Numerical Study on Performance Analysis for OWC WEC Applicable to Breakwater

Sewan Park[#1], Kyong-Hwan Kim[#2], Bo Woo Nam[#3], Jeong-Seok Kim[#4], Keyyong Hong[#5*]

[#]Korea Research Institute of Ships and Ocean Engineering
32 Yuseong-daero 1312 beon-gil, Yuseong-gu, Daejeon, Korea

[1]sewanpark@kriso.re.kr
[2]kkim@kriso.re.kr
[3]bwnam@kriso.re.kr
[4]js-kim@kriso.re.kr
[5*]khong@kriso.re.kr

***Abstract*—The present paper introduces CFD (Computational Fluid Dynamics) analysis results to evaluate the performance of the sloped OWC (Oscillating Water Column) chamber. The CFD analysis method is validated by comparing the results of the 2D wave flume model test for the opened chamber without considering the turbine interaction. The performance of the sloped OWC chamber is evaluated through the 3D CFD analysis by modelling the orifice to take into account the interaction between turbine and oscillating water column in the chamber for various size of the orifice.**

***Keywords*—Oscillating Water Column (OWC), Wave Energy Converter (WEC), Computational Fluid Dynamics (CFD), Orifice Modelling**

I. INTRODUCTION

Oscillating Water Column (OWC) Wave Energy Converter (WEC) has been commonly interested in the past studies and widely used in some of the demonstration projects owing to its advantages, reliability and simplicity [1]. In Korea, a research project for development of 500kW OWC WEC, Yongsoo OWC, has been done since 2003, and real sea demonstration test is proceeding in Jeju Island [2]. In addition, another research project for development of small OWC applicable to breakwaters located in isolated islands has been launched since 2016 funded by Korean government. During the project implementation, a sloped OWC chamber, which is developed to improve constructability for integration with breakwaters, has been designed and studied in the previous study [3] that evaluated effects of various design parameters including the type of the chamber and geometric variables and thus proved prominent performances of the sloped chamber. In the extension of that study, to do 2D CFD simulations in order to validate the previous experimental results presented in [3] in addition to the present CFD method and to carry out 3D CFD analysis considering turbine effects are the objective of this study. Note that many previous researches have been studied a model experiment using 2D wave flume and CFD simulations for a conventional OWC chamber [4]-[11]. This paper is organized as follows. First, the subject of this study is defined, followed by the description of the CFD analysis method. Next, the results and discussions are stated, and finally concluding remarks appear.

II. PROBLEM DESCRIPTION

A. Opened Chamber

To do a validation test for the present CFD analysis method, opened OWC chamber, which was used for the 2D model experiment in [3], was modelled in CFD simulations. Due to openness of the chamber, there are no cover and duct at the top of the chamber, only the skirt and rear wall with a designed chamber length, a horizontal distance between the skirt and the real wall, and a designed skirt depth, a vertical submersion from the mean water level. The CFD simulation was done in 2D with benefits of this problem.

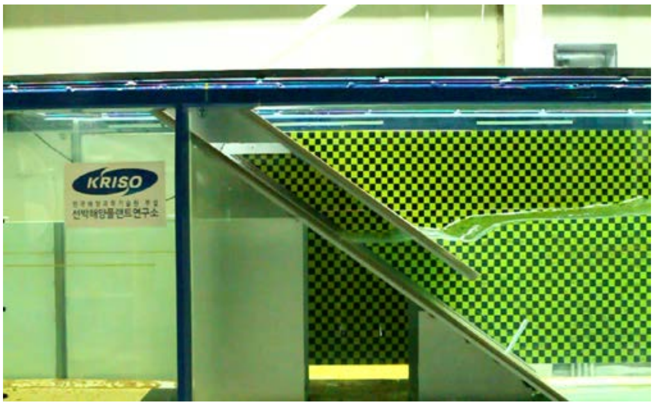


Fig. 1 Snapshot of 2D wave flume experiment performed in previous study [3]

