NUMERICAL SIMULATION OF SEA WAVE ENERGY CONVERTERS USING THE OPENFOAM SOFTWARE

Jeferson Avila Souza, jasouza@furg.br
Liércio André Isoldi, liercioisoldi@gmail.com
Elizaldo Domingues dos Santos, elizaldo@ibest.com.br
Universidade Federal do Rio Grande – FURG, Escola de Engenharia, Av. Itália, Km 8, S/N, CEP 96203-900, Rio Grande, RS, Brasil

Mateus das Neves Gomes, mateus.gomes@ifpr.edu.br
Instituto Federal do Paraná (IFPR) - Rua Antônio Carlos Rodrigues, 453. Paranaguá/PR, Brasil.

Luiz Alberto Oliveira Rocha, laorocha@gmail.com
Universidade Federal do Rio Grande do Sul, Departmento de Engenharia Mecânica, Porto Alegre, Rio Grande do Sul, Brasil.

Abstract. *In this work OpenFOAM software is used to simulate the generation of regular gravity waves inside a rectangular tank. Interaction between the generated waves and an oscillating water column (OWC) device, used to convert wave energy into electrical energy, is also simulated. The main goal of this work is to develop a numerical strategy to simulate OWC converters using OpenFOAM. Obtained results for the regular wave generation were compared with analytical solution. Maximum errors of 10% were observed between numerical and analytical results for the wave crest elevation. For the OWC device, it was investigated the influence of the lip, distance between the water mean level and the depth of submergence of the OWC, on the mass flow passing through the device's chimney. Results showed that for the wave climate tested, an increase in the lip length results into a decrease in the mass flow rate.*

Keywords: *wave energy, OpenFOAM, Numerical simulation, VOF*

1. INTRODUCTION

Energy generation is a key issue for technological development of a country. It is needed in all segments of the industry and must be adequately supplied. Currently, most of the energy generated in the world is provided by non renewable sources like coal, petroleum and nuclear fuel. These sources can be used to continuously produce large amounts of energy since they are not subject to climatic factors, however they are not renewable and highly pollutant. Alternatively, hydroelectric sources are non pollutant and renewable and have been successfully used worldwide for almost a century, however this is a limited resource and not available for all countries. Some of other non pollutant renewable options are the wind, solar and tide (wave) energy. Wind and solar energy are well develop technologies while wave energy is still under development. There are a number of wave energy converters prototypes and a few installed text facilities, however there is no device ready for commercial utilization.

Because of its long coast and the fact that most of the population lives close to the sea, wave energy may be of great interest to Brazil. In this context, it is necessary to develop new technologies adequately adapted to the Brazilian sea conditions.

There are many types of energy converter devices which can be classified according to their operating principal as: (i) Oscillating Water Column (OWC), (ii) Oscillating Bodies (point absorbers or surging devices) and (iii) Overtopping devices. These devices are usually also classified according to their positioning on the coast as: (i) onshore, (ii) near shore (devices placed at depths lower than 20 m) and (iii) offshore (devices placed at depths higher than 20 m) (Cruz and Sarmento, 2004). More information about wave converters technology and current stage of development can be verified in the works of Clement at al. (2002), Zhang at al. (2009), Falcao (2010) and Bahaj (2011).

Design and construction of such devices (prototypes) is usually a complex and highly expensive operation. In this context, numerical simulation appears as an attractive tool for the early stages of development and/or optimization of geometry and operating condition of these equipment. Overtopping and oscillating water column are two types of wave energy converters that have been subject to numerical investigations by Brazilian researchers (Gomes et al., 2009; Iahnke at al., 2009, Machado et al., 2011). In all of these works, the VOF (Volume of Fluid) method have been successfully used to model sea wave movement and its iteration with such devices. All simulations have been performed with FLUENT software which is a general CFD (Computational Fluid Dynamics) code provided by ANSYS (http://www.ansys.com/). Alternatively to FLUENT, OpenFOAM is an open source CFD code that is also capable of simulate this kind of problem. The main advantage of OpenFOAM, if compared with FLUENT which is a payed software, relies in that fact that it is a free software licensed under the GNU General Public Licence (http://www.gnu.org/licenses/gpl.html).

The use of OpenFOAM has constantly increased in the last few years. Recently a new version (2.0) has been released with a number of improvements. In the particular case of modeling problems with the VOF method, there are a number of works that have successfully used OpenFOAM (Habla et al., 2011, Favero et al., 2010).

