Comparison of impulsive wave forces on a semi-submerged platform deck, with and without columns and considering air compressibility effects, under regular wave actions

Gangjun Zhai\textsuperscript{a}, Ting Zhou\textsuperscript{b}, Zhe Ma\textsuperscript{a}, Nianxin Ren\textsuperscript{c}, Jingjie Chen\textsuperscript{b} and Hee-min Teh\textsuperscript{d}

\textsuperscript{a}Dept. of State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian, People’s Republic of China; \textsuperscript{b}Dept. of School of Naval Architecture, Dalian University of Technology, Dalian, People’s Republic of China; \textsuperscript{c}Dept. of Civil Engineering and Architecture, Hainan University, Haikou, People’s Republic of China; \textsuperscript{d}Dept. of Civil and Environmental Engineering, Universiti Teknologi Petronas, Seri Iskandar, Malaysia

**ABSTRACT**

The decks of oil-extracting platforms could be damaged by great wave impact loads under harsh ocean conditions. Our understanding of the pressure distribution characteristics under the deck, especially those influenced by columns, is incomplete. A series of experiments are carried out to study the spatial and instantaneous distribution of the impact pressure generated by regular waves acting on the deck, and two cases are considered: a flat plate (FP for short) and a plate with four vertical columns (CP for short). Three-dimensional numerical simulations using a modified wave generating tool are conducted and compared to experimental data. Both compressible and incompressible flow solvers are applied to further quantitatively analyse the effect of air compressibility on impacting forces. The results show marked changes in both the pressure distribution scatter under the deck and the magnitude of the impact pressure when comparing the two cases. It has been demonstrated that run-up along the columns can cause intense localized pressure on the deck, and a close relationship exists between this increased pressure and fluid velocity. Additionally, the preliminary results showed that the phase compressibility increases the peak pressure compared with that of an incompressible solver, and the former matches the experimental measurements better.

1. Introduction

Semi-submerged platforms, as a common type of marine structure, are normally constructed in deep water and water of intermediate depth to extract crude oil and alleviate the shortage of terrestrial resources. As shown in Figure 1, the main body of a semi-submerged platform consists of a deck, columns and pontoons. This type of permeable structure has a relatively low construction cost and has been widely used in marine engineering. However, without breakwater protection and when encountering harsh ocean environments, the deck may lose stability and suffer from severe damage caused by large slamming loads (see Figure 2).

Wave slamming is a phenomenon that is characterized by impact loads with large magnitude and short duration, and they can cause great damage to structures both locally and globally (Chella et al., 2012; French, 1979 July; Renzi et al., 2018). Precise estimation and impact load prediction have become critical problems for designers and engineers of marine structures over the years. The original studies on wave slamming focused on the wave entry problem in the fields of shipping and aeronautics. Based on momentum theory and water-added mass assumptions, Von Karman (1929) performed pioneering research on water entering subjects using a wedge-shaped model. Subsequently, research on slamming has been conducted with various types of structures in various fields and can be divided into the following four types: wet deck slamming, sloshing, wave entry and green water (Faltinsen et al., 2004).

As computers have become more powerful, computational fluid dynamics (CFD) has seen rapid growth and, along with physical experiments and theoretical analysis, has become one of the three research methods (Ghalandari et al., 2019; Salih et al., 2019) aimed at studying fluid-structure interactions. However, the wave-in-deck
impact on a floating platform is too complicated to be solved due to the coupling problem between the motion of the platform, the fluid dynamics and the constraint response of the anchor chain. Therefore, researchers have simplified several aspects to study this issue.

The first is the simplification of the structural form. A fixed flat plate or a box is often used to model the structure. There is a large body of work on CFD investigations of wave impact loading on fixed deck-shaped structures, as well as experimental studies. Ren and Wang (2004) numerically investigated wave slamming on marine superstructures in a splash zone with different incident wave forms, air clearance levels and structural sizes. Hsiang (1970) performed foundational work on this subject in which experimental analysis was used to study horizontal wave loads on decks under breaking and nonbreaking wave impacts. Further considering different inclined angles to simulate the minor declining posture when wave impact occurs in actual sea conditions, Ma et al. (2019) provided a fitting formula to predict the wave uplift force on a fixed plate. Ma and Swan (2020) provided an improved physical understanding of wave-in-deck loading over a wide range of incident wave conditions.

In addition, various assumptions have been developed into theoretical methods to investigate the interactions between fixed plates and waves. Based on potential flow theory, which assumes that waves are incompressible and irrotational, Kaplan (1992, may 04) and Kaplan et al. (1995, december 31) obtained an analytical solution to predict the wave slamming loads acting on deck structures of offshore platforms. However, recent studies (Bredmose et al., 2009; Elhimer et al., 2017; Leng & Chanson, 2019; Ma et al., 2016; Sun et al., 2018; Sun et al., 2019a) have suggested that under circumstances of waves with disordered free surfaces, the compressible air pocket existing between the free surface and the structure will have a significant effect on the impact loads. Ma et al. (2016) clarified the importance of carefully handling fluid compressibility in numerical models, which leads to positive and negative gauge pressures and is extremely important for air-enclosed plunging wave impact problems. Sun et al. (2019a) performed several tests and concluded that air entrapment and compressibility greatly influence breaking wave loads. Abdussamie et al. (2014, june 15) studied the vertical loads on the bottom surface of a rigidly mounted box-shaped structure by using Kaplan’s method and the commercial CFD code FLUENT, which is based on the volume of fluid (VOF) method. They concluded that in many situations, the magnitude of forces is seriously underestimated when using Kaplan’s method, while CFD force predictions are more consistent with the experimental forces.

