Hydrodynamic Properties of a Moored Floating Breakwater using CFD Approach

S.F. Abdullah¹, A. Fitriadhy¹, M.H. Mohd¹, M.A.A. Rahman¹, Suntoyo², A. Kurniawan³ and W. Pratikto²

¹Program of Maritime Technology, Faculty Ocean Engineering Technology and Informatics, Universiti Malaysia Terengganu, Malaysia.
²Department of Ocean Engineering, Faculty of Marine Technology, Institut Teknologi Sepuluh Nopember (ITS), Surabaya, Indonesia.
³Program of Ocean Engineering, Faculty of Civil and Environmental Engineering, Institut Teknologi Bandung, Indonesia
Email: sfakhuradzi@gmail.com

Abstract. In presence of complex-hydrodynamic interaction between water wave and moving structure, a reliable method that can analyse nonlinear phenomena is necessarily required. This paper presents numerical investigation of a moored floating breakwater using computational fluid dynamic (CFD) approach. The mathematical model is based on the extended Reynolds Average Navier-Stokes (RANS) solver for solid-porous obstacle. A high amplitude wave with several wave periods were deliberately considered in the simulation to allow nonlinear wave effects on the floating structure such as wave breaking, overtopping, including viscous friction. Here, the two fluid calculation method for interface boundary between water and air is proposed to capture the complex free surface changes. In addition, the fractional average volume obstacle representation (FAVOR) using partial cell treatment method is employed to simulate the motion of breakwater boundary on the free surface. Approximations and validations on the hydrodynamic properties of the structure have been carried out which include wave transmission coefficient, sway, heave, pitch, and mooring forces. The results show that the CFD model can fairly simulate well on hydrodynamics of the floating breakwater. The discrepancies between numerical and experimental data can be partly attributed to the nonlinearity in the incident wave definition.

1. Introduction
Floating breakwaters are engineering structures basically applied to protect ports and harbours from wave action. Also, they can be an alternative measure for shoreline protection compared to conventional breakwaters, considering the compatibility with incoming wave energies [1]. Furthermore, floating breakwaters offer several advantages of application in terms of low capital cost, deeper water depth, flexibility as well allowing continuous refreshment of coastal water. In designing of such coastal structures hence, the integrated hydrodynamic modelling using numerical and laboratory experiments increasingly becomes necessary.
Numerical modelling of floating breakwaters involving moving boundary, breaking process, and viscous effects remains a challenging task in the field of ocean and coastal engineering. Most of the studies were performed using potential flow theory. The model uses linearised governing equations and free surface boundary conditions. When the flows include complex-nonlinear phenomena such as wave breaking
and overtopping, the model cannot produce meaningful results [2]. In presence of these extreme phenomena, a reliable method that can analyse nonlinear wave effects is essentially required. The advancement in the computer programming has led to the development of numerical models and sophisticated software platforms to simulate more complicated flows, which is based on the computational fluid dynamic (CFD). Smoothed Particle Hydrodynamic (SPH) is one of the CFD simulation methods. SPH has shown high potential for simulation of a wide range of numerical models such as; coastal waves, wave form of landslide, wave impact on structures and dams [3]. However, the method is developed based on assumptions on the fluid compressibility, lacking of boundary condition consistency in the continuous approximation, and susceptible to modelling instability in some circumstances; involving bubble generation.

In the interest of an accurate modelling with reasonable computational time, this paper presents numerical investigation on the hydrodynamic properties of a moored floating breakwater using Reynolds Average Navier-Stokes (RANS) solver. Here, twin pontoons with attached fishing net of floating breakwater is employed. The two fluid calculation method and the fractional average volume obstacle representation (FAVOR) using partial cell treatment method are proposed to simulate the complex-nonlinear phenomena including breakwater motion on the free surface. Several wave periods with a high wave amplitude were deliberately considered in the simulation to allow many nonlinear wave effects. The present study focuses on the wave transmission coefficient, sway, heave, pitch, and mooring forces.

