Hydrodynamic analysis of a full-scale U-OWC breakwater: comparison between analytical models and CFD simulations

Luana Gurnari , Pasquale Filianoti, Sergio M. Camporeale and Marco Torresi

Abstract—**The U-OWCs are a family of Wave Energy Converter, belonging to Oscillating Water Column devices. The peculiarity is a duct in front of the wave beaten wall, forming a U-conduit with the plenum with a U-shape of the vertical duct.**

In this work, we put in comparison a two-dimensional mathematical model of a full scale U-OWC plant with a 1D model. We prove that, after a quick calibration of the 1D model, we obtain a perfect correspondence between simulations, in terms of power absorption, water discharge, pressure fluctuation, water level and so on. Also the thermal transformation of the air mass contained inside the plenum is quite similar. A well performing 1D mathematical model permit us to make, very quickly, a high number of simulations in order to fix the main geometrical and working parameters of the plant, letting to the 2D simulation the work of refinement.

In this work we carried out a comparison between a bi-dimensional numerical simulation on a full-scale model of an U-OWC with the one-dimensional numerical model. The bi-dimensional simulations are clearly more complete than 1D ones, but, obviously, increase the computational effort. After a quick calibration the one-dimensional model shows a perfect correspondence with the 2D simulation. This 1D model can be used to design, preliminarily, an U-OWC device, quickly. Therefore, is possible to optimize the shape of the absorber, reducing internal head losses improving the overall plant efficiency

Index Terms—**Computational Fluid Dynamics (CFD), Oscillating Water Column (OWC), Porous media, Volume Of Fluid, Wells Turbine.**

I. Introduction

AN Oscillating Water Column device consists of a chamber with an opening to the sea below the water level. Under the wave action, the surface of water column rises and compresses (or decreases and decompresses) the air volume in the plenum chamber. The air is forced through a duct containing a self-rectifying turbine, which connects the plenum to the atmosphere. The REsonant Wave Energy Converters ([1], [2], [3], [4], [5]), belongs to the family of OWC. The main difference in respect to a conventional OWC is the presence of an additional vertical duct on the wave beaten wall conferring to the structure a typical U–shape. The first experimental test on a small-scale model of a U-OWC plant was placed in Reggio Calabria [5] in the eastern coast of the Straits of Messina (Southern Italy). The performance of a U-OWC with a Wells turbine has been investigated through a small-scale field experiment by [3]. The first prototype of a full-scale U-OWC is under construction in Civitavecchia harbour (Rome, Italy) in the Tyrrhenian sea and the plant is described in [6]. [7] and [8] carried out a numerical simulation in a 2D wave flume equipped by a wave maker piston type to generate waves. The U-OWC has the same size of the plant described in [6] and it was subjected to different wave conditions. The multi-phase flow, in the CFD simulation, was solved by the unsteady Reynolds-Averaged Navier-Stokes equations using the commercial code Ansys Fluent 17.0, Academic version. To reproduce the pressure drop induced by the presence of the Wells turbine, we set inside the air tube a porous medium. Analysing the interaction between waves and the full-scale model of the U-OWC plant we were able to reproduce the hydrodynamic behaviour of the plant. These analyses highlighted the importance of a preliminary design of the plant, being the capability of energy absorption strongly influenced by shape of the plant. In this work, in order to determine a preliminary design, fixing relationship to the dimension of the chamber and the vertical duct, we made numerical simulation making use of analytical 1-D models, already developed by [5]. The 1-D simulations have been carried out considering the records of the fluctuating pressure head on the outer opening of the plant obtained by means of the CFD simulation. The paper shows comparisons of 1D simulations with the results of CFD simulation. This procedure can be applied to optimize the shape of the absorber reducing internal head losses in order to improve the overall plant efficiency.