In recent research, the role of columns has been shown to have a significant influence on the impact forces on a deck. In a physical experiment by Scharnke and Henning (2015, october 21), a square column was attached to a fixed box-type deck structure to study the effect of the column. They concluded that the magnitudes of global vertical forces and local pressures greatly increase due to the column. Abdussamie et al. (2017) studied the distribution and magnitude of the impact loads on a fixed platform with four cylinders attached at each corner of the deck. They found that the presence of the columns causes the pressure magnitude to nearly double at the deck underside. However, articles addressing the impact forces on this composite structure with columns based on semi-submerged platforms are still quite limited.

Interestingly, the two-dimensional (2D) column-deck model has been widely utilized in the study of forces on coastal bridges. Seiffert et al. (2014) and Hayatdavoodi et al. (2014) numerically and experimentally studied the forces acting on a coastal-bridge horizontal deck with several vertical girders when a solitary wave propagates. Furthermore, Sarfaraz and Pak (2017) used smoothed
particle hydrodynamics (SPH) to study the effect of numerically derived tsunami wave loads on bridges and developed a simple nondimensional equation for computing the forces on bridge superstructures. The two-dimensional column-deck model in the background of coastal bridges mentioned above is similar to the model of semi-submerged platforms. However, the designed water depths for the two engineering backgrounds are different, and the model postures are completely different, as coastal bridges can be fully submerged, while air clearance exists in a semi-submerged platform. Therefore, the research methods used to study coastal bridges can be used for reference, but the impact loading under the deck and the change laws should be totally different.

Although the impact forces under a deck have been documented, they are not well understood. The questions that need more investigation include the following: (1) how does the pressure distribution react when multiple columns are considered in three dimensions; (2) what factors affect the action of the column on the impact pressure of the deck; and (3) what influence does the air compressibility effect have on impulsive forces? This study aims to fill this knowledge gap by conducting experimental and three-dimensional numerical studies of wave slamming on a fixed platform deck.

Faltinsen et al. (2004) noted that the three-dimensional effect is dominant when wet deck slamming occurs on offshore platforms. However, in the published literature on wet deck slamming, most of the studies were conducted based on two-dimensional experiments or numerical models. More importantly, when using a complex structure such as a plate with columns to study wet deck slamming issues, the three-dimensional effects, including wave diffraction and the scattering effects of columns, should not be neglected; however, they have rarely been studied in detail. At the same time, the existence of columns changes the evolution of the free surface and then influences the entrained gas content, which would affect the impulsive forces on the deck. To date, few systematic studies have discussed the deck-column model or specified the column effect. The connection between the column and the deck is a sensitive and relatively fragile area, but few articles have focused on it.

To carefully study the influence of columns and the air compressibility effect on the distribution of impact forces acting on a deck because of the heavy wave impact, in the present work, two structural models are considered, i.e. a flat plate (FP) and a plate with four vertical columns (CP), which is designed based on mainstream offshore oil production platforms, which have four columns with square sections. A compressible two-phase flow solver and an incompressible flow solver are applied to study the effect of air compressibility on pressure. A series of three-dimensional experiments involving a regular wave impact on a fixed deck platform were carried out. In addition, to establish the impact distribution under the deck and the wave run-up along the columns, a three-dimensional numerical model is utilized to simulate the diffraction phenomenon of the flow field. As mentioned before, phase compressibility may have an effect on the peak pressures according to the papers mentioned above (Bredmose et al., 2009; Elhimer et al., 2017; Leng & Chanson, 2019; Ma et al., 2016; Sun et al., 2018; Sun et al., 2019a). Therefore, a preliminary numerical simulation based on a compressible solver is conducted to initially explore the influence of compressibility on wave-induced forces.

The rest of the work includes the following important aspects. In Section 2, the physical wave tank setup and the wave generation capability are introduced. In Section 3, a 3D numerical model and solution scheme, as well as governing equations, are presented, followed by a verification of the numerical wave tank (NWT). In Section 4, the global uplift forces and local pressure impact are investigated experimentally and numerically. By comparing the results from the FP and CP models, the role of columns is investigated quantitatively. A comparison between compressible and incompressible flow solvers is conducted to study the influence of the air compressibility effect on forces. In Section 5, conclusions are offered to close the paper.

2. Experiment

Physical model experiments are conducted in an experimental wave flume at the Research Institute of Ocean Engineering, Dalian University of Technology. The tank, which is approximately 50.0 m in length, 1.0 m in width, and 1.5 m in depth is filled with tap water to a depth of 0.6 m, as shown in Figure 3. The wave flume is equipped with a hydraulic servo wave generation system, which can simulate regular waves, irregular waves and other common wave spectra. The target waves are generated by a moving wave board at the left end of the tank. At the other end, a wave absorbing device is installed using a rough rock-covered sloped beach.

A 1:100 scale model is mounted centrally in the tank, 25.0 m away from the wave board. The lateral interval is 0.25 m from the sidewalls of the tank to the edge of the plate, and the ratio of the tank width to the submerged model breadth in this paper is not less than 5, thus effectively reducing the influence of the sidewall on the experimental results. The relative freeboard Δh is defined as the distance between the deck bottom surface and the still wave surface and is constantly 0.03 m. The origin of the coordinate system is at the centre
of the wave board and the bottom of the wave flume. The x-axis is positive in the wave propagation direction, the z-axis is positive upwards opposite to gravity, and the y-axis complies with a right-hand system with x-y axes.

The model is made of organic glass, and no elastic deformations are discussed in the subsequent analysis. Due to the large slamming load of waves on the model, the plate-shaped deck model is connected to a supporting structure with independent settings, as shown in Figure 4.

