AN OPENFOAM-BASED TWO-PHASE FLOW MODEL FOR SIMULATING THREE-DIMENSIONAL OSCILLATING-WATER-COLUMN DEVICES: MODEL VERIFICATION AND VALIDATION

CONGHAO XU[1], ZHENHUA HUANG[2]
[1] *Dept. of Ocean and Resources Engineering, University of Hawaii at Manoa, conghaox@hawaii.edu*
[2] *Dept. of Ocean and Resources Engineering, University of Hawaii at Manoa, zhenhua@mhawaii.edu*

Keywords: *Wave energy converter, oscillating water column, three-dimensional simulation*

1 Introduction

Wave power is one of the most promising renewable energy sources. Many types of wave energy converters (WECs) have been proposed in the past to extract energy from wave field for electricity generation, and the oscillating water columns (OWC) are one type of the promising WECs that have been widely tested and investigated. A typical OWC consists of an air chamber above the water surface with one turbine mounted to the pneumatic chamber for electricity generation; the bottom of the chamber is open and submerged so that the incident wave can cause the water column inside the chamber to oscillate. The oscillation of the water surface in side the chamber generates a fluctuation of the air pressure inside the chamber, causing the air flow to drive the turbine for electricity generation. One advantage of the OWC-type wave energy converters is its robustness in simplicity of wave energy extraction mechanism.

Many studies on the hydrodynamics and energy extraction of various OWC devices can be found in the literature; the methods used include wave-flume test, frequency-domain analysis based on potential flow theory, numerical simulations based on potential flow theory, computational-fluid-dynamics (CFD) simulations. CFD simulations of OWC devices can provide important information about the complex flow field around an OWC device and the spatial distribution of the water surface inside the OWC chamber, which is otherwise very difficult to obtain in laboratory experiments. Most of existing CFD simulations of OWC devices focused on two-dimensional configurations such as a rectangular shaped OWC [1]. OpenFOAM is an open-source CFD library for solving CFD problems using a volume of fluid method to track the free surface [2]. In this study, we introduce an OpenFOAM-based two-phase flow model developed for a circular OWC wave-energy converter integrated into a large pile. The power-takeoff is modeled using an orifice on the top cover of the OWC chamber. This study focused on the validation and verification of an OpenFOAM-based two-phase flow model using an existing set of wave-flume test results[3].

2 Model description

The relaxation-zone-based wave-generation method [2] is used for wave generation. In this method, the computational domain is divided into three sections: a wave generating relaxation section, a test section and the wave-absorbing relaxation section (numerical beach), see Fig.1 for the numerical setup and the three sections. In this study, the length of the wave generating relaxation section is 4 m long, the test section is 6 m long, and the wave absorbing section is 4 m long. Numerical tests have shown that the reflection coefficient from the wave absorbing relaxation zone is less than 0.05 for the waves examined in this study. The turbulence model used is a $k - \omega$ model. The air-water surface is tracked using a VOF method. The thin wall of the OWC chamber and the high velocity through the orifice requires very fine mesh in the vicinity of the orifice. Fig. 2 shows the quality of the waves generated in the numerical wave tank.

The oscillating-water-column device studied here is an axisymmetric oscillating-water-column (OWC) device supported by a coaxial tube-sector-shaped structure, which integrates an oscillating water column with a circular pile [3], see the right panel of Fig. 4 for a 3D view of the OWC device. The model has an overall dimension of 40 cm in total height. The distance from the lower tip of the OWC chamber to the flume bed is 24.4 cm. The inner diameter of the cylinder is 12.5 cm and the thickness of all walls is 3 mm. On the top of the tube sector an orifice of diameter 1.4 cm is used to simulate a nonlinear power takeoff (PTO) device.

3 Key Results

Representative results are presented in this section to show the agreement between the simulated and measured results. These results include surface displacements at three locations (G_1, G_2, and G_3 as shown in Fig, 1), the relative pressure of the air in the pneumatic chamber, the simulated velocity and vorticity.

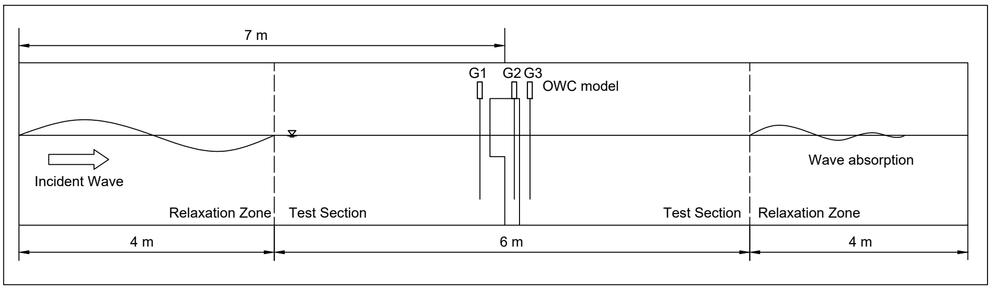

Figure 1: Numerical setup of the numerical wave tank. Not drawn to scale. G_1, G_2 and G_3 are the three locations where waves are measured by wave gauges.

