ", In: Procedings of the Eighteenth (2008) International Offshore and Polar Engineering Conference, Vancouver, Canada.

Maliska, Clóvis R., 2004, "Transferência de Calor e Mecânica dos Fluidos Computacional", 2a ed., LTC, Rio de Janeiro, 472 p.

Marjani, A.E., Ruiz, F.C., Rodriguez, M. A.; Santos, M. T. P,2008, "Numerical modelling in wave energy conversion systems", Energy, Vol. 33, pp. 1246-1256.

Pontes, T. et. al, 2003, "Generic technologies", Results from the work of the European Thematic Network on Wave Energy. WaveNet, European community, 484 p.

9. RESPONSIBILITY NOTICE

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