The sampling rate for the wave probe is 100 Hz, while a high sampling frequency of 1000 Hz is chosen for the pressure measurements, and the data are recorded every 0.001 s. The dimensions for two different models are shown in Figure 5. Twenty-five pressure transducers are uniformly installed on the underside of the deck in the FP model and are marked as PT#1 to PT#25; similarly, in the CP model, the serial numbers remain unchanged, and there are 21 pressure transducers, excluding the four corners (PT#1, 5, 21 and 25).

Table 1 shows the wave heights and periods of the test conditions, as well as the prototype. The tested waves are regular and within the range of intermediate wave conditions \(T = 1.4-2.0\) s; the present paper will focus only on those harsh wave conditions that result in wave overtopping, as a constant wave amplitude \(A = 0.05\) m is larger than a relative freeboard \(\Delta h = 0.03\) m, and impulsive wave forces are predictable underneath the deck.
Figure 4. Images of the two models: the FP model is on the left; the CP model is on the right.

Figure 5. Coordinate information and numbering of the pressure transducers under the deck surface (unit: mm).

Table 1. Summary of the tested cases.

| Test | Wave height $H$ (m) | Wave period $T$ (s) | Water depth $d$ (m) | Wave height (m) | Wave period (s) | Water depth (m) |
|------|---------------------|---------------------|---------------------|-----------------|-----------------|-----------------|
| SW1  | 0.10                | 1.8                 | 0.6                 | 10.0            | 18              | 60              |
| SW2  | 0.10                | 1.9                 | 0.6                 | 10.0            | 19              | 60              |
| SW3  | 0.10                | 2.0                 | 0.6                 | 10.0            | 20              | 60              |

3. Model description

3.1. Model description

3.1.1. The incompressible two-phase solver

In this paper, our in-house solver, DUT-FOAM (Wang et al., 2017; Wang et al., 2019), which was developed based on OpenFOAM (Release 4.1), is used for simulating wave-in-deck impacts with regular waves. The basic governing equations for two-phase flows include the following three-dimensional incompressible Reynolds averaged Navier–Stokes (RANS) equations:

$$ \nabla \cdot \mathbf{u} = 0 $$  \hspace{1cm} (1)

$$ \rho \frac{D\mathbf{u}}{Dt} - \nabla \cdot (\mu \nabla \mathbf{u}) = C \kappa \nabla \alpha + g \rho - \nabla p $$  \hspace{1cm} (2)

where $\mathbf{u}$ is the averaged velocity; $g$ is the acceleration of gravity; and $\mu$ and $\rho$ are the averaged dynamic viscosity...
and density of flow, respectively, and are calculated by the following:

\[
\begin{align*}
\rho &= \alpha \rho_1 + (1 - \alpha) \rho_2 \\
\mu &= \alpha \mu_1 + (1 - \alpha) \mu_2
\end{align*}
\]

(3) (4)

\(C \kappa \nabla \alpha\) in Eq. (2) represents the surface tension, \(\alpha\) stands for the fraction of water, \(\kappa\) is the curvature of the air–water interface and \(\kappa = -\nabla \cdot \mathbf{n}\), where \(\mathbf{n}\) is the normal vector of the interface surface element, the direction is from the higher phase fraction to the lower phase fraction, the modulus is 1, and \(C = 0.07\) N/m for pure water at 19.7°C.

Since the numerical model is set based on the laminar assumption in this article, the Reynolds stress term is not considered here, and no turbulence model was used. In the following simulation, only nonbreaking and nearly breaking waves were considered, while no broken waves are considered. In the cases of wave breaking, choosing a proper turbulence model is necessary (Gemmrich & Farmer, 2004); however, in cases such as those involving nonbreaking waves (Cao & Wan, 2017; Sun et al., 2016) and freak waves (Sun et al., 2019b) or even fully nonlinear numerical simulations based on the potential theory for spilling and plunging breakers until the inception of breaking (Khait & Shemer, 2018), reasonably good results can also be obtained without turbulence modeling in CFD simulations.

The PIMPLE algorithm, which is a hybrid of the PISO (pressure-implicit splitting of operators) and SIMPLE (semi-implicit method for pressure-linked equations) algorithms, is applied to solve the coupling problem of velocity and pressure. In addition, the governing equation of the volume fraction \(\alpha\) is solved based on the VOF method (Hirt & Nichols, 1981) aimed at obtaining the interface between water and air. \(\alpha\) is a value varying between 0 and 1, representing the volume fraction of water. The contour line of \(\alpha = 0.5\) is set as a free surface. The equation for \(\alpha\) can be written as follows:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot \mathbf{u} \alpha + \nabla \cdot [\alpha(1 - \alpha) \mathbf{u}_r] = 0
\]

(5)

where \(\mathbf{u}_r\) stands for the relative velocity and the last term in Eq. (5), defined as the artificial compression term, is used to reduce numerical error (Weller, 2008).

3.1.2. The compressible two-phase solver

For our compressible solver, both water and air are treated as ideal fluids and ideal gases. For the water phase, \(\rho_w = \rho_{w0} + p/R_w T\) with \(\rho_{w0} = 1000\) kg/m³ and \(R_w = 3000\) J/kg·K. For the air phase, \(\rho_a = p/R_a T\) with \(R_a = 287\) J/kg·K. The continuity equation needs to be adapted considering the change in density, as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]

(6)

The momentum equation is as follows:

\[
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{uu}) + \nabla \cdot (\mu \nabla \mathbf{u}) = C \kappa \nabla \alpha + g \rho - \nabla p
\]

(7)

The phase equation of the compressible VOF model can be written as follows:

\[
\frac{\partial \alpha_1}{\partial t} + \nabla \cdot (\alpha_1 \mathbf{u}) + \nabla \cdot [\alpha_1(1 - \alpha_1) \mathbf{u}_r]
= \alpha_1 \alpha_2 \text{dgd}t + \alpha_1 \nabla \cdot \mathbf{U}
\]

