Ocean Engineering 125 (2016) 147–154

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Study of the turbine power output of an oscillating water column device by using a hydrodynamic – Aerodynamic coupled model

Fernando R. Torres [a], Paulo R.F. Teixeira [a,*], Eric Didier [b,c]

[a] Universidade Federal do Rio Grande, Escola de Engenharia, Av. Itália, km 8, Campus Carreiros, Rio Grande, RS, 96201-900 Brazil
[b] Laboratório Nacional de Engenharia Civil – LNEC, Departamento de Hidráulica e Ambiente, Av. do Brasil, 101, Lisboa, 1700-066 Portugal
[c] Universidade Nova de Lisboa, Faculdade de Ciências e Tecnologia – FCT-UNL, UNIDEMI, Monte de Caparica, 2829-516 Portugal

ARTICLE INFO

Article history:
Received 24 February 2016
Received in revised form
10 August 2016
Accepted 13 August 2016

Keywords:
Finite element method
Wave energy
Oscillating water column
Turbine model
Air pressure control

ABSTRACT

The availability of clean energy of wind waves is enough to meet current demand of the world, attracting researches to find devices with good performance to extract this type of energy. The oscillating water column converter is a type of device that has been extensively studied for this purpose. This paper describes a numerical analysis of an onshore oscillating water column device taking into account hydrodynamic and aerodynamic coupling, performance curves of the turbine and air pressure control by a relief valve. The numerical analyses are carried out by the Fluinco model, based on slightly compressible flow method to solve the Navier–Stokes equations, and employing the two-step semi-implicit Taylor–Galerkin method. The aerodynamic model is based on the first law of thermodynamics applied to the air column inside the chamber and on turbines similar to Pico's. The turbine power output with air pressure control inside the chamber due to a relief valve is compared to the one without control. A significant increase in the power output by using the pressure control in comparison with that without control is observed. The analysis of the turbine power output considers different turbine diameters and ranges of rotational speeds.

1. Introduction

Wave energy along coastal areas has attracted interest of many researches to find the most efficient technology to extract it. However, there has been no consensus about the best device to be used. One of the most experimentally and numerically studied devices is the Oscillating Water Column (OWC), in which several prototypes have been developed in situ (Tofteshallen, Norway (500 kW); Sakata, Japan (60 kW); Pico, Portugal (400 kW); Limpet, Scotland (500 kW); Mutriku, Spain (300 kW)). In this device, the oscillatory movement of the water surface inside a chamber induces inhalation and exhalation of the air through a turbine, which is responsible for converting aerodynamic energy into mechanical one.

To cope with the reversing air flow, self-rectifying turbines are used in OWC devices. Two different self-rectifying turbines have currently been used (Amundarain et al., 2010): the impulse and the Wells turbines. Some recent overviews on self-rectifying turbines were described by Starzmann (2012), Falcão and Gato (2012) and Falcão and Henriques (2016). New concepts of self-rectifying turbine, as the biradial turbine (Falcão et al., 2013) and the twin-rotor radial-inflow turbine (Falcão et al., 2015), have been developed in the last few years. The present study considers only the use of the Wells turbine. The Wells turbine rotor consists of several symmetrical aerofoil blades set around a hub. The symmetric construction of aerodynamic profiles is responsible for operating at the same rotation direction regardless the direction of the flow (inhalation or exhalation). Although the performance could be improved by using turbine rotational speed control and pressure control inside the air chamber, the correct sizing of the turbine is an important step to achieve optimal performance.

There are several studies of numerical analysis of the hydrodynamics of the OWC, including those that use models based on 2D and 3D Navier Stokes equations (Horko, 2007; Liu et al., 2008, 2009a, 2009b; Marjani et al., 2008; Lopéz et al., 2014; Luo et al., 2014; Iturrioz et al., 2015; Teixeira et al., 2013; Kamath et al., 2015). Some of the difficulties refer to taking into account the hydrodynamic and the aerodynamic coupling inside the chamber, besides the turbine performance to determine its power output. In this context, this paper shows a numerical simulation of the hydrodynamic and the aerodynamic fields in an OWC device subjected to regular incident waves by using the Fluinco code, which is based on 2D Navier Stokes equation. The calculation of the turbine power output takes into consideration 3D real turbines,

* Corresponding author.
E-mail address: pauloteixeira@furg.br (P.R.F. Teixeira).

http://dx.doi.org/10.1016/j.oceaneng.2016.08.014
0029-8018/

similar to the Wells one located in the Pico's plant.

The 2D hydrodynamic module of Fluinco is based on the semi-implicit two-step Taylor–Galerkin method (Teixeira and Awruch, 2005) which adopts a triangular linear element. An arbitrary Lagrangian Eulerian (ALE) formulation is used for enabling the solution of problems involving movements of free surface. The spatial velocity mesh distribution is such that distortion of elements is minimized by the use of functions that consider the influence of the velocity of each node belonging to boundary surfaces.

The 3D aerodynamic module is based on the first law of thermodynamics and ideal gas isentropic transformation applied to the air inside the chamber (Teixeira et al., 2013; Josset and Clément, 2007; Sheng et al., 2013). Therefore, the variation of the air volume along the time, promoted by the free surface movement in the OWC chamber, induces the turbine flow and variation of the pressure inside the chamber, characterizing a coupling between hydrodynamic and aerodynamic systems, as observed in the prototype device.