In the present work, a 2D model developed in OpenFOAM software is used to simulate numerically gravity waves generated in a rectangular tank. The solution is first validated by direct comparison between numerical and analytical solution for regular waves and in a flowing step used to model the interaction these waves with an OWC device.

2. MATHEMATICAL FORMULATION

In present solution, the VOF method (Hirt and Nichols, 1981) is used. The VOF is a multiphase model used to solve fluid flow problems with two or more inviscid fluids. In this formulation, all phases are well defined and the volume occupied by one phase can not be occupied by other phases. In the VOF method, a volume fraction for each phase f_i is defined such as:

i) if $f_i \neq 0$ the cell is empty with fluid of phase *i*;
ii) if $f_i = 0$ the cell is full with fluid of phase *i*;
iii) if $0 < f_i < 1$ the cell contains the interface between phase *i* and one or more other fluids.

For the particular case of modeling water and air, only two phases are considered in the formulation.

A single set of momentum and continuity equations is applied to both fluids, and the volume fraction of each fluid in every computational cell (control volume) is tracked throughout the domain by the addition of a transport equation for the volume fraction *f*. The model is composed by the continuity, volume fraction and momentum equations as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V} \right) = 0 \tag{1}$$

$$\frac{\partial \left(\rho f \right)}{\partial t} + \nabla \cdot \left(\rho f \vec{V} \right) = 0 \tag{2}$$

$$\frac{\partial \left(\rho \vec{V} \right)}{\partial t} + \nabla \cdot \left(\rho \vec{V} \vec{V} \right) = -\nabla p + \nabla \cdot \left(\mu \bar{\bar{\tau}} \right) + \rho \vec{g} \tag{3}$$

where f is the volume fraction of resin, ρ is the density [kg/m^3], μ the absolute viscosity [Pa s], t the time [s], $\vec{V}$ the velocity vector [m/s], $\vec{g}$ the gravity vector [m/s^2], $\bar{\bar{\tau}}$ the stress tensor [Pa] and p the pressure [Pa].

As a single set of momentum and continuity equations is used for both phases, average properties ρ and μ need to be defined. These properties can be approximated as (Srinivasan et al., 2011)

$$\rho = f \rho_{water} + \left(1 - f \right) \rho_{air} \tag{4}$$

$$\mu = f \mu_{water} + \left(1 - f \right) \mu_{air} \tag{5}$$

A schematic representation of the computational domain with the used boundary conditions is shown in Fig. 1.

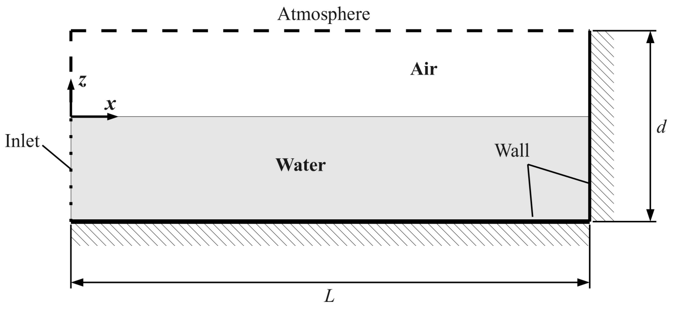

Fig. 1 - Computational domain for the wave tank simulation

In Fig. 1, a rectangular tank with dimension $L \times d$ is initially half-filled with water. The inlet section is defined at the lower part of the left wall (dotted line) where a transient prescribed velocity boundary condition is used to induce the wave generation (Eqs. (7) and (8)). The other boundary conditions are: prescribed pressure equal to zero (gauge) at the sections represented with a dashed line in Fig. 1, no-slip condition at the bottom and right walls and for the OWC problem (Fig. 4), no-slip condition at the chamber and chimney walls.

The second order Stokes wave theory, presented by Dean and Dalrymple (1991), is used to evaluate the velocity profile at the inlet section. A schematic representation of the wave with its characteristic parameters is shown in Fig. 2.

In this theory, the free-surface of water (η) is expressed as

$$\eta = A\cos(kx-\omega t) \tag{6}$$

where A is the wave amplitude [m], $k = 2\pi/l$ is the wave number [m^{-1}], $\omega = 2\pi/T$ is the angular frequency [s^{-1}], T the wave period [s], l the wave length [m] and t the time [s].