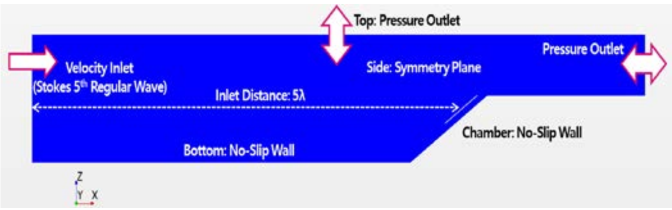
B. Chamber with Orifice

To consider the effect of turbine interactions, the orifice was modelled in the CFD analysis with the presence of the cover and duct at the top of the chamber. The diameter of the duct is 0.04 m in model scale with a ratio of 1/20 and several orifice diameters corresponding to 40% and 60% of the duct diameter in addition to 100%, means that there is no orifice. Among the considered orifice diameter, 40% is a representative size that reflects the performance of the turbine being designed together as part of the project. The background of the selection for the orifice size is beyond the scope of this study and is therefore not covered in this paper. The CFD analysis should be expanded to 3D problem due to the three dimensional geometry of the orifice.

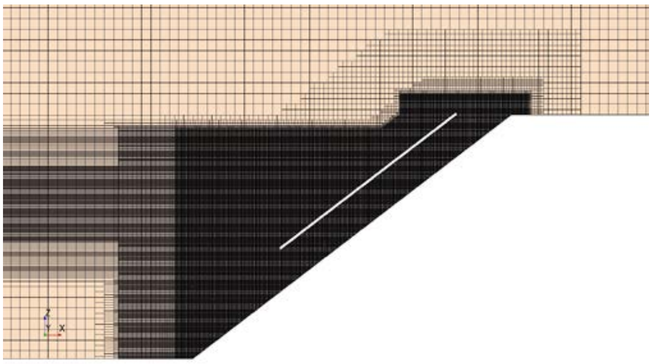
III. CFD ANALYSIS METHOD

A. *Opened Chamber*

CFD simulations were done by using a Star-CCM+, Ver. 11.06. 2D incompressible RANS (Reynolds Averaged Navier-Stokes) equations were used with a turbulence closure, k-epsilon model. VOF (Volume Of Fraction) model was adopted to capture the free surface motion. Time step for unsteady calculations was 1/1,000 to the period of the incoming regular wave, and the Star-CCM+ trimmer mesh with a horizontal and vertical spacing of which is 1/100 to the wave length and 1/20 to the wave height, respectively, was utilized. Meshes with from 300 to 500 thousands cells were used for the analysis domain depending on the incoming wave length. Fig. 2 shows an analysis domain with applied boundary conditions and a mesh used for the CFD calculation in 2D.



(a) Analysis domain and boundary conditions



(b) CFD mesh

Fig. 2 CFD analysis domain with boundary conditions and CFD mesh for opened chamber case

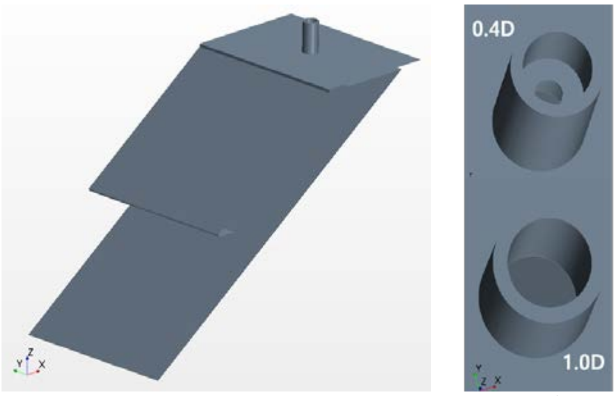
B. *Chamber with Orifice*

Same CFD methods including the calculation physics, time step, meshing strategy, domain selection, and boundary conditions with those used in the opened chamber case were adopted in the 3D simulation considering the orifice. 0.006 m in model scale for the thickness of the orifice was used, and Fig. 3 shows the geometry of the ducted chamber and the orifice. Meshes with from 4 to 9 million cells depending on the incoming wave length were used for 3D simulations.

IV. RESULTS & DISCUSSIONS

A. *Validation Test with Opened Chamber*

An example of the 2D CFD results is shown in Fig. 4 in model scale. Time series for wave elevations extracted at the centre position of the chamber and at the far field are plotted together for a condition of the incoming wave period and height of 1.75 s and 0.07 m, respectively. Fig. 5 shows changes of wave elevation in the chamber over time during one cycle for that wave condition. Simulations for various incoming wave periods and heights were performed. Fig. 6 shows comparison results for the relative wave height, which means the ratio the diffracted wave height in the chamber to the incident wave height, over the incoming wave periods. Note that *Lc* and *Hsk* means the chamber length and the skirt depth in the figure. For two designs which have different geometric variables of the sloped chamber, the measured and calculated relative wave height well agrees, thus the present CFD methods shows a reasonable validity.