It should be emphasized that the interaction among waves and OWC devices has predominantly a 2D hydrodynamic behavior; thus, consequently, the use of a 2D modeling demands lower computational cost by comparison with a 3D modeling. Besides, the aerodynamic field is fundamentally 3D, due to its dependence to the air volume inside the chamber and the characteristics of the turbine. Studies based on the volume of fluid (VOF) concept need to include the air inside the chamber in their computational domain and, because of this, both hydrodynamic and aerodynamic fields are 3D. Moreover, the characteristic of the Fluinco model allows the analysis of the turbine power output of an OWC device considering a 2D hydrodynamic modeling and, simultaneously, a simpler, but accurate and efficient 3D aerodynamic modeling.

A description of the equations and numerical methods used by the Fluinco numerical code is shown in Section 2. The study case under investigation is described in Section 3. In Section 4, results of waves with three different periods (6 s, 9 s and 12 s) for a turbine with diameter $D=1.5$ m are shown and the use of controls is discussed. Finally, the analysis of the turbine power output is carried out for a wave with $T=9$ s and turbines with different diameters ($D=1.5$ m, 2.0 m and 2.5 m).

2. Fluinco model

2.1. Hydrodynamic module

The Fluinco model is based on the mass conservation equation for slightly compressible fluids, assuming constant entropy, which may be expressed as follows:

$$\frac{\partial\rho}{\partial t} = \frac{1}{c^2}\frac{\partial p}{\partial t} = -\frac{\partial U_i}{\partial x_i} \ (i = 1, 2) \tag{1}$$

where ρ is the specific mass, c is the sound speed, p is the pressure, $U_i=\rho\ v_i$ and v_i are the fluid velocity components. The momentum equations, in ALE description, complete the 2D governing equations of the isothermal fluid flow problem:

$$\frac{\partial U_i}{\partial t} + \frac{\partial f_{ij}}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial p}{\partial x_i} - \rho g_i = w_j\frac{\partial U_i}{\partial x_j} \ (i, j = 1, 2) \tag{2}$$

where w_i are the mesh velocity components, g_i are the gravity acceleration components and $f_{ij}=v_j\ U_i$. τ_{ij} are the components of the deviatory tensor, given by:

$$\tau_{ij} = \mu\left(\frac{\partial v_i}{\partial xj} + \frac{\partial v_j}{\partial xi}\right) \ (i, j = 1, 2) \tag{3}$$

where μ is the viscosity coefficient. Initial and boundary conditions must be added to Eqs. (1) and (2) in order to define the problem uniquely.

Variables U_i are discretized in time domain using a Taylor series expansion in two steps. The classical Galerkin weighted residual method is applied to the space discretization adopting triangular elements. In the variables at $t+\Delta t/2$ instant, a constant shape function is used whereas, in the variables at t and $t+\Delta t$, a linear shape function is employed.

The kinematic free surface boundary condition (KFSBC) is imposed on the free surface (interface between the air and the water), where the atmospheric pressure is considered constant. By using the ALE formulation, the KFSBC can be expressed as (Ramaswamy and Kawahara, 1987):

$$\frac{\partial\eta}{\partial t} + {}^{(S)}v_1\frac{\partial\eta}{\partial x_1} = {}^{(S)}v_2 \tag{4}$$

where η is the free surface elevation, ${}^{(S)}v_i$ ($i=1,2$) are the velocity components on the free surface. The Eulerian formulation is used in the horizontal direction x_1, while the ALE formulation is employed in the x_2 or vertical direction. The time discretization of KFSBC is carried out in the same way as the one for the momentum equations, by applying expansion in Taylor series.

The mesh velocity vertical component w_2 is computed to diminish element distortions, keeping prescribed velocities on moving (free surface) and stationary (bottom) boundary surfaces. The mesh movement algorithm adopted in this paper uses a smoothing procedure for the velocities based on these boundary surfaces. The updating of the mesh velocity at node i of the finite element domain is based on the mesh velocity of the nodes j that belong to the boundary surfaces.

2.2. Aerodynamic module

The aerodynamic and the hydrodynamic phenomena inside the chamber are strongly coupled. The 3D aerodynamic model implemented in the Fluinco code is based on the first law of thermodynamics applied to the air column, considering the open system hypothesis and the ideal gas (Teixeira et al., 2013; Josset and Clément, 2007). Assuming negligible isentropic transformation and kinetic and potential energies, the energy balance equation results in the following relation:

$$\frac{\dot{p}(t)}{p(t)} = \gamma\left[\frac{Q_t(t)}{V(t)}\left(1 - \varepsilon\frac{\rho(t) - \rho_0}{\rho(t)}\right) - \frac{\dot{V}(t)}{V(t)}\right] \tag{5}$$

where $Q_t = \left(p - p_0\right)/k_t$ is air flow through the turbine; p and $\dot{p}$are pressure and its rate, respectively; p_0 is atmospheric pressure; V and $\dot{V}$are air volume inside the chamber and its rate, respectively; k_t is turbine characteristic relation; $\gamma=c_p/c_v$ (equal to 1.4 for the air), where c_p and c_v are specific heats at constant pressure and constant volume, respectively; ρ and ρ_0 are specific mass inside and outside the chamber, respectively; ε is null for negative value of air flow (air exiting the chamber) and one for the opposite case.

The compressibility effect is considered by the following relation of isentropic transformation:

$$p(t)\,\rho(t)^{-\gamma} = p_0\rho_0^{-\gamma} \tag{6}$$

In the Fluinco code, Eq. (5) is discretized in time by using Taylor series up to second order, in which the air pressure is updated in two steps. The air volume inside the chamber and its rate are calculated at each instant, taking into account the values of free surface elevation and mesh velocity vertical component, respectively, of all nodes that belong to the free surface inside the chamber. The updated air pressure, obtained by Eq. (5), is imposed at each time step as a boundary condition on the free surface of