The velocity in the x and z directions, respectively, are expressed as

$$u=\frac{Agk}{\omega}\frac{\cosh(kz+kh)}{\cosh(kh)}\cos(kx-\omega t)+\frac{3}{4}A^2\omega k\frac{\cosh[2k(h+z)]}{\sinh^4(kh)}\cos[2(kx-\omega t)] \tag{7}$$

$$w=\frac{Agk}{\omega}\frac{senh(kz+kh)}{\cosh(kh)}sen(kx-\omega t)+\frac{3}{4}A^2\omega k\frac{\sinh[2k(h+z)]}{\sinh^4(kh)}\sin[2(kx-\omega t)] \tag{8}$$

where g = 9.81 m/s² is the gravity acceleration.

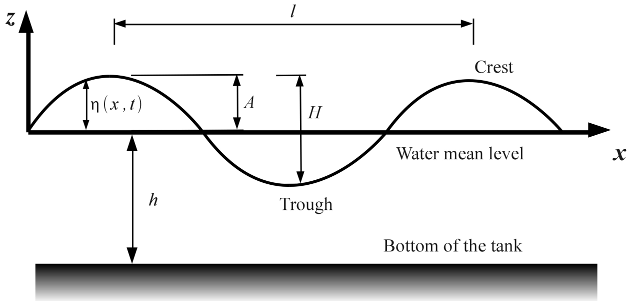

Fig. 2 - Wave characteristic parameters

3. WAVE TANK SIMULATION

The first case simulated consists in evaluate the water free-surface elevation as function of time in a particular region of the tank. The numerical solution obtained with OpenFOAM is compared with the analytical solution given by Eq. (6). The tank geometry and wave parameters used in the current simulation are presented in Tab. 1.

Table 1. Geometry and wave parameters for the wave tank simulation

Variable	***L* [m]**	***d* [m]**	***A* [m]**	***l* [m]**	***h* [m]**	***T* [s]**
Value	200	32	0.5	53.6	16	6

The free-surface elevation obtained with the numerical and analytical solutions at x = 20 m is shown in Fig. 3. Initially the wave obtained numerically is not fully developed and the magnitude of the free-surface oscillation increases with the time advance. At t ~ 10 s the wave reaches its periodic states and starts to oscillate with constant amplitude. Since the analytical solution does not take into account the inertial behavior at the beginning of the flow, large deviations are observed in this period.

Numerical and analytical solutions comparison is considered only for t > 10 s. The difference between these solutions is define as

$$\beta=\frac{\eta_a-\eta_n}{\eta_a}\times 100 \tag{9}$$

where the subscripts a and n indicate analytical and numerical solutions, respectively.

A maximum value of β ~ 10% was observed for the crest amplitude of the wave. In spite of this, the numerical results show a satisfactory agreement in comparison with the analytical predictions.

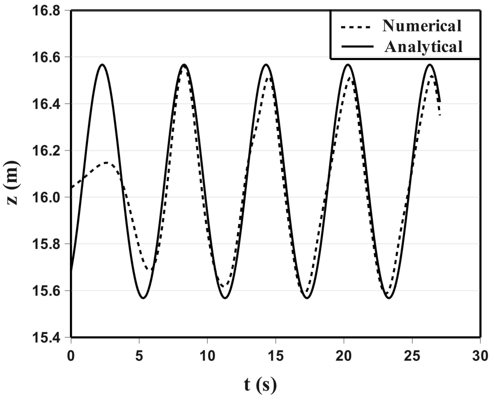

Fig. 3 - Numerical and analytical solution comparison (x = 20 m)

4. *lip* INFLUENCE TO THE OWC DEVICE PERFORMANCE

The problem geometry is shown in Fig. 4. Current solution is similar to the wave tank simulation presented in section 3 with the difference that an OWC device is placed inside the tank. In this assembly, the wave (water) movement forces the air contained within the chamber to flow into and out through the chimney. Power is extracted from the equipment by adding a Wells turbine (usually) to the device's chimney, however in the present simulation, the turbine influence is not accounted in the model.

Geometry and wave parameters used in the OWC simulation are show in Tab. 2.