2. Numerical model

2.1. Governing equations of fluid flow

A numerical model based on the Reynolds Average Navier-Stokes (RANS) equations coupled with $k - \varepsilon$ turbulence closure model and the volume of fluid (VOF) method is presented in this research. For simulating wave motion across the permeable floating breakwater, the fluid inertia and drag resistances were introduced into the continuity and momentum conservation equations, rewritten as:

$$\frac{\partial \gamma_x u}{\partial x} + \frac{\partial \gamma_x v}{\partial z} = 0$$  \hspace{1cm} (1)

$$\frac{\lambda_v}{\rho} \frac{\partial \theta u}{\partial t} + \lambda_x \theta u \frac{\partial \theta u}{\partial x} + \lambda_z \theta v \frac{\partial \theta v}{\partial z} = -\frac{\gamma_v}{\rho} \frac{\partial \theta p}{\partial x} + \theta \frac{\partial}{\partial x} \left[ \gamma_x 2(v + v_t) \frac{\partial \theta u}{\partial x} \right] + \theta \frac{\partial}{\partial z} \left[ \gamma_x v \left( \frac{\partial \theta u}{\partial z} + \frac{\partial \theta u}{\partial x} \right) \right] - R_x$$  \hspace{1cm} (2)

$$\frac{\lambda_v}{\rho} \frac{\partial \theta v}{\partial t} + \lambda_x \theta u \frac{\partial \theta v}{\partial x} + \lambda_z \theta v \frac{\partial \theta v}{\partial z} = -\frac{\gamma_v}{\rho} \frac{\partial \theta p}{\partial z} + \theta \frac{\partial}{\partial x} \left[ \gamma_x 2(v + v_t) \frac{\partial \theta v}{\partial x} \right] + \theta \frac{\partial}{\partial z} \left[ \gamma_x v \left( \frac{\partial \theta v}{\partial z} + \frac{\partial \theta u}{\partial x} \right) \right] - R_z$$  \hspace{1cm} (3)

where, $u$ and $v$ are the mean velocity components of fluid in the cartesian coordinate directions $x$ and $z$ directions respectively; $t$ is time, gravitational acceleration is denoted by $g$; $\nu$ and $\nu_t$ are respective fluid and eddy viscosities. Fluid density is denoted by $\rho$; $p$ represents the mean pressure and $\theta$ is the parameter of partial cell treatment. For geometric properties, $\gamma_v$ is volume porosity; $\gamma_x$ and $\gamma_z$ are the surface porosities defined as the ratio of porous area to the sectional area of the cell surface. The parameters $\gamma_v$, $\lambda_x$ and $\lambda_z$ are defined from $\gamma_v$, $\lambda$ and $\gamma_x$, respectively as $\lambda = \gamma + (1 - \gamma)C_m$, where $C_m$ is the inertia coefficient. The $R_x$ and $R_z$ are the drag force components in the $x - z$ direction, defined as:
Isoceen 2020

IOP Conf. Series: Earth and Environmental Science 698 (2021) 012030
doi:10.1088/1755-1315/698/1/012030

\[ R_x = \frac{1}{2} C_D \left( 1 - \gamma_x \right) u \sqrt{u_x^2 + v^2} \] (4)

\[ R_z = \frac{1}{2} C_D \left( 1 - \gamma_z \right) u \sqrt{v^2 + u_x^2} \] (5)

where \( C_D \) is the drag coefficient; \( \Delta_x \) and \( \Delta_z \) are the mesh sizes in the corresponding components, respectively. The transport equations of turbulence model are given as:

\[ \frac{\partial \theta_k}{\partial t} + \theta u \frac{\partial \theta_k}{\partial x} + \theta v \frac{\partial \theta_k}{\partial z} = \frac{\partial}{\partial x} \left[ \left( v + \frac{\nu_l}{\sigma_k} \right) \frac{\partial \theta_k}{\partial x} \right] + \frac{\partial}{\partial z} \left[ \left( v + \frac{\nu_l}{\sigma_z} \right) \frac{\partial \theta_k}{\partial z} \right] + C_{\varepsilon 1} (P_r) \frac{\varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon}{k} \] (6)

\[ \frac{\partial \theta_{\varepsilon}}{\partial t} + \theta u \frac{\partial \theta_{\varepsilon}}{\partial x} + \theta v \frac{\partial \theta_{\varepsilon}}{\partial z} = \frac{\partial}{\partial x} \left[ \left( v + \frac{\nu_l}{\sigma_{\varepsilon}} \right) \frac{\partial \theta_{\varepsilon}}{\partial x} \right] + \frac{\partial}{\partial z} \left[ \left( v + \frac{\nu_l}{\sigma_{\varepsilon}} \right) \frac{\partial \theta_{\varepsilon}}{\partial z} \right] + P_r \] (7)

\[ \nu_t = C_d k^2 \varepsilon \] (8)

The advection equations for variable densities of air, \( \rho_a \) and water, \( \rho_w \) as well viscosities of air, \( \nu_a \) and water, \( \nu_w \) can be expressed using the fraction VOF function, as follows:

\[ \rho = (1 - F) \rho_a + F \rho_w \] (11)

\[ \nu = (1 - F) \nu_a + F \nu_w \] (12)

In addition, smoothing of the density and viscosity functions is necessarily employed at the fluid interface to allow a large gradient ratio between water and air using level set function [4, 5]. In particular, this two fluid calculation method enables to describe complex free surface changes on the interface deformation caused from the nonlinear effects such as viscous friction and drag.