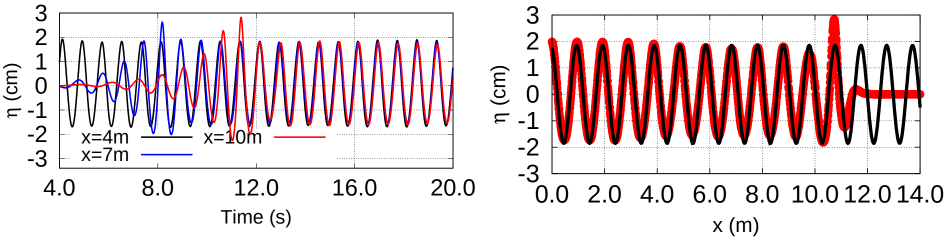

Figure 2: Left: Times series of the waves measured at three locations in the numerical wave tank with x being the distance from the left end of the numerical wave tank. Right: A snapshot of the waves in the numerical wave tank with red line being the simulation result and the black line being the theoretical Airy wave solution.

Comparisons between the time series of the measured and simulated relative pressure and the time series of the measured and simulated surface displacements at three locations for monochromatic waves are shown in the left panel in Fig. 3 for wave period= 0.8 s, wave height= 0.04 m and water depth=0.31 m and in the right panel of Fig. 3 for wave period = 1.3 s, wave height = 0.04 m , and water depth= 0.29 m. The simulated and measured surface displacements agree very well at all three locations. The relative pressure of the air in the pneumatic chamber, which is basically the pressure drop across the PTO, can be modeled by

$$p(t) = \frac{1}{2} c_f \rho_a |u)t)|u(t) + \rho_a L_g \frac{du(t)}{dt} \tag{1}$$

where $u(t)$ is the air velocity, ρ_a the air density, c_f a quadratic loss coefficient, and L_g an empirical length scale related to inertia effect. For sinusoidal waves, the relative pressure in the air chamber is not sinusoidal because of the nonlinear PTO (Eq. 1) used in the study. As shown in Fig. 3, the numerical model can simulate the relative pressure of the air inside the pneumatic chamber very well, implying that the numerical stimulation can simulate the behavior of the nonlinear PTO very well.

The left panel of Fig. 4 shows an example of the simulated velocity of the air and the air-water interface. Even though the wave height is just 4 cm, the maximum velocity of the air through the PTO device can reach as high as 12 m/s. The fine mesh and high velocity speed in the vicinity of the PTO requires very small time step. Therefore, the bottleneck in the 3D simulation of an OWC device is the air flow through the PTO.

The right panel of Fig. 4 shows an example of the simulated vorticity in water. In this example, the Keulegan–Carpenter number, which can be defined by $KC = UT/D$ with D being the diameter of the OWC chamber, T the wave period and U a velocity scale. If we take $U = \omega H/2$, we have $KC = \pi H/D$. For H =0.04 m and T=1 s, KC=1.0, which means the vortex does not shed from the OWC and its support structure. This is confirmed by the computed vorticity distribution shown in right panel of Fig. 4, which shows that the vortex shedding mainly occurs at the lower tip of the OWC skirt and weak vortex shed from the sharp edge of the support structure.

4 Conclusions

An OpenFOAM-based numerical wave tank was used to study a three-dimensional OWC device supported by a C-shaped structure. The numerical results were able to reproduce the measured surface displacements around the OWC device and inside the pneumatic chamber very well. The model can also simulate the relative pressure of the air inside the pneumatic chamber very well, suggesting that numerical model can provide satisfactory modeling of the behavior of the nonlinear PTO used in the physical model tests. For the particular OWC device examined here, the vortex shedding occurs mainly at the low tip of the OWC chamber skirt. Study of how the vortex shedding affect the wave energy extraction efficiency is on the way and will be reported elsewhere.