(8)

where \(\text{dgd}t\) represents the compressibility term, which can be calculated by the following:

\[
\text{dgd}t = \left( \frac{1}{\rho_2} \frac{D \rho_2}{Dt} - \frac{1}{\rho_1} \frac{D \rho_1}{Dt} \right)
\]

(9)

3.1.3. Numerical procedures for the mass source method

The mass source method is adopted for wave generation in this paper, which is explicitly calculated. Taking the incompressible model as an example, the source term is added to the continuity equation, as shown as follows in Eq. (10):

\[
\frac{\partial u_i}{\partial t} = s(x_i, t)
\]

(10)

where \(s(x_i, t)\) is defined as the mass source term and is only nonzero in the source region \(\Omega\). According to Lin and Liu (1999), wave generation is realized by the change in mass, and the source function \(s(x_i, t)\) and the evolution of interface \(\eta(t)\) of the target wave satisfy the following relationship:

\[
\int_0^1 \int s(x, z, t) d\Omega dt = 2 \int_0^1 C \eta(t) dt
\]

(11)

where \(C\) represents the wave’s phase velocity. Furthermore, Eq. (11) can be expressed as follows:

\[
s(x, z, t) = \frac{2 C \eta}{A}
\]

(12)

The simulation of highly nonlinear waves is a longstanding problem, and several existing articles discuss the rules to improve the wave-generating accuracy for the mass source method. Lin and Liu (1999) provided a proper range of source positions and noted that raising the elevation of the mass source could be used to obtain a larger wave height. However, this method only works on weakly nonlinear waves and is ineffective when combined with
highly nonlinear waves. Considering that the simulated wave height is often less than the desired height, Perić and Abdel-Maksoud (2015) proposed a new point of view regarding the cause of the substandard simulated wave height from the perspective of energy. They believed that this is because of insufficient energy utilization during the wave generation process. A coefficient \( K \) is added to Eq. (12) as follows:

\[
s(x, z, t) = \frac{2CK}{A} \eta
\] (13)

where \( K \) is a number greater than 1; thus, a greater wave must be generated. Assuming that the obtained wave height is less than the height of the target wave, \( K \) is the ratio of the target wave height to the obtained wave height. The method works on nonlinear wave generation, but adjustments are needed, and this process is time-consuming.

An originally modified mass source function is proposed to simulate highly nonlinear waves more accurately. According to the wave-generation assumption, each grid satisfies the following relationship in the mass source region:

\[
s(x_i, t)L_sH_s\Delta t = 2u(x_i, t)H_s\Delta t + 2v(x_i, t)L_s\Delta t
\] (14)

where \( L_s \) and \( H_s \) are the length and height of the mass source region, respectively, and \( u \) and \( v \) represent the horizontal and vertical velocities, respectively. Eq. (14) indicates that during a certain period \( \Delta t \), all mass output from the source area is intended for target wave generation. Eq. (14) can be simplified as follows:

\[
s(x_i, t) = \frac{2u(x_i, t)}{L_s} + \frac{2v(x_i, t)}{H_s}
\] (15)

A slow start factor (SSF) is introduced to Eq. (15) to ensure the stability of the numerical simulation; an initial short period \( t_0 \) is defined to ensure that the source term starts from 0. The SSF is calculated as follows:

\[
SSF = \begin{cases} 
0.5 \times \left( 1 - \cos \frac{\pi t}{t_0} \right) & t \leq t_0 \\
1 & t > t_0
\end{cases}
\] (16)

Finally, the mass source function is displayed as follows in Eq. (17):

\[
s^*(x, z, t) = SSF \cdot s(x, z, t)
\] (17)

In Eq. (12), the source term is only associated with the horizontal phase velocity \( C \). By comparison, the modified mass source term in Eq. (15) takes the horizontal and vertical mass output into consideration. In fact, when considering a quadrilateral mass source region, four different directions of mass output exist. Various articles (Chen & Hsiao, 2016; Hafsi et al., 2009a; Hafsi et al., 2009b) have restricted the shape of the mass source region when using the origin mass source method. Here, the new source term Eq. (15) can be used to effectively avoid the problem of shape restriction.

The time step is \( \Delta t = 0.001 \) s initially, and it is auto-adjusted to ensure that the maximum Courant number is smaller than 0.5. Boundary conditions are set in the NWT as follows: (i) on the left, right and bottom boundaries are free-slip with \( n \cdot \nabla p = 0 \); (ii) on the atmospheric boundary \( p = 0 \) and \( (\nabla u) \cdot n = 0 \); (iii) the front and back surfaces are set as symmetric boundary conditions; (iv) the model surface is no-slip and \( n \cdot \nabla p = 0 \).

3.2. Validation of the numerical model

The numerical model is validated in the present study from several aspects, including the interactions between fluid and structures, target wave generation and mesh verification.

First, a three-dimensional benchmark dam-break problem, which was conducted by Raad and Bidoae (2005), is presented to assess the capability of our incompressible model to simulate the horizontal forces of an impacting column. The experimental tank is 0.6 m in width, 1.6 m in length and 0.75 m in height, with a 0.12 m × 0.12 m square-shaped column located in the center of the tank. A water depth of 0.3 m is set up behind a gate, which is raised to start the experiment. On the tank floor, a 0.01 m thin layer of water was set as well. Images of the experimental and numerical wave tank are shown in Figure 6. Note that a real gate is needed in the experiment, while in the numerical model, it is eliminated. The mesh level is chosen to be 0.01 m in three directions to obtain a good comparison, as shown in Figure 7. The water rushes forward because of gravity and impacts the column at approximately \( t = 0.3 \) s. Considering that a short time is needed for the gate to rise, a transient phase difference exists between the numerical results and experimental results when the maximum forces occur, and this phenomenon is common in such cases. Overall, the comparison results are good and indicate that the present model can simulate the interactions between waves and structures.