Paper ID number: 1465 - Conference track: Wave device development and testing

S. Camporeale and M. Torresi are with Department of Mechanics,Mathematics and Management (DMMM), Politecnico di Bari, Bari, via Orabona 4, 70125 Bari - ITALY (sergio.camporeale@poliba.it;marco.torresi@poliba.it)

P. Filianoti and L. Gurnari are with Department D.I.C.E.A.M., University Mediterranea of Reggio Calabria Via Graziella, Loc. Feo di Vito - 89122 Reggio Calabria – ITALY (filianoti@unirc.it; luana.gurnari@unirc.it).

II. 1D Mathematical Model: The Flow Motion Inside the Absorber

The flow motion inside the absorber was studied by [9], [10]. Referring to the scheme of Fig. 1, the equation of the water flow inside the absorber can be expressed as

$$\frac{l^{''}-\xi}{g}\frac{\mathrm{d}^2\xi}{\mathrm{d}t^2}+\frac{l^{'}}{g}\frac{\mathrm{d}u}{\mathrm{d}t^2}=h^{'}-h^{''}-\Delta h_w, \quad (1)$$

where $l^{'}$ is the length of the vertical duct, ξ is the height of the air pocket, $l^{''}$ is the height of the inner room, u is the water velocity in the vertical duct (positive

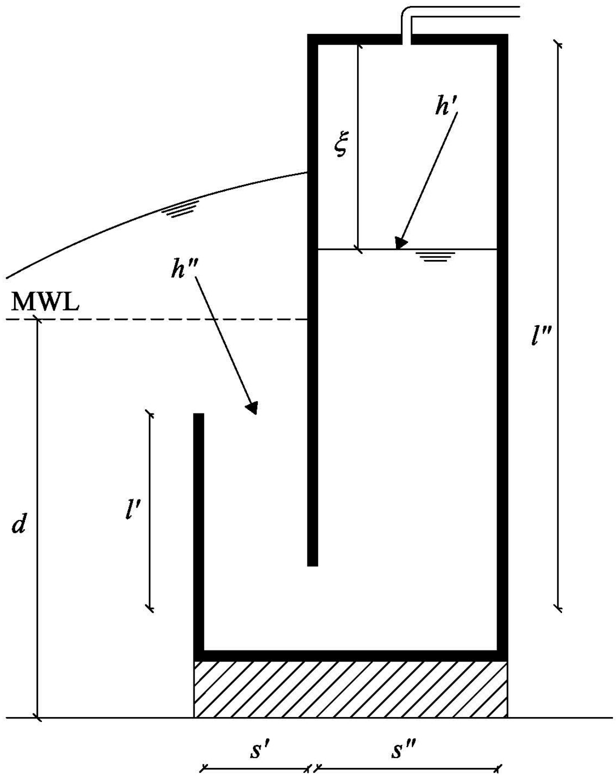

Fig. 1. Reference scheme of U-OWC.

upward) and Δh_w is the head losses (both continuous and minor) defined as:

$$\Delta h_w = \left(\frac{\lambda_w}{4\Re} l' + K_w\right) \frac{|u|\, u}{2g}, \tag{2}$$

where $\Re$ is the hydraulic radius, λ_w is the friction coefficient and K_w is the minor loss coefficient, assumed as unique value independently from the flow direction. The velocity u in the vertical duct is related to the time derivative of ξ by the continuity equation

$$u = \frac{s''}{s'} \frac{\mathrm{d}\xi}{\mathrm{d}t}, \tag{3}$$

where s' is the width of the vertical duct, and s'' is the width of the oscillating water column.

The energies per unit weight are

$$h' = \xi_0 - \xi + \frac{1}{2g}\left(\frac{\mathrm{d}\xi}{\mathrm{d}t}\right)^2 + \frac{p_a - p_{atm}}{\rho g}, \tag{4}$$

$$h'' = \frac{\Delta p}{\rho g}, \tag{5}$$

where p_a is the pressure in the air pocket, p_{atm} is the atmospheric pressure, Δp is the wave pressure on the opening of the vertical duct and ξ_0 is the still water value of ξ.