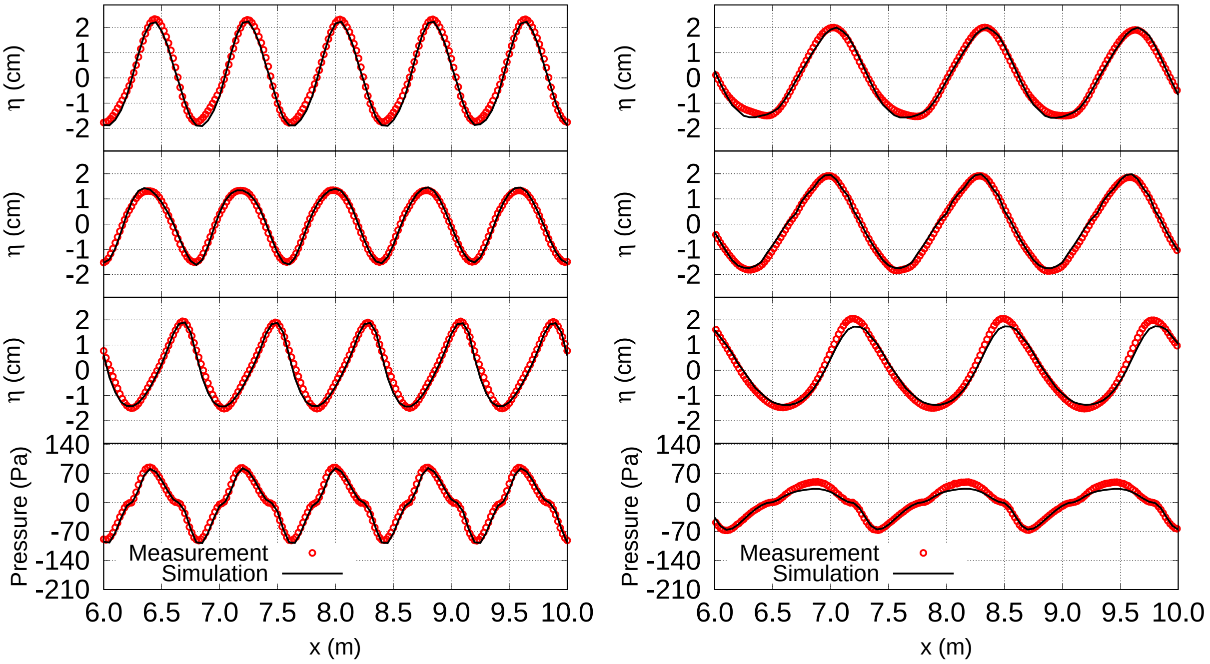

Figure 3: Comparisons between the simulated and measured surface displacements at three locations and the air pressure for T =0.8 s (left) and T=1.3 s (right). From top to bottom, the plots show the measured surface displacements at the locations G_1, G_2 and G_3, and the relative air pressure. The circles are measurement and the solid lines are numerical simulations.

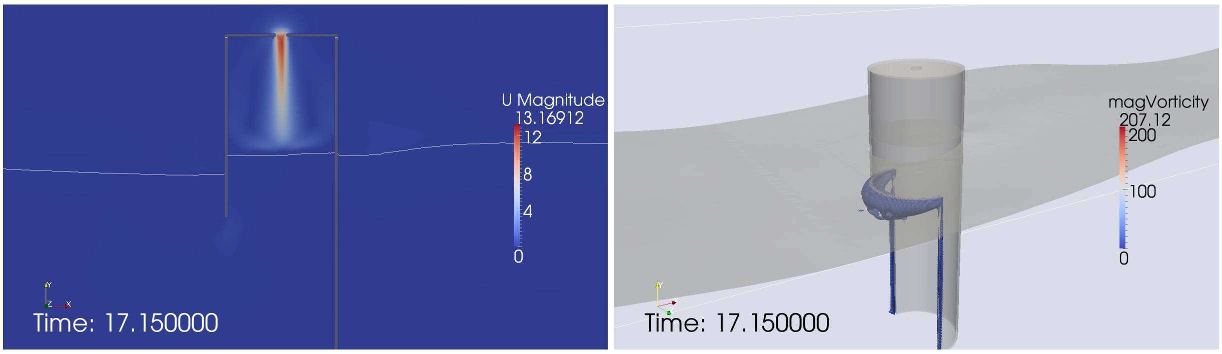

Figure 4: Left: an example of the simulated air velocity and the air-water interface. Right: An example of the simulated vorticity. For both panels. T =1.0s, H =0.04 m, and h =0.31 m.

Acknowledgments

This material is based upon work supported by the US National Science Foundation under Grant No. 1706938. Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation.

References

[1] A. Iturrioz, R. Guanche, J. L. Lara, C. Vidal, and I. J. Losada, "Validation of OpenFOAM® for Oscillating Water Column three-dimensional modeling," *Ocean Engineering*, vol. 107, pp. 222–236, 2015.

[2] N. G. Jacobsen, D. R. Fuhrman, and J. Fredose, "A wave generation toolbox for the open-source CFD library: OpenFoam," *International Journal for Numerical Methods in Fluids*, vol. 70, no. 9, pp. 1073–1088, 2012.

[3] C. Xu, Z. Huang, and Z. Deng, "Experimental and theoretical study of a cylindrical oscillating water column device with a quadratic power take-off model," *Applied Ocean Research*, vol. 57, pp. 19–29, apr 2016.