The classic benchmark of a water column free drop in a closed tank, which involves remarkable gas compression, will be simulated by our compressible solver, and the results will be compared with other results. The initial shape of the closed tank is 20 × 15 m², and a rectangular liquid patch with dimensions of 10 × 8 m² is at rest surrounded by atmosphere at \( p_0 = 1 \) bar. The dimensions of the problem are shown is Figure 8. The four walls of
the tank are all no-slip boundary conditions. Under gravity, the water patch drops and impacts the bottom of the tank. The total simulation time is 1.0 s, with a maximum Courant number of 0.5. The impact pressure was measured at the center of the bottom wall. Meshes of 800 x 600 (each cell is 0.025 m) are proven to obtain good results, as shown in Figure 9. A small amount of air is trapped between the water column and the bottom of the tank at the moment of impact. The compressible solver predicts a higher peak pressure of approximately 50 bar at $t \approx 0.65$ s, while the incompressible solver predicts an earlier and lower pressure peak of approximately 30 bar.

In an NWT, if the wave absorption zone is highly effective, there is no need to model the full length of the physical wave tank to achieve the required level of accuracy and obtain a reasonable computational run time. However, the three-dimensional effect of the structure-wave interaction requires the width of the NWT to be consistent with that of the physical wave tank. An NWT of 31.0 m in length (x-direction), 1.0 m in width (y-direction) and 1.0 m in height (z-direction) is prepared and discretized by the snappyHexMesh preprocessor in DUT-FOAM (Figure 10).

To minimize the adverse effects of wave reflection from the right end wall and re-reflection, 7.6 m long artificial linear damping zones (Shorter, 2004) are located at both ends of the calculating domain. An inserted mass source is located at $x = 8.2$ m to generate waves. The same wave condition SW1 in the experiments is applied in the NWT.

The simulation results are sensitive to the mesh size in the z-direction; therefore, the mesh size is set to remain constant in the x and y directions, and three different mesh resolutions in the z-direction are adopted to test the mesh convergence of the numerical model. Detailed information on the mesh is listed in Table 2.

The numerical results of wave elevations measured at $x = 12.0$ m and the analytical results based on the potential flow are compared in Figure 11. At the lowest resolution in Mesh 1, the numerically predicted wave elevations are evidently larger and less stable than the analytical elevations. As the resolution is refined in Mesh 2 and Mesh 3, the numerical accuracy increases and can generally obtain good wave elevation results compared to the theoretical results. To quantify the numerical result, the relative error (RE) is introduced, as follows:

$$RE(\varphi_i) = \frac{\varphi_i - \varphi_0}{\varphi_0} \times 100\%$$  \hspace{1cm} (14)

![Figure 6. Sketch of the dam break benchmark geometry (dimensions: m).](image)

![Figure 7. Force calculation comparison between the computational results and experimental data.](image)

![Figure 8. Water column free drop benchmark: free surface contour plotted at $t = 0$ s.](image)
1940

G. ZHAI ET AL.

Figure 9. Water column free drop benchmark: time history of the maximum pressure for the present solvers and a comparison with the previous results from Ferrer et al. (2016).

Table 2. Mesh configuration used in the grid study. An Intel(R) Xeon(R) E5-2650 v4 processor (30 M Cache, 2.2 GHz) CPU is used for the grid convergence study.

| Mesh ID | \( \Delta x / m \) | \( \Delta y / m \) | \( \Delta z / m \) | \( N_{\text{cells}} \) |
|--------|-------------------|------------------|------------------|------------------|
| 1      | 0.05              | 0.05             | 0.02             | 620,000          |
| 2      | 0.05              | 0.05             | 0.01             | 1,240,000        |
| 3      | 0.05              | 0.05             | 0.005            | 2,480,000        |

where \( \phi_i \) is the mean wave height calculated by the data within the time range of \([15, 30 \, \text{s}]\), and \( \phi_0 \) is the target value. The relative error calculated using the mean wave period for Mesh 1 is 3.6%, while it is 1.8% and 0.75% for Mesh 2 and Mesh 3, respectively. A slightly increasing coefficient of variation can also be observed in the wave trough; in the present study, the wave though is secondary and will have little effect on the wave-in-deck and the resulting wave impacts.

The mesh verification is proven to be convergent in Figure 11, where Mesh 2 and Mesh 3 can both obtain good wave elevation results compared to the theoretical results. To reduce the computational cost but maintain the highest possible level of mesh refinement, a comprehensive mesh distribution is adopted: Mesh 2 is applied as background mesh for the whole computational region: \( x \in [0.0, 31.0], \ y \in [0.0, 1.0] \); Mesh 3 is applied for free surface region: \( x \in [8.2, 23.4], \ y \in [0.0, 1.0], \ z \in [0.45, 0.7] \); To ensure important physical quantities in the flow field, i.e. wave velocity, are better displayed, the optimal mesh aspect ratio should be as close as possible to 1. The aspect ratio of Mesh 2 is \( \Delta x:\Delta y:\Delta z = 10:10:1 \). Therefore, the meshes near the structure (\( x \in [11.8, 12.7], \ y \in [0.0, 1.0], \ z \in [0.45, 0.7] \)) are refined at 3 levels (which means one grid is evenly divided into 8 small grids) in the \( x \) and \( y \) directions, where the final grid ratio is \( \Delta x:\Delta y:\Delta z = 1.25:1.25:1 \). The comprehensive global and local mesh details are displayed in Figure 12.