(a) Ducted chamber (b) Orifice

Fig. 3 Geometry of ducted chamber and orifice

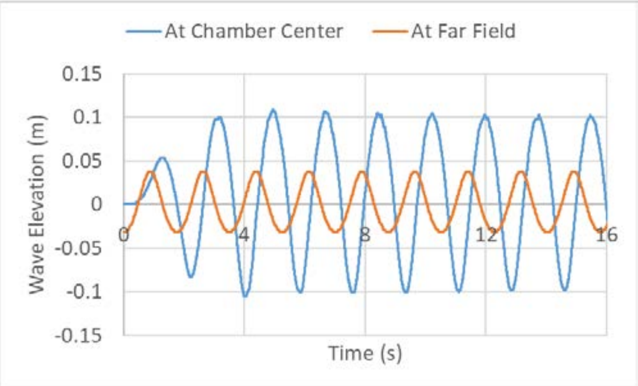
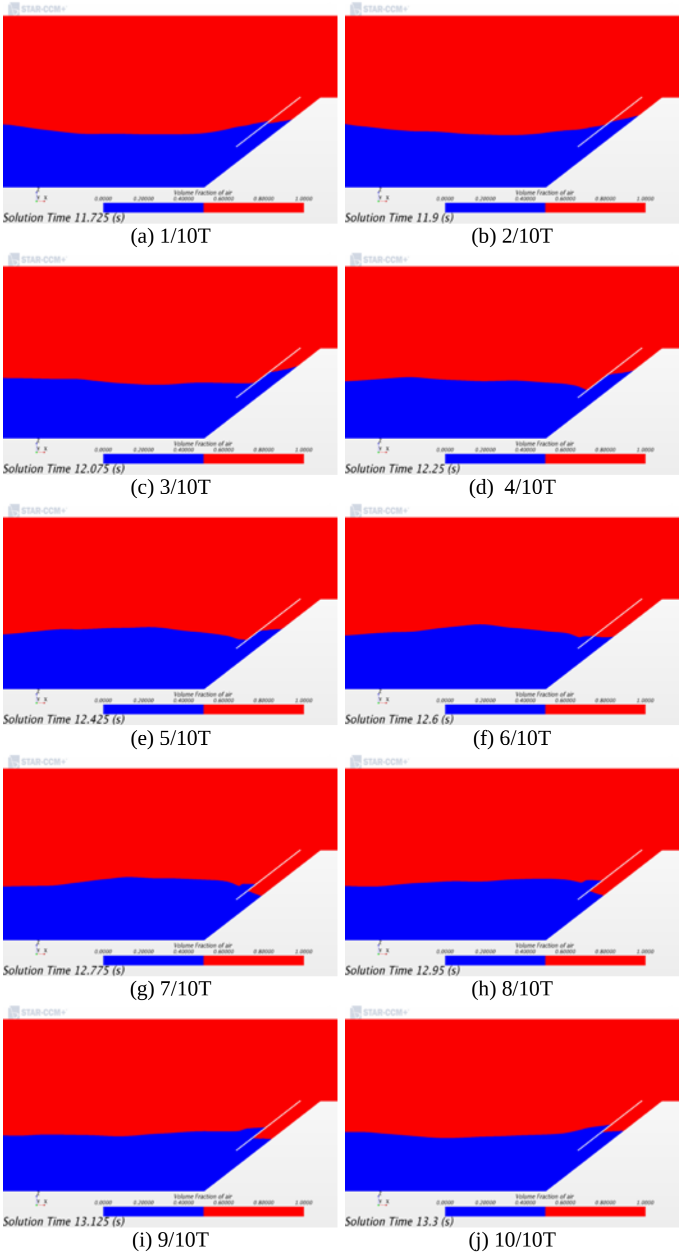


Fig. 4 Time series for diffracted wave elevation in chamber and incident wave elevation at 1.75 s of incoming wave period and 0.07 m of incoming wave height



(a) 1/10T (b) 2/10T

(c) 3/10T (d) 4/10T

(e) 5/10T (f) 6/10T

(g) 7/10T (h) 8/10T

(i) 9/10T (j) 10/10T

Fig. 5 VOF contours over 1/10 interval for one cycle of part of regular wave simulation at 1.75 s of incoming wave period and 0.07 m of incoming wave height