The air density in the plenum is

$$\rho_a = \frac{M_a}{b s'' \xi}, \tag{6}$$

and it varies with time because of variations of air mass M_a and because of variations of height ξ. The pressure in the air pocket is related to the air density by the equation of state, according to the polytrophic law:

$$\frac{p_a}{p_{atm}} = \left(\frac{\rho_a}{\rho_{atm}}\right)^m, \tag{7}$$

being m the exponent of polytrophic.

The velocity in the air tube u_a, is related to pressure p_a and to the length of the vent tube,l_a, by the relation

$$l_a \frac{\mathrm{d}u_a}{\mathrm{d}t} = \frac{p_a}{\rho_{atm}} \frac{m}{m-1}\left[\left(\frac{p_a}{p_{atm}}\right)^{1-\frac{1}{m}} - 1\right] - g\Delta h_a, \tag{8}$$

where

$$\Delta h_a = \frac{1}{2g}\left(\frac{\lambda_a}{D} l_a + K_a\right)|u|\, u + \frac{F_t}{g} u_a \tag{9}$$

accounts for the energy (per unit weight) dissipation in the air tube due to continuous (friction factor:λ_a) and minor (loss coefficient: K_a) losses, and due to the pressure drop across the turbine (applied turbine damping factor:F_t).

Considering operating conditions into which the turbine damping factor B_t is almost constant, F_t can be determined from the geometrical characteristics and the rotational speed of the turbine:

$$F_t = \frac{B_t U_{tip}}{2} \frac{A_a}{At}, \tag{10}$$

where U_{tip} is the tangential velocity at the blade tip, and A_t is the turbine bladed area.

Expanding terms Δh_a, and F_t in (8) and (9), respectively by means of (9) and (10), we obtain

$$l_a \frac{\mathrm{d}u_a}{\mathrm{d}t} + B_a |u_a|\, u_a - \frac{p_a}{\rho_{atm}} \frac{m}{m-1}\left[\left(\frac{p_a}{p_{atm}}\right)^{1-\frac{1}{m}} - 1\right] + F_t u_a = 0 \tag{11}$$

where

$$B_a \equiv \frac{1}{2g}\left(\frac{\lambda_a}{D} l_a + K_a\right) \tag{12}$$

The rate of change of the air mass in the air pocket is related to a_a and ρ_a :

$$\frac{\mathrm{d}M_a}{\mathrm{d}t} = -\frac{1}{2}\left(\rho_a + \rho_{atm}\right)\frac{\pi D^2}{4} u_a \tag{13}$$

Equations (1) and (11) are integrated numerically from the knowledge of Δp and the initial conditions at time $t = 0$:

$$\frac{\mathrm{d}\xi}{\mathrm{d}t} = 0; \qquad \xi = \xi_0; \qquad M_a = \rho_{atm} b s'' \xi_0. \tag{14}$$

The mathematical model has been validated and calibrated against the experimental results obtained in the field on a small-scale model [5].

III. 2D NUMERICAL MODEL

The computational domain was described in [7] and [8], and it is constituted by a wave-flume, having a piston-type wavemaker placed in the left extremity and a U-OWC breakwater in the right extremity (see Fig. 2). The 2D wave flume is 1km long and 30m high. The length of the flume has been chosen in

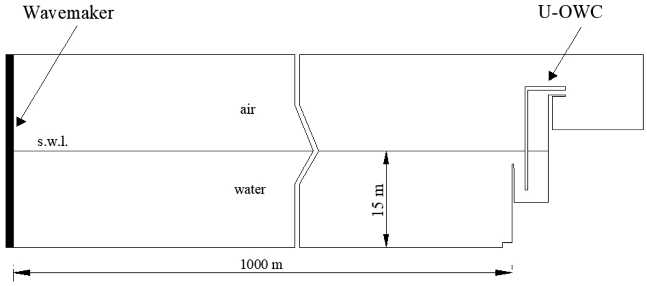

Fig. 2. Sketch of the computational domain.