We conduct a mesh convergence study for the flow variables (free surface elevation \( \eta \) and pressure \( p \)). The performance of this composite grid setting in wave generation and wave absorption has been validated, as shown in Figure 13. At three different cross-sections, the wave profiles are similar. The effectiveness of the sponge layer at the two ends of the flume is also examined. The wave
Figure 11. (i) Time history of free surface displacement at $x = 12.0$ m for three refined meshes for SW1. (ii) Two curves of the free surface in the wave absorption region, G1 with $x = 27.2$ m and G2 with $x = 31.0$ m.

Figure 12. (a) Sketch of the meshing domain; (b) details of the mesh, where the mesh aspect of $\Delta x: \Delta y: \Delta z = 5:5:1$ in the background, $\Delta x: \Delta y: \Delta z = 10:10:1$ near the free surface, $\Delta x: \Delta y: \Delta z = 1.25:1.25:1$ near the structure; and (c) the three-dimensional view of the CP model.

height of G1 (located at $x = 27.2$ m in the middle of the right absorption area) is stable at first but decreases sharply once waves propagate into the sponge layer. The wave amplitude of G2 (located at $x = 31.0$ m at the end of the sponge layer) is reduced to almost zero, with only approximately 1.29% remaining. The stable long-term history of G1 also suggests that almost no waves reflect on the right wall and interfere with the left flow field. The performances of the meshes at pressures of PT#11 $\sim$ 15 are shown in Figure 14, where an obvious phase difference and amplitude deviation exist in Mesh 1, while convergent results are obtained with Mesh 2, Mesh 3 and the composite mesh. Good performance of this composite grid setting in wave generation and wave absorption.

Figure 13. Time history of free surface displacement at $x = 12.0$ m with different cross-sections in the case of SW1.
was obtained. Thus, the combined mesh is applied to simulate the following interactions between the waves and the structure.

The flat plate is placed in the center of the numerical water tank, and the leading edge is \( x = 12.0 \) m. In the numerical setup, the relative positions of PT#1 \( \sim \) PT#25 to the plate are the same as those, in the experiments, and the detailed coordinate information is listed in Table 3.

### Table 3. Detailed coordinate information regarding the pressure transducers in the numerical wave tank.

| PT# | x (m) | y (m) | PT# | x (m) | y (m) | PT# | x (m) | y (m) |
|-----|-------|-------|-----|-------|-------|-----|-------|-------|
| 1   | 12.05 | 0.7   | 11  | 12.05 | 0.5   | 21  | 12.05 | 0.3   |
| 2   | 12.15 | 0.7   | 12  | 12.15 | 0.5   | 22  | 12.15 | 0.3   |
| 3   | 12.25 | 0.7   | 13  | 12.25 | 0.5   | 23  | 12.25 | 0.3   |
| 4   | 12.35 | 0.7   | 14  | 12.35 | 0.5   | 24  | 12.35 | 0.3   |
| 5   | 12.45 | 0.7   | 15  | 12.45 | 0.5   | 25  | 12.45 | 0.3   |
| 6   | 12.05 | 0.6   | 16  | 12.05 | 0.4   |     |       |       |
| 7   | 12.15 | 0.6   | 17  | 12.15 | 0.4   |     |       |       |
| 8   | 12.25 | 0.6   | 18  | 12.25 | 0.4   |     |       |       |
| 9   | 12.35 | 0.6   | 19  | 12.35 | 0.4   |     |       |       |
| 10  | 12.45 | 0.6   | 20  | 12.45 | 0.4   |     |       |       |

Notes: The vertical coordinates of all pressure transducers are constantly \( z = 0.63 \) m.

The total mesh for the FP model is 2,735,860 and that for the CP model is 2,558,264. For both cases, the simulation of the physical time is 20 s. Taking the CP model as an example, it takes approximately 35 h on a 16-core integrated server with an Intel(R) Xeon(R) E5-2650 v4 processor (30 M Cache, 2.2 GHz) and costs approximately 1.75 h for 1 s on average.

### 4. Results and analysis

In this section, the numerical results and experimental measurements are presented, including the flow field, wave run-up, local pressure and global impact forces.

#### 4.1. Numerical pressure field description of a slamming event

A typical slamming event of the FP model from the front view and bottom-up view is shown in Figures 15 and 16 through a sequence of images. This event begins as the wave approaches the leading edge of the flat plate and finishes with one wave period. The slamming event can be divided into two periods as pressure varies, i.e. wave
Figure 15. Numerical water surface field snapshots consisting of wave upwelling and dropping in the case of SW1 for the FP model (unit: Pa). The snapshots are obtained from the front view at \( y = 0.5 \) m. The relative freeboard is 0.03 m at \( t = 0 \) s.

upwelling (Frames a-f in Figures 15 and 16) and wave exiting (Frames g-k in Figures 15 and 16). At the beginning of the first stage, the wave has not touched the deck, and the pressure equals 0 (\( t = 5.0 \) s). At \( t = 5.14 \) s, the wave first reaches the leading edge of the deck (see Frame b in Figure 16). With the upwelling propagation of the wave, the interaction between the wave and deck becomes violent (Frames c-e in Figure 15), and the pressure distribution along the \( y \)-direction is almost uniform except at the front and back edges, where the boundary effect is small (Frames c-e in Figure 16). At \( t = 5.22 \) s, the highest positive pressure appears, with a magnitude of 0.92 kPa, in a relatively short period, causing damage and instability to the structure.