Table 2. Geometry and wave parameters for the OWC simulation

Variable	***L*** **[m]**	***d*** **[m]**	***A*** **[m]**	***l*** **[m]**	***h*** **[m]**	***T*** **[s]**
Value	6	1	0.07	1.2	0.5	0.8

The *lip* is the distance between the water mean level and the depth of submergence of the OWC. Its height has a direct influence to the water flow inside the chamber and consequently to the amount of energy transferred from the water to the air. Consequently, the amount of power produced by the turbine will also be effect by the *lip.*

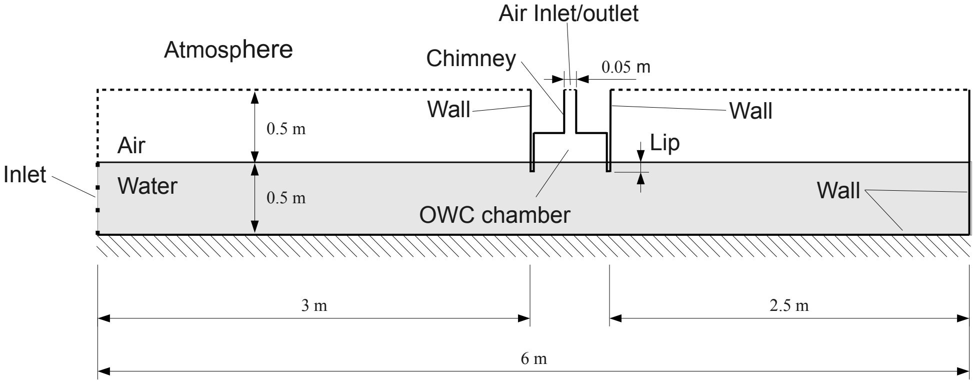

Fig. 4 - Computational domain for the OWC device simulation

The *lip* influence to the OWC performance is investigated by running a set of simulations on which all parameters (geometry, boundary and initial conditions) are kept constant and the *lip* is made to vary from 0.01m to 0.1m. In this work the investigated parameter is the amount of air (mass) that flows through the device's chimney.

Mass flow rate is evaluated at the air inlet/outlet section of the chimney (Fig. 4). The mass flow rate as a function of time for different *lip* lengths is shown in Fig. 5.

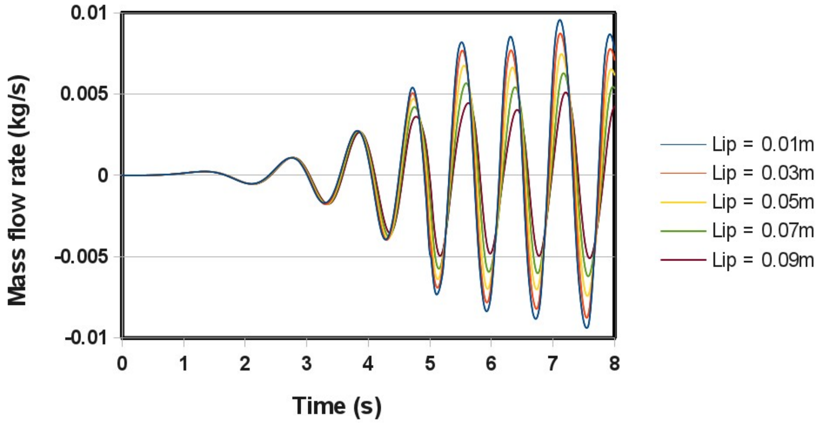

Fig. 5 - *Lip* influence on mass flow rate

In Fig. 5, positive flow rate indicates that the air is leaving the chamber through the chimney while negative flow rate indicates that the air is entering the chamber. In this analysis, the chamber volume above water level is kept constant, however the submerged volume increases with the increasing of the *lip*. For this particular case, as show in Fig. 5, the effect of increasing the *lip* results in decreasing the flow rate through the chimney, what is an undesired behavior.

The total mass (m_{total}) of air passing through the chimney's inlet/outlet section during the 8s simulation is accounted by

$$m_{total} = \sum \rho |\dot{\forall}| \Delta t \tag{10}$$

where ρ is the air density [kg/m³], $\dot{\forall}$ is the volumetric flow rate [m³/s], and Δt the numerical solution time step [s].

The amount of air passing through the chimney during the 8s simulation is show in shown Fig. 1. This figure shows an almost linear (inversely) relationship between the l*ip* and the mass passing through the chimney.

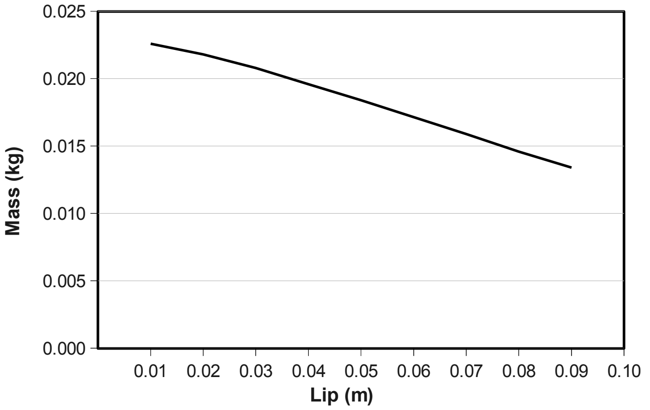

Fig. 6 - Mass of air passing thought the chimney during the 8 s simulation period

5. CONCLUSIONS

OpenFOAM software, which is and open source CFD package, has been used to simulate an OWC device. The wave climate was artificially generated by imposing a prescribed velocity boundary condition to the left wall of the domain. Solution was first validated in a simple wave tank simulation where the analytical water level is compared with numerical results and then used to investigate the influence of the *lip* to the mass flow rate passing through the device's chimney.