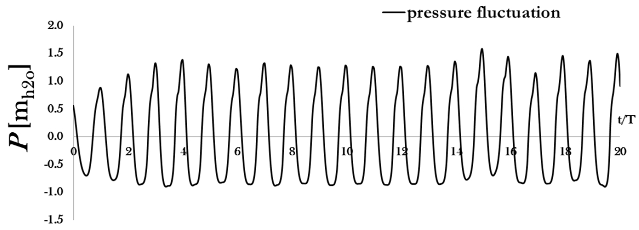

Fig. 4. Pressure fluctuation at the outer opening of the plant, produced by a wave train with Hs=3.5m and Tp=8s.

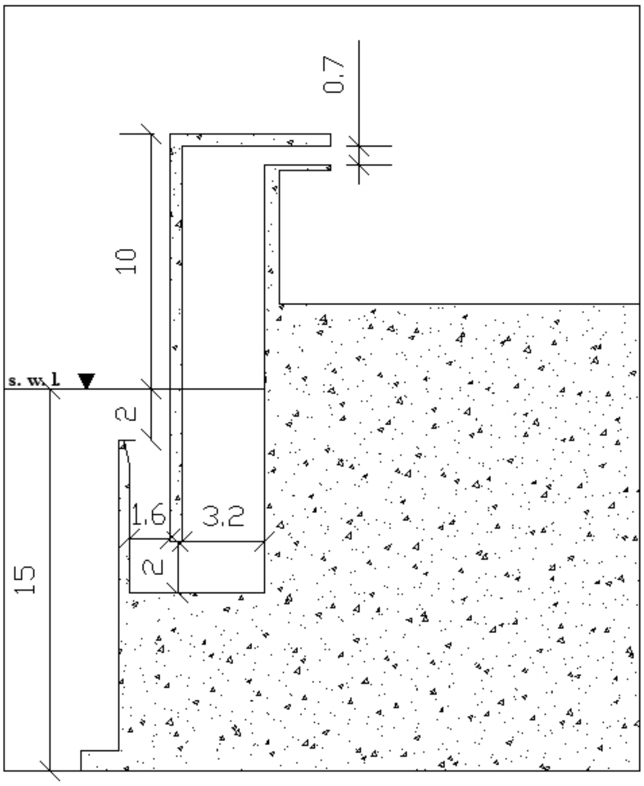

Fig. 3. The cross section of the caisson of the breakwater converter installed at Civitavecchia Harbour (measures are in meter).

order to have enough distance (at least 10 wavelengths) between the wavemaker and the U-OWC breakwater to achieve stationary conditions inside the plant after the incoming waves impact the breakwater wall.

The plant has the same size as the U-OWC installed at the Civitavecchia harbour in Rome (Italy) in the central Mediterranean Sea, and described in [6] and shown in Fig. 3. The spatial discretization of the computational domain was made by a hybrid mesh, formed by rectangular elements in the overall length of the flume, whereas near the U-OWC device triangular elements were adopted. The mesh has approximately 300,000 cells. Starting from the water at rest, the wave generation process has been simulated assigning a sinusoidal motion to the left wall of the wave flume, by means of a User Defined Function (UDF). The maximum displacement of the piston type wavemaker was calculated referring to the complete first order piston wavemaker solution, reported by [11]].