B. *Effects of Turbine Interaction*

3D CFD simulations considering the turbine effect that the pressure drops by the presence of the turbine dampen the induced free surface in the OWC chamber were done by using the validated CFD methods. Fig. 7 shows an example of the simulation, time series for wave elevation in the chamber, differential pressure between before and after the orifice, and air flow speed through the orifice for a condition of the incoming wave period and height of 1.83 s and 0.0241 m, respectively. Change of water surfaces in the chamber and the pressure distributions synchronized each together during one cycle among the plotted section in Fig. 7 was shown in Fig. 8.

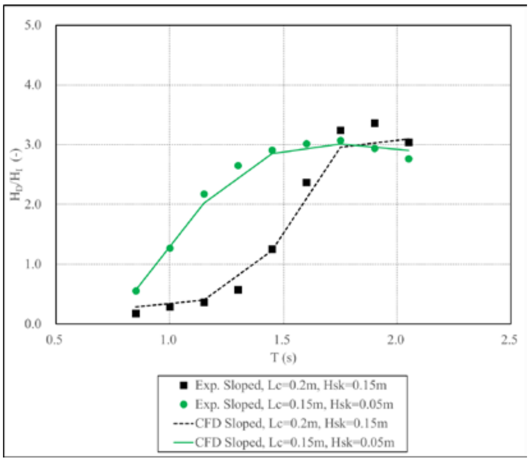


Fig. 6 Comparison between experimental results and CFD results for relative wave height induced in chamber

Whole of regular wave simulations are summarized in Fig. 8. The relative wave height, the differential pressure between before and after the orifice, and the air flow speed through the orifice were plotted over various incoming wave periods. In Fig. 9 (a), results from the opened chamber are compared together, even though it has slightly different geometric variables. The relative wave height is significantly reduced by the presence of the orifice, and it is shown that the smaller the orifice diameter, the greater the reduction of the diffracted wave elevations in the chamber. The differential pressure and the air flow speed are categorized into two different processes, rise and fall by the regular waves. The largest differential pressures is formed under the 0.4D orifice condition even though nearly negligible levels of differential pressures are shown for the 1.0D orifice condition.

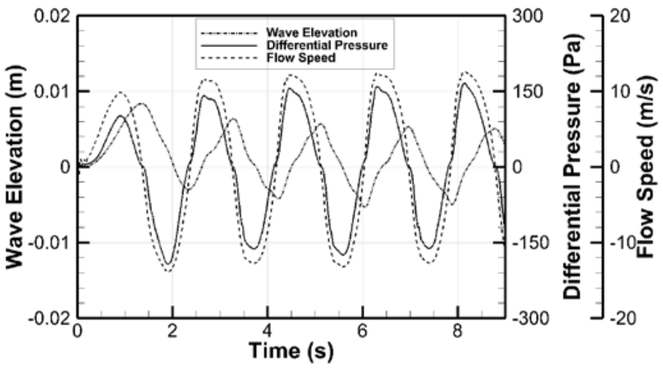
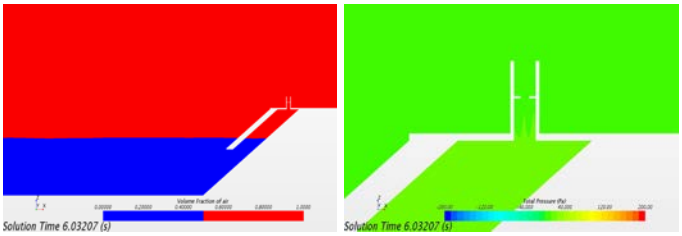
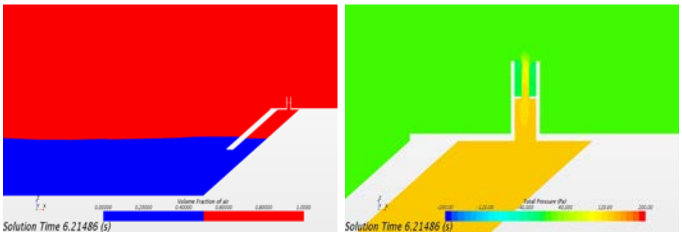