2.2. Boundary condition

Figure 1 shows the assigned boundaries for the numerical computational domain. On the inflow boundary, the Dirichlet-type boundary condition is used to generate the wave into the domain. Additionally, the weakly reflecting boundary condition is applied to this and outflow boundary to reduce intermixing to the unphysical reflected waves. The expression for the inflow boundary is given as

\[ \frac{\partial \varphi}{\partial t} - C \frac{\partial \varphi}{\partial x} = \frac{\partial \varphi_i}{\partial t} - C \frac{\partial \varphi_i}{\partial x} \] (13)

while for the outflow boundary,

\[ \frac{\partial \varphi}{\partial t} + C \frac{\partial \varphi}{\partial x} = 0 \] (14)
where $\varphi_i$ is the variables of incident wave signals, $C$ is the wave celerity, and $\varphi$ is the computed spatial variables. Further in the internal obstacle boundary, the non-slip boundary condition for velocities, pressures and $F$ is properly applied to the pontoons as well the attached fishing net. At the bottom, the free-slip rigid wall is taken as the boundary condition, while an open boundary condition is applied for the top. Besides that, the turbulence boundary conditions are assumed with zero vertical fluxes of $k$ and $\varepsilon$ at the free surface.

![Computational domain](image)

**Figure 1.** Computational domain.

### Table 1. TPFB structure particular

| Characteristics                  | Value   |
|----------------------------------|---------|
| Length                           | $l$     | m   | 0.760 |
| Width                            | $W$     | m   | 0.500 |
| Height                           | $D$     | m   | 0.510 |
| Draft                            | $d$     | m   | 0.410 |
| Mass                             | $m$     | kg  | 20.10 |
| Pitch inertia                    | $l_{xx}$| kg\cdot m$^2$ | 0.537 |
| Initial mass center location     | $C_g$   | m   | 0.400 |
| Under keel clearance             | U.K.C   | m   | 0.600 |

2.3. **Numerical scheme**

The total numbers of 302 and 32 cell meshing in the respective $x$ and $z$ directions were used in the computational domain with variable grid size nested towards the structure. The partial differential equations from Equations (1) to (10)) are solved to obtain the solution of the unknown variables of $u$, $v$, $p$, $F$, $k$ and $\varepsilon$. Here, the finite difference method is employed and the time evolution of the unknown variables is advanced based on the Simple Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm. For one computational cycle, the velocity components are first approximated using Equations (2) and (3), where the effects of pressure change is initially neglected. The advection terms of these equations are discretized using a combination of the first order upwind central-differencing scheme, while the diffusion terms are discretized using the second order central-differencing scheme. The pressure field is then updated by adjusting the approximated velocity components iteratively until satisfying the continuity equation, Equation (1), where the Successive Over Relaxation (SOR) method
is used. The calculated new velocity are updated thereafter by taking into account the effects of pressure change. Based on the Equations (6)−(9), the turbulence field components \( k \), \( \varepsilon \) and \( \nu_t \) are then computed, where the same discretization schemes as in the first step is used. For each cell, the new \( F \)-value is updated using the donor–acceptor algorithm, in which the free surface configuration is reconstructed based on the new \( F \)-values. In addition, the boundary conditions are applied in each step. And, the new and next cycles are done by repeating all these steps until the computational time is exceeded.

3. Mechanic of floating breakwater

A series of numerical experiments were carried out in a 2D numerical wave flume model. The flume is 20.0 m length and 2.0 m height, as shown in Figure 1. For the validation purpose, the model scale of 1:20 was adopted for the whole hydrodynamic physical modelling. A floating breakwater is located approximately 10 m between wave maker and outflow boundaries. The structure is 0.50 m width and 0.51 m height with specified permeability. Details on the structural characteristics are presented in Table 1. In addition to solid pontoon boundary, the fishing net obstacle is treated as undeformed plane screen with zero thickness, in which some flow losses will generate across this boundary. A series of parametric studies were conducted to test the effectiveness of the floating breakwater as a wave breaker on the coefficient of transmission, \( K_t \), motion responses, and mooring-line tension, as depicted in Table 2. Meanwhile, the wave height, water depth, and the structural height was kept constant at \( H = 0.20 \) m, \( d = 1.00 \) m, and \( h = 0.50 \) respectively for the whole test conditions.