The two-dimensional CFD simulation was conducted using the commercial code Ansys Fluent v.17 academic version. The water-air interaction is taken into account by means of the Volume Of Fluid (VOF) model. Both air and water flow fields are assumed to be unsteady and are computed solving the Reynolds-Averaged Navier-Stokes (RANS) equations. These equations are discretized according to a Finite Volume approach, adopting a pressure-based algorithm in its implicit formulation. In order to consider effects like the convection and the diffusion of turbulent energy we used the Standard k- turbulence model. The two-dimensional CFD simulation is carried out, in order to reduce the computational effort and resources with respect to a full three-dimensional simulation. The conduit, in which a Wells turbine was placed, has a diameter of 0.75 m. In particular, the pressure drop induced in this region by the presence of the turbine, was reproduced by a porous medium. It is characterised by viscous and inertial losses which are used to reproduce the actual pressure losses in the air duct according with the procedure described by [7] and [8].

IV. The calibration procedure

In the present work the sea state 7 reported in [6] has been simulated. In detail, we simulated a wave train formed by periodic waves energetically equivalent to the sea state 7 which have significant height H_s= 3.5 m and T_p = 8 s. The simulated periodic waves have the wave period T equal to the peak period T_p of the sea state spectrum and the wave height H, equivalent, in terms of energy spectrum, to the significant wave height H_s. When the incident wave train hits the absorber-breakwater, the water inside the plant starts to oscillate, alternatively compressing and expanding the air mass in the plenum. In Fig. 4 is shown the pressure fluctuation obtained by means of the 2D numerical simulation. It was evaluated considering the average of the pressure fluctuation in 5 points placed along an horizontal cross section inside the vertical U-duct.

Results of numerical 2D simulations are already been described in [7] and [8]. The pressure fluctuation at the outer opening was used as an input for the one-dimensional model. The first step is to calibrate the 1D numerical model in term of loss coefficient K_a, and K_w (see eqs.(12) and (2) respectively). We found that the optimal value of the loss coefficient is K_a=1.5 and K_w=0.75. Friction coefficients, λ_a and λ_w was set 0.025 and 0.16, respectively. For what concern the exponent

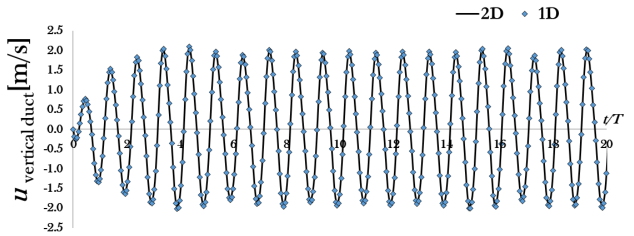

Fig. 5. water velocity inside the vertical U-duct. Comparison between 2D (continuous line) and 1D (points) simulation. The latter is obtained though(3).

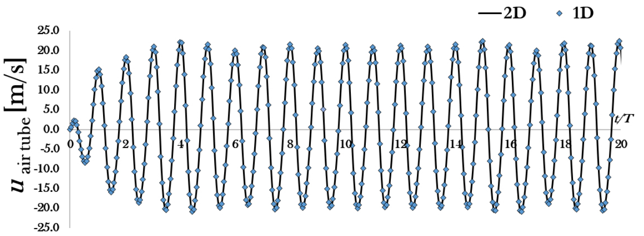

Fig. 7. Air velocity inside the vent tube. Comparison between 2D (continuous line) and 1D (points) simulation. The latter is obtained though (11).

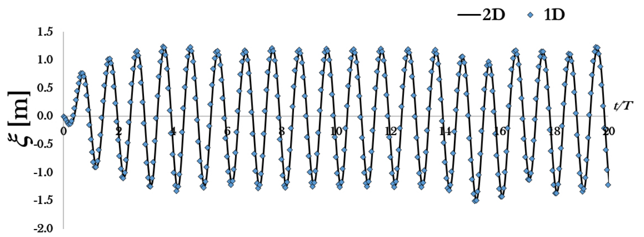

Fig. 6. Water surface displacement inside the chamber. Comparison between 2D (continuous line) and 1D (points) simulation.