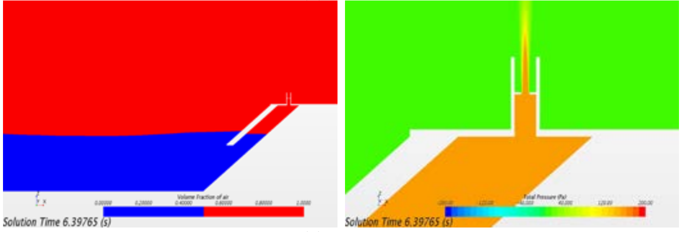
Fig. 7 Time series for wave elevation in chamber, differential pressure, and air flow speed at 1.83 s of incoming wave period and 0.0241 m of incoming wave height



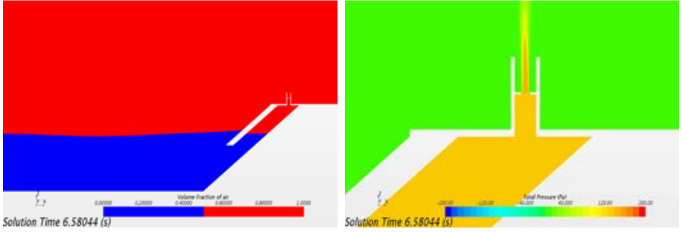
(a) 1/10T



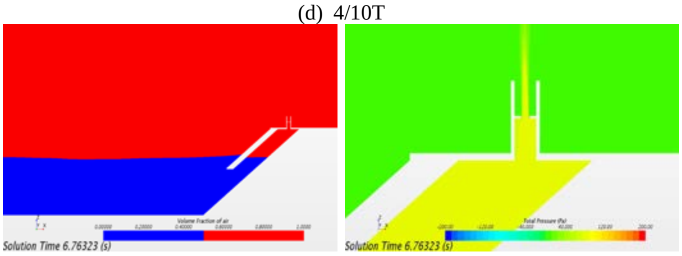
(b) 2/10T



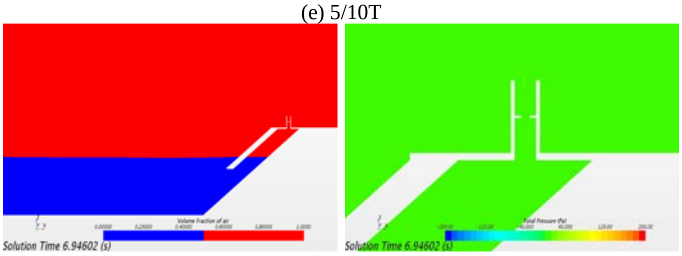
(c) 3/10T



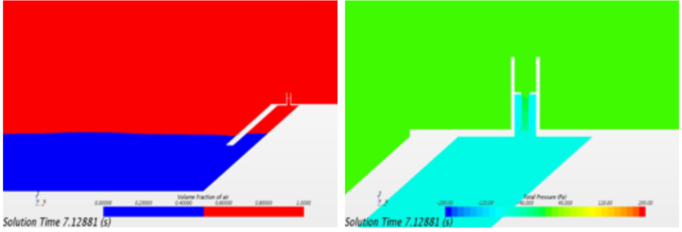
(d) 4/10T



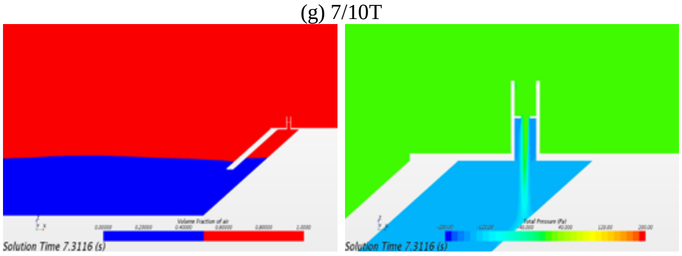
(e) 5/10T



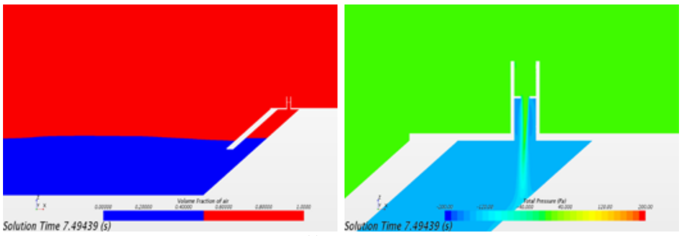
(f) 6/10T



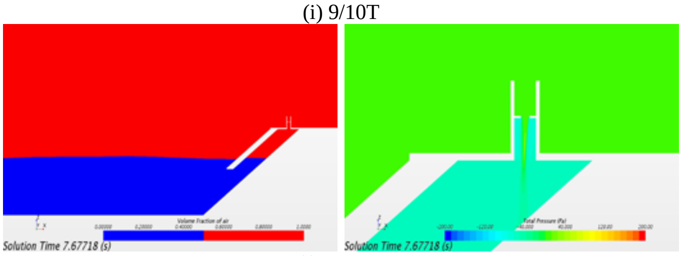
(g) 7/10T



(h) 8/10T



(i) 9/10T



(j) 10/10T

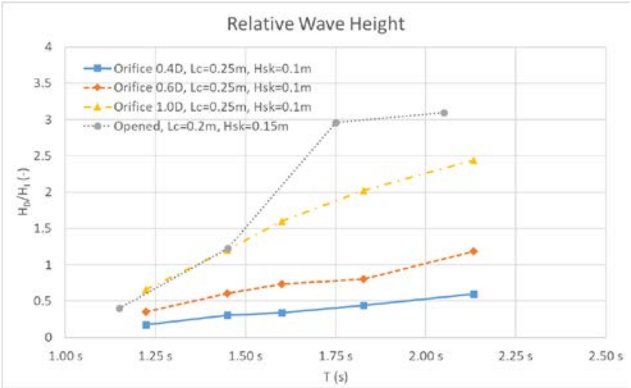
Fig. 8 VOF and pressure contours over 1/10 interval for one cycle of part of regular wave simulation at 1.83 s of incoming wave period and 0.0241 m of incoming wave height

V. CONCLUDING REMARKS

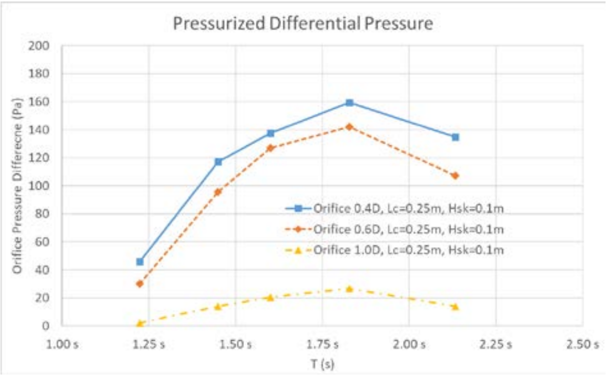