It was observed that increasing the *lip*, the mass flow passing through the chimney (entering or leaving the chamber) is reduced. Since the energy that can be extracted from the air is proportional to the volumetric flow rate, results presented in Fig. 6 indicates that the *lip* parameter is an important variable to be controlled.

6. ACKNOWLEDGMENTS

The authors thank the Fundação de Amparo à Pesquisa do Estado do Rio Grande do SUL (FAPERGS) and the Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq) for the financial support.

7. REFERENCES

Bahaj, A.S., 2011. "Generating electricity from the oceans". Renewable & Sustainable Energy Reviews, Vol. 15, No. 7, pp. 3399-3416.

Clement, A., McCullen, P., Falcao, A., Fiorentino, A., Gardner, F., Hammarlund, K., Lemonis, G., Lewis, T., Nielsen, K., Petroncini, S., Pontes, M., Schild, P., Sjostrom, B., Sorensen, H. and Thorpe, T., 2002. "Wave energy in Europe: current status and perspectives". Renewable & Sustainable Energy Reviews, Vol. 6, No. 5, pp. 405-431.

Cruz, J.M.B.P. and Sarmento, A.J.N.A.. 2004. Energia das Ondas: Introdução aos Aspectos Tecnológicos, Econômicos e Ambientais. . Fernandes & Terceiro, Lda., .

Dean, R.G. and Dalrymple, R.A.. 1991. Water Wave Mechanics for Engineers & Scientists. . World Scientific Pub Co Inc, .

Falcao, A.F.D.O., 2010. "Wave energy utilization: A review of the technologies". Renewable & Sustainable Energy Reviews, Vol. 14, No. 3, pp. 899-918.

Favero, J.L., Secchi, A.R., Cardozo, N.S.M. and Jasak, H., 2010. "Viscoelastic fluid analysis in internal and in free surface flows using the software OpenFOAM". Computers & Chemical Engineering, Vol. 34, No. 12, SI, pp. 1984-1993.

Gomes, M.N., Isoldi, L.A., Olinto, C.R.O., Rocha, L.A.O. and Souza, J.A.. 2009. "Numerical simulation and length optimization of the water column oscillating device". In Proceedings of the 30th Iberian-Latin-American Congress on Computational Methods in Engineering - CILAMCE. Armação dos Búzios, RJ, Brazil.

Habla, F., Marschall, H., Hinrichsen, O., Dietsche, L., Jasak, H. and Favero, J.L., 2011. "Numerical simulation of viscoelastic two-phase flows using openFOAM (R)". Chemical Engineering Science, Vol. 66, No. 22, pp. 5487-5496.

Hirt, C.W. and Nichols, B.D., 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries". Journal of Computational Physics, Vol. 39, No. 1, pp. 201-225.

Iahnke, S.L.P., Gomes, M.N., Isoldi, L.A. and Rocha, L.A.O.. 2009. "Energy from the Sea: Computational Modeling of an Overtopping Device". In Proceedings of the Third Southern Conference on Computational Modeling (MCSUL). Rio Grande, Brazil.

Machado, B.N., Zanella, M.M., Gomes, M.D.N., Teixeira, P.R.F., Isoldi, L.A., Santos, E.D.D. and Rocha, L.A.O.. 2011. "Constructal Design of an Overtopping Wave Energy Converter". In Proceedings of the Constructal Law Conference. Porto Alegre, Brazil.

Srinivasan, V., Salazar, A.J. and Saito, K., 2011. "Modeling the disintegration of modulated liquid jets using volume-of-fluid (VOF) methodology". Applied Mathematical Modeling, Vol. 35, No. 8, pp. 3710-3730.

Zhang, D., Li, W. and Lin, Y., 2009. "Wave energy in China: Current status and perspectives". Renewable Energy, Vol. 34, No. 10, pp. 2089 - 2092.

8. RESPONSIBILITY NOTICE

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