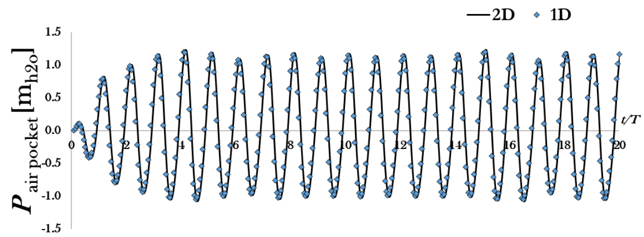

Fig. 8. Pressure fluctuation in the air chamber. Comparison between 2D (continuous line) and 1D (points) simulation.

of the polytropic law, we found a perfect correspondence between 1D and 2D simulations considering an isentropic transformation. The parameter of the turbine was set considering the characteristic curve shown in [6].

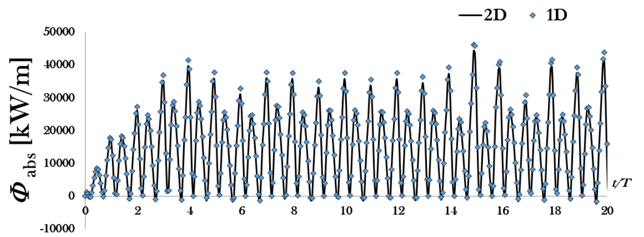

Fig. 9. Instantaneous energy flux absorbed by the plant. Comparison between 2D (continuous line) and 1D (points) simulation.

V. Comparison between 2D and 1D results

Starting from the pressure fluctuation at the outer opening of the plant and following the procedure described in sect. II, and IV we were able to evaluate all fluid dynamic variables to estimate the energy absorbed by the plant though the 1D model. In Fig.5 a comparison between the 2D and 1D values of the vertical velocity, u, in the vertical U-duct is shown. Points are been obtained by means of 1D simulation, solving the equation of the water flow inside the absorber. In detail this model solves as a first step the variation in time of the water free surface displacement ξ , in the chamber and then, using the continuity equation (3) between the chamber and the vertical duct, it obtains the velocity in the vertical duct.

Integrating $\mathrm{d}\xi/\mathrm{d}t$, it was obtained the water free surface displacement inside the chamber. As we can see in Fig. 6, there is a perfect match between 2D and 1D simulations.

Fig. 7 shows the comparison between 1D and 2D simulations of the velocity of the air in the vent tube. The results of numerical 2D simulations were obtained in the inlet section of the vent tube in which was set the porous zone to reproduce the pressure drop induced by the presence of the turbine. Points represent the 1D simulations and they are been obtained solving the(11).

At the same cross section of the vent tube, was evaluated the pressure fluctuation for the bidimensional numerical simulation. It was been compared with the non dimensional one calculated solving the equation of state as we can see in Fig. 8.

Finally, we estimated the energy flux absorbed by the plant, multiplying the pressure fluctuation to the vertical velocity inside the duct. As we can see in Fig. 9, we have an excellent correspondence between the two simulations.

The mean energy flux absorbed by the plant is equal to 15.6 kW/m, evaluated though both 2D and 1D simulations.

VI. Conclusions

Oscillating Water Column devices are the most common Wave Energy Converters. They are able to convert a large share of incident wave energy into pneumatic energy by means of a PTO system. The U-OWCs are a particular kind of Oscillating Water Column device having a U-shape of the vertical duct on beaten side of the breakwater wall. Under the wave action the water inside the chamber oscillates and forced the air through the air duct in which is placed a self-rectifying turbine. Therefore, the more the oscillation period is close to the eigenperiod of the plant the more the plant itself captures energy. It is clear that the optimal design of the plant is fundamental in capturing energy from waves. CFD (both 2D and 3D) is a powerful and effectiveness tool to reach this goal. Anyway, the use of CFD from the earlier stages of the plant design procedure is too costly from the view of computational time. The use of a 1D numerical model reduces dramatically the computational effort in the preliminary design of the plant, keeping the role of CFD for the refining of the solution.