In this study, 2D CFD simulations without considering the turbine effects for the purpose of validation by comparing with the previous experimental results and 3D CFD analysis with considering the turbine interactions for understanding the effects were done. Presented CFD method was validated with a reasonable agreement, and effects for various size of the orifice were discussed. The processes of pressurizing and depressurizing of the OWC chamber by the presence of the orifice well simulated, and the damping effects of the orifice were evaluated based on the simulation results for various orifice size. In the future, model experiments considering the orifice in a scaled model will be done and validation of the CFD methods is also planned by comparing those results each together.

ACKNOWLEDGMENT

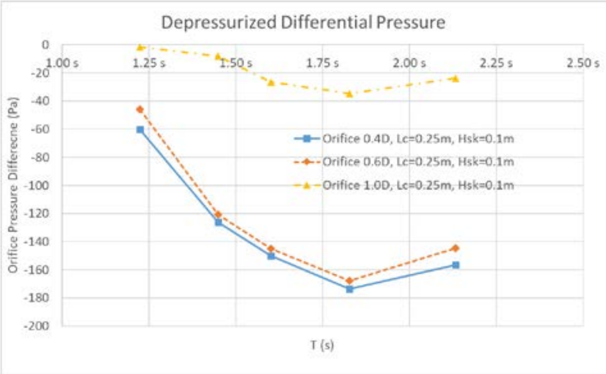
This research was supported by a grant from National R&D Project “Development of Wave Energy Converters Applicable to Breakwater and Connected Micro-Grid with Energy Storage System“ funded by Ministry of Oceans and Fisheries, Korea(PMS3780). All support is gratefully acknowledged.