The second stage begins when the water moves towards the bottom of the deck. After hitting the bottom deck, the wave flow has to divert because its pathway is blocked by the rigid structure, which induces a quick drop in pressure from positive to zero. Afterwards, the wave flow recedes from the platform under gravity, similar to a water-exit process. Sun et al. (2019b) and Faltinsen et al. (2004) both noted that the water-exit process will cause negative pressure. In essence, this may be due to the viscosity of the water, such as the drag force in the water-exit process, which makes the pressure further decrease from zero to a negative value. At \( t = 5.86 \) s, a large, wetted surface occurs (Frame j in Figure 15), and the maximum negative pressure occurs with a magnitude of \(-0.25\) kPa, whose absolute value is comparable to the maximum pressure during the first stage and pulls the structure down (resembles gravity) during the wave slamming process.

Notably, recent studies have proven that other factors could contribute to negative pressure. Yan et al. (2019) found that a position closer to the waterfront underneath the deck will obtain a larger negative pressure. They concluded that this was the reason for the stronger interaction between the wavefront and the bottom surface of the deck, where entrained air would have an effect and cause pressure oscillation. A similar conclusion was obtained by Park et al. (2018), where the negative pressure could be caused by a large jet of splashing water. However, the numerical model here is based on the incompressible N-S equation, and the effect of the air phase’s compression and expansion is not considered in this section. In the experimental results in the following sections, the influence of air cannot be ignored, and the mechanism of negative pressure becomes complicated.

Similar to the FP model, the front-view computational evolution of water surface snapshots and bottom-up view pressure contours for the CP model are presented in Figures 17 and 18, respectively. The slamming event can also be divided into two periods as pressure varies, i.e. wave upwelling (Frames a-f in Figure 18) and wave exiting (Frames g-k in Figure 18). The maximum positive pressure occurs at \( t = 5.20 \) s with a magnitude of 2.92 kPa, and the maximum negative pressure occurs at \( t = 5.84 \) s with a magnitude of \(-0.39\) kPa.

Three-dimensional snapshots of water surface elevation are featured for the CP model compared to the FP model and are plotted in Figure 17. The relative freeboard is 0.03 m, and the incident wave amplitude is 0.05 m; therefore, wave run-up and overtopping can be observed.
Figure 16. A series of numerical pressure distributions during a typical slamming process in the case of SW1 for the FP model (unit: Pa). The snapshots are obtained from the bottom-up view.

Figure 17. Computational evolution of the water surface at different time instances. The wave run-up along the columns and wave overtopping are described in the case of SW1 for the CP model.

In Figure 17 during the wave impact process. In this study, we focus on the impact pressure underneath the deck, so the wave overtopping and the forces on the top surface of the deck are not discussed here.

In the early stage of wave propagation, the wave climbs up along the two fore columns and quickly exceeds the height of the top surface of the deck (Figure 17b-e). During this period, two fore columns are completely submerged, as well as the front half of the deck (under-side). As the wetted surface area increases, the impact area increases and spreads to the backside of the deck (Figure 18c-e).
It is worth noting that at $t = 5.16$ s in Figure 17 (d); the front and rear parts of the deck are submerged and out of the water, respectively. Due to three-dimensional effects, including wave diffraction and the scattering effects of columns, the impact region develops backward. Furthermore, the two bow columns block wave propagation, and waves climb along the rear column and impact the area above that is under the deck (Figure 17e), causing a local stress region at the lateral edge of the deck (Figure 18e). At $t = 5.20$ s, high pressure occurs in the narrow area between two columns, as shown in Figure 18 (f), and the junction between the deck and the columns becomes the area of the structure in danger. The run-up along the columns leads to a nonuniform distribution of the pressure pattern along the $y$-direction and can cause locally increased forces to be applied to the deck.

4.2. Numerical velocity field corresponding to peak pressures

As discussed in Section 4.1, the magnitudes of the positive and negative pressures for the FP model and CP model are different. We hope to demonstrate the results in terms of the velocity of the fluid.

The numerical results of the velocity vector at the time of maximum positive and negative pressures for the FP and CP models discussed in Section 4.2 are shown in Figures 19 and 20, respectively. In particular, Part A at the left column displays the total velocity vector $v$, and Part B in the right column represents the vertical velocity component $v_z$.

In Figure 19 (a) and (c), the maximum velocity of the water under the deck for the FP and CP models at periods when maximum positive pressure occurs is almost the same in magnitude for approximately 0.4 m/s. However, we note that there are differences in the flow fields within the two models. For the CP model, a clockwise vortex is tightly developed close to the rear columns. This is a result of the comprehensive effect of wave run-up along the rear columns, and the three-dimensional effects include wave diffraction and the scattering effects of the columns. Waves with high speed are likely to impact the deck driven by the wave run-up loop and the motion of the vortex, which may lead to a large impact pressure under the deck in the CP model.

The velocity vector for the CP model is more perpendicular to the deck. To further understand the relationship between velocity and pressure by comparing the two models, the component of vertical velocity $v_z$ is extracted...
separately in Figure 19 (B). It is apparent in the comparison of Figure 19 (b) and (d) that the $v_z$ of the CP model is larger than that of the FP model. Figure 21 clearly shows that around the moment when the positive pressure peak occurs (a tiny phase difference in time is allowed), the vertical component velocity $v_z$ values of the FP model at PT#12 ~ 14 are smaller than those of the CP model at different levels, which again verifies the positive relationship between vertical velocity $v_z$ and pressure.

A similar vortex phenomenon is observed in the CP model when maximum negative pressure occurs (Figure 20). Unlike at the time of maximum positive pressure, two tight counterclockwise vortices are found close to the fore and rear columns (Figure 20c). For this case, with a wide range of flow fields, such as vortices, at the time of maximum negative pressure, the characteristics of velocities are difficult to easily reflect in Figure 21 with the three points at the same height. Because of the two scattered vortices, waves with a large downward velocity exist along the region between the front and rear columns, which exactly corresponds to a large area of maximum negative pressure (Figure 18j).