In the analysis of structural dynamics, a floating algorithm was necessarily established on the basis of the geometry and fluid force on the face of the breakwater structure. The algorithm was solved to approximate the magnitude of surge, heave, pitch, and mooring tension force in each simulation time step. It should be noted here that the treatment for the breakwater motions is handled by fractional average volume obstacle representation (FAVOR) parameter, \( \theta \) in the equation of fluid motions. Here, the values of \( \theta \) modifies periodically the quantities of fluid flow in the continuity and momentum equations due to a complex-moving breakwater boundary inside the computational domain.

### Table 2. Environmental tested conditions.

| Wave height, \( H \) (m) | Water depth, \( d \) (m) | Wave period, \( T \) (s) |
|------------------------|------------------------|------------------------|
| 0.20                   | 1.00                   | 1.00                   |
|                        |                        | 1.10                   |
|                        |                        | 1.20                   |
|                        |                        | 1.30                   |
|                        |                        | 1.40                   |

4. Results and discussion

Figures 2 to 6 show that the results of transmission coefficient, motion response and mooring force are successfully computed in this study. The pertaining discussions with an appropriate reproduction figure of nonlinear phenomena are presented in the following subsections.

4.1. Transmission coefficient

The effectiveness of a breakwater depends largely on it’s induced hydrodynamics effects such as friction to wave motion, reflection and dissipation of incoming wave that ultimately diminishing the attribute of transmission coefficient, defined as the ratio of the transmitted wave amplitude \( (A_t) \) to the incident wave amplitude \( (A_i) \),

\[
K_t = \frac{A_t}{A_i}
\]  

(15)

Following the similar breakwater structure, known as cylindrical floating breakwater, the numerical results were compared to measurements by ji et al., 2015 [6]. Figure 2 shows the changes of \( K_t \) with
various wave periods for $H = 0.20$ m. The results generally show a qualitative and quantitative agreement between numerical and experimental results. A possible reason for this acceptable prediction can be dedicated to the effect of nonlinear effects on the interaction between wave and the floating breakwater. Besides that, it is interesting to note that the CFD model is capable of validating the wave attenuating effect of the floating breakwater in longer waves ($T = 1.4$ s) similar to the experimental counterpart. One of the reasons is because of increased viscous and inertial drag damping due to increasing flow rate of water across the fishing net. In fact, this hydrodynamic consequent is proportional to the drop amount of pressure and thus wave energy. The effect of enhancing this wave energy dissipation hydrodynamically reduces the wave crest and transmission on the leeward side of structure. This investigation reveals that the relative breakwater draft ($h/H > 2.000$) is mostly effective at reducing $K_t$ particularly in high amplitude wave.

![Figure 2. Transmission coefficient](image1)

![Figure 3. Surge motion responses](image2)

![Figure 4. Heave motion responses](image3)
4.2 Motion response

Figures 3, 4 and 5 depict the variable trends of surge, heave and pitch motions of the floating breakwater, respectively. It is shown that the numerical results of surge and heave agree very well with measurements. With increasing wave kinetic energy the magnitudes of surge and heave markedly increase excepting for heaving mode declination beyond $T = 1.3$ s. This resulted trend is also supported previously on the results of $K_t$ in Figure 2. On the other hand, the pitch mode exhibits a slight fluctuation for the whole range of wave period considered. Nevertheless, the results show a consistency between the experimental and numerical data.

It is primarily observed that from Figures 3 to 5, the reduction of $K_t$ beyond $T = 1.3$ s is basically due to the nonlinear phenomena during wave motion across the floating breakwater such as; wave breaking, overtopping, and drag friction on the fishing net boundary, as supported in Figure 7. Note that in the figure capture, the computed characteristic wave force are basically reproduced using fluid pressure, $p$ in the equations of motion. At $t/T = 8.00$ and 8.20, it can be seen that some overtopped wave pressures with white-blue scaled colours are markedly observed on the face of left pontoon as the incident wave approaching the floating breakwater. Up to $t/T = 8.20$, the consequent causes the hydrodynamic pressure to reduce at inner and rear portions of the structure due to the wave dissipation from the wave overtopping and fishing net friction, as indicated by the less intense red-yellow scaled colours. Here, the floating breakwater moves slightly towards negative magnitudes of surge (to the left) and pitch in clockwise direction but with positive upward heave.