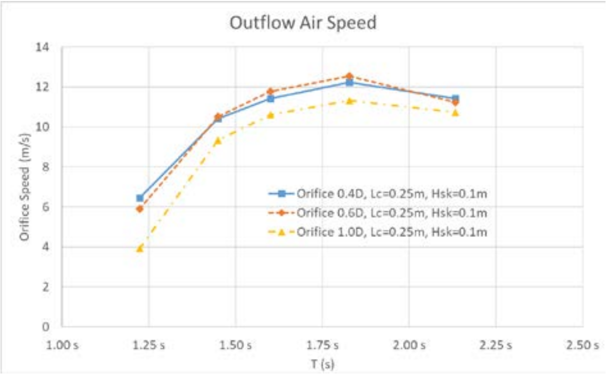
(a) Relative wave height



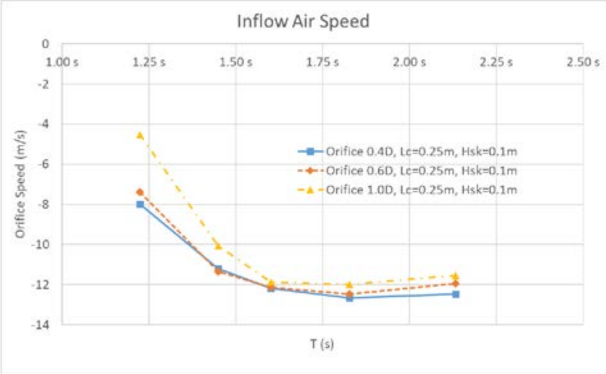
(b) Pressurized differential pressure



(c) Depressurized differential pressure



(d) Outflow air speed



(e) Inflow air speed

Fig. 9 OWC chamber performance characteristics for various orifice size in regular waves

REFERENCES

[1] A. F. O. Falcão and J. C. C. Henriques, “Oscillating-water-column wave energy converters and air turbines: A review,” *Renewable Energy*, vol. 85, pp. 1391-1424, Jan. 2016.

[2] The Executive Committee of Ocean Energy Systems, “Annual report: An overview of activities in 2017,” Ocean Energy Systems, IEA OES, 2017.

[3] S. Park, B. W. Nam, K-H. Kim, K. Hong, “Parametric study on oscillating water column wave energy converter applicable to breakwater,” *Journal of Advanced Research in Ocean Engineering*, 2018 (Under Review).

[4] A. Elhanafi, A. Fleming, G. Macfarlane, Z. Leong, “Numerical hydrodynamic analysis of an offshore stationary-floating oscillating water column-wave energy converter using CFD,” *Journal of Naval Architecture and Ocean Engineering*, vol. 9, pp. 77-99, Jan. 2017.

[5] T. Vyzikas, S. Deshoulières, O. Giroux, M, Barton, D. Greaves, “Numerical study of fixed oscillating water column with RANS-type two-phase CFD model,” *Renewable Energy*, vol. 102, pp. 294-305, Mar. 2017.

[6] I. López, A. Castro, G. Iglesias, “Hydrodynamic performance of an oscillating water column wave energy converter by means of particle imaging velocimetry,” *Energy*, vol. 83, pp. 89-103, Apr. 2015.

[7] I. López, B, Pereiras, F. Castro, G. Iglesias, “Optimisation of turbine-induced damping for an OWC wave energy converter using a RANS-VOF numerical model,” *Applied Energy*, vol. 127, pp. 105-114, Aug. 2014.

[8] A. Iturrioz, R. Guanche, J. L. Lara, C. Vida, I. J. Losada, “Validation of OpenFOAM ® for oscillating water column three-dimensional modeling”, *Ocean Engineering*, vol. 107, pp. 222-236, Oct. 2015.

[9] A. Kamath, H. Bihs, Ø. A. Arntsen, “Numerical modeling of power take-off damping in an oscillating water column device”, *International Journal of Marine Energy*, vol. 10, pp. 1-16, Jun. 2015.

[10] A. Kamath, H. Bihs, Ø. A. Arntsen, “Numerical investigations of the hydrodynamics of an oscillating water column device”, *Ocean Engineering*, vol. 102, pp. 40-50, Jul. 2015.

[11] Y. Luo, J-R. Nader, P. Cooper, S-P. Zhu, “Nonlinear 2D analysis of the efficiency of fixed oscillating water column wave energy converters”, *Renewable Energy*, vol. 64, pp. 255-265, Apr. 2014.