To this aim we have compared the numerical results obtained through a bidimensional simulation with those obtained using a one-dimensional numerical model. The bidimensional simulation was conducted using a commercial code Ansys Fluent v.17 academic version. The results of the one-dimensional simulation were obtained researching numerically the equations shown in sect. II. In the 2D simulations, to reproduce the pressure drop induced by the presence of the turbine, inside the air duct was set a porous media. The one-dimensional model the presence of the turbine, including a viscous loss in the air duct equation, are been considered an additional term of losses, evaluated knowing the characteristic curve of the Wells turbine. Results show the capability of the 1D model to reproduce results strictly fitted to those obtained with 2D model, confirming the capability of the 1D model to represent a useful tool to investigate the dynamic behaviour of the U-OWC in the preliminary stage of their design.

Acknowledgement

he computing resources and the related technical support used for this work have been provided by CRESCO/ENEAGRID High Performance Computing infrastructure and its staff ([12]). CRESCO/ENEAGRID High Performance Computing infrastructure is funded by ENEA, the Italian National Agency for New Technologies, Energy and Sustainable Economic Development and by Italian and European research programmes, see http://www.cresco.enea.it/english for information".

References

[1] P. Boccotti, "6450732 B1," *US Patent*, 2002.
[2] F. Arena and P. Filianoti, "A small-scale field experiment on a submerged breakwater for absorbing wave energy," *ASCE Journal of Waterway, Port, Coastal, and Ocean Engineering*, vol. 133, pp. 161–167, 2007.
[3] P. Filianoti and S. M. Camporeale, "A small-scale field experiment on a wells turbine model," *proceedings of 7th European Wave and Tidal Energy Conference, 11 - 14 Sept., Porto.*, 2007.
[4] P. Filianoti and R. Piscopo, "Sea wave energy transmission behind submerged absorber caissons," *Ocean Engineering*, no. 93, pp. 107–117, 2015.
[5] P. Boccotti, P. Filianoti, V. Fiamma, and F. Arena, "Caisson breakwaters embodying an OWC with a small opening - Part II: A small-scale field experiment," *Ocean Engineering*, no. 34, pp. 820–841, 2007.
[6] F. Arena, V. Fiamma, V. Laface, G. Malara, A. Romolo, and F. M. Strati, "Monitoring of the U-OWC under construction in Civitavecchia (Rome, Italy)," *Proceedings of the 11th European Wave and Tidal Energy Conference (EWTEC), 6-11 September 2015, Nantes, France*, 2015.
[7] P. Filianoti, L. Gurnari, M. Torresi, and S. M. Camporeale, "CFD analysis of the energy conversion process in a fixed oscillating water column (OWC) device with a Wells turbine," *Energy Procedia*, vol. 148, pp. 1026–1033, 2018.
[8] F. Scarpetta, L. Gurnari, M. Torresi, P. Filianoti, and S. Camporeale, "A CFD simulation of a full-scale U-OWC breakwater," *Proceedings of the 11th European Wave and Tidal Energy Conference (EWTEC), 6-11 September 2015, Nantes, France*, 2015.
[9] P. Boccotti, "Caisson breakwaters embodying an OWC with a small opening - Part I: Theory," *Ocean Engineering*, no. 34, pp. 806–819, 2007.
[10] ——, "On a new wave energy absorber," *Ocean Engineering*, no. 30, pp. 1191–1200, 2003.
[11] R. G. Dean and R. A. Dalrymple, "Water wave mechanics for engineers and scientists," *Prentice Hall, Inc. Reprinted Singapore: World Scientific Publishing Co.*, no. 30, pp. 170–178, 1984.
[12] G. Ponti, "The role of medium size facilities in the HPC ecosystem: the case of the new CRESCO4 cluster integrated in the ENEAGRID infrastructure," *Proceedings of the 2014 International Conference on High Performance Computing and Simulation*, pp. 1030–1033, 2014.