Then at $t/T = 8.40$, the pulsating wave again starts to approach the floating breakwater. Simultaneously, the increased forward pressure force suppress the mode motions, where the breakwater is restored approximately at it’s original position. Just after a certain time of wave run up upon the pontoon at $t/T = 8.60$, the structure moves dramatically with maximum positive magnitudes of surge, heave and pitch towards right, upward and anti-clockwise directions, respectively. Furthermore, the pressure force in the vicinity of structure is noticeably higher than $t/T = 8.40$ case. Just after the wave run up phenomenon, the plunging breaker with sharp white-blue coloured contour on the left pontoon appears at $t/T = 8.80$, and simultaneously reducing all the modes of breakwater motion. At the end of wave cycle ($t/T = 9.00$), the pressure distribution at inner and rear portions of the breakwater is significantly diminished due to the augmented wave dissipation effect from the wave breaking and drag forces of fishing net indicated by the mild red-yellow scaled colours. In general, the numerical model can fairly simulate and reproduce many complex-nonlinear phenomena on the hydrodynamics of floating breakwater in waves.
4.3 Mooring Force

The approximation on the changes of mooring-line tension with various wave periods is presented in Figure 6. F1 represents the seaward mooring lines while F2 is the leeward mooring lines. The Figure generally shows that the mooring force increases linearly with increased wave periods. This can be explained from increasing of forward fluid pressure on the face of solid boundary of the floating breakwater as the wave kinetic energy increases. In consequences, the tensile force in the mooring lines increases simultaneously when attempting to hold the structure about it’s original position. Moreover, the observed trend is also supported in the characteristics of surge, heave and pitch motions in Figure 3, 4 and 5, respectively. In comparison to F2, however a sharp increase in tension force can be observed for F1. This is due to the fact that F1 exposes to more severe incoming wave energy.

5. Conclusion

The hydrodynamic properties of a floating breakwater were investigated in this study using computational fluid dynamic (CFD) method. The proposed numerical model is based on the extended Reynolds Average Navier-Stokes (RANS) solver for solid-porous obstacle, and including the fractional average volume obstacle representation (FAVOR) technique as well level set algorithm to treat nonlinearity in the moving boundary and wave fields.

The effects of various wave period parameters were examined on the characteristics of transmission coefficient, surge, heave, pitch, and mooring-line tension. In addition, several nonlinear hydrodynamic phenomena were reproduced and captured in the model by some visualisations on the characteristic of pressure distribution.

Figure 7. Instantaneous position of existing TPFB in a characteristic of pressure pattern for one wave cycle ($T = 1.4$ s).
The capability of the CFD model has been demonstrated, in which the numerical results agree fairly well with measurements. The numerical results show a significant correlation between the environmental tested conditions and the characteristics of $K_t$, as well as the trends of motions and forces on the mooring line. The discrepancies of the results can be partly attributed to the nonlinearity in the incident wave definition.

6. References

[1] Abdullah S F and Fitriadhy A 2020 Application of genetic algorithm for optimum hydrodynamic performance of twin pontoon floating breakwater J. Water. Port Coast. Ocean Eng. 146 2 04019040

[2] Kwang-Leol J and Young-Gill L 2014 Numerical simulations of two dimensional floating breakwaters in regular waves using fixed cartesian grid Int. J. Naval Arch. Ocean Eng. 6 pp 206-218.

[3] Najafi-Jilani A and Rezaie-Mazyak A 2011 Numerical investigation of floating breakwater movement using SPH method Int. J. Naval Arch. Ocean Eng. 3 pp 122-125.

[4] Sussman M, Smereka P and Osher S 1994 A level set approach for computing solutions to incompressible two-phase flow J. Comp. Phys. 114 pp 146-159.

[5] Khenner M, Averbuch A, Israeli M and Nathan M 2001 Numerical simulation of grain-boundary grooving by level set method J. Comp. Phys. 170 pp 764-784.

[6] Ji C-Y, Chen X, Cui J, Yuan Z-M and Incekik A 2015 Experimental study of a new type of floating breakwater Ocean Eng. 105 (Sep) pp 295-303.

Acknowledgements

The authors would like to thank and express their great appreciation to University of Malaysia Terengganu and Sepuluh Nopember Institute of Technology for its support in the completion of this research.