In Figure 20 (a), there is a large velocity region with a magnitude of 0.4 m/s; however, this region exhibits free wave elevation, and there is no contact with the lower deck surface; thus, it does not affect the pressure under the deck. For further analysis, Figure 20 (b) and (d) shows that the $v_z$ of the CP model is slightly larger than that

---

**Figure 19.** Comparison of the numerical results of the velocity vector at the time of maximum positive pressure. A: total velocity ($v$); B: component of vertical velocity ($v_z$) (unit: m/s). The snapshots are obtained from the front view.

**Figure 20.** Comparison of the numerical results of the velocity vector at the time of maximum negative pressure. A: total velocity ($v$); B: component of vertical velocity ($v_z$) (unit: m/s). The snapshots are obtained from the front view.
of the FP model. This is also similar to the consequence mentioned above. Therefore, in general, a large negative velocity leads to a large negative pressure, which concurs with the relationship between positive velocity and positive pressure.

4.3. Numerical local pressure

Valuable insight into wave slamming can be obtained through the evolution of the pressure impact on the deck. The time histories of the corresponding wave-in-deck impact pressures measured at pressure transducers PT#2 and PT#4 from 4.5 s to 10.0 s are presented in Figure 22.

For PT #4, three high-pressure peaks are found at approximately 5.2, 7.0 and 8.8 s, whose peak values are all greater than 2 kPa, at intervals of the wave period ($T = 1.8$ s). A low pressure of approximately $-0.3$ kPa approaches the high pressure because of the viscous drag forces when waves drop away from the bottom surface of the deck. The time history of the impact pressure for PT #2 is similar to that for PT #4, but the pressure amplitude decreases obviously.

The maximum pressures of six specific positions are listed in Table 4. To better compare the influence of the columns on all three pressure points near the columns, the maximum pressure values for PT #2, PT #3 ($x = 12.25$ m), and PT #4 are presented in one figure, as shown in Figure 23.

The maximum pressures of the FP model for PT #2, PT #3 and PT #4 are 0.90, 0.78 and 0.41 kPa, respectively. The maximum pressures of the CP model for the corresponding three points are 1.72, 2.10 and 2.92 kPa. The existence of columns has significantly increased the pressure values of the three measuring points on the edge of the deck, among which PT #2 and PT #3 have increased by 1.91 and 2.69 times, respectively. The PT #4 measuring point is increased by up to 7.12 times that of the original. In addition, when adding the columns into the FP model,
the difference in the maximum pressure between PT #2 and PT #4 increases from 0.49 kPa to 1.20 kPa. Considering that the two measuring points are located at the corner points on the same cross-section, an increase in the overturning moment can take place.

A comparison of computational maximum pressures between compressible and incompressible models is also displayed in Figure 24. In general, the compressible results are similar to the incompressible results regarding the influence of columns. In addition, the compressible results at each position are slightly larger than the incompressible results at different degrees, which is consistent with the results in Figure 29 (which will be discussed later).

4.4. Experimental maximum pressure distribution

For offshore structures, the peak values of pressure pose a threat to structural integrity and personnel security. However, the time history of impact pressure under the deck varies at different positions. To quantitatively analyse the peak value and pressure distribution, a contour map of the 1/2 maximum values is shown in Figure 25. The figure shows that the maximum value of pressure is close to the column from both sides of the deck’s centerline due to the existence of the column.

In Figure 25(a), two high-pressure regions, which are distributed on both sides near the centerline, are obvious in the FP model. In the experiment, although the model is fixed by a steel frame and necessary reinforcement is applied, minor vibrations are inevitable, and the plate cannot remain absolutely horizontal when continuous-wave slamming impacts occur. Therefore, a small offset in the x-axis direction exists between the two high-pressure regions.

For the CP model in Figure 25(b), two high-pressure regions, each with a magnitude of 4.8 kPa, are located at both sides near the edge of the plate. The added columns cause the high-pressure region to transition from the centerline to both sides. Note that there are differences between the magnitudes of the maximum pressures measured in the experiments and those of the numerical pressures in the CP model. The differences may be due to wave breaking and the high randomness of the impact pressure simulation when waves impact the deck in the experiments. Therefore, the results are reasonable and within an acceptable range considering that this phenomenon is common in most of the literature on wave...
slamming. In addition, the consistency of the most dangerous region is obtained by comparing the experimental data and numerical results.

4.5. Experimental global uplift force under the deck

For the experimental and numerical models, the uplift forces under the deck are both integrated by the pressure records on the underside of the deck. Taking the FP model as an example, there are 25 pressure transducers that are evenly distributed under the deck, dividing the plate into 25 parts, with each small square being 0.1 m × 0.1 m in scale. The number of pressure samplings is sufficient to obtain a reasonable result. The total uplift force $F(t)$ is the sum of the 25 components and is calculated as follows:

$$F(t) = \sum_{i=1}^{25} p_i(t) \cdot A$$

where $p_i(t)$ is the measured pressure at every sampling interval for each sensor, $i$ ranges from 1 to 25, and $A = 0.01$ m$^2$ is the area of each smaller square. For the CP model, the calculation method is the same, but $i = 21$.

Figures 26, 27 and 28 show the comparison between the results of integrated impact forces for both models at different wave periods. Note that time phase differences exist between the original data from the experimental measurements and the numerical results, and the moment when the first peak value occurs is artificially adjusted at the same time to allow for better comparison. All curves record upward and downward values, which correspond to the positive and negative pressures measured at each instrument. The present numerical model predicts the positive impact forces in the FP model fairly well, with a slight deviation regarding the negative forces compared to the experimental results (Figures 26 and 27(a)). In the CP model, the comparison of negative peak forces by numerical and experimental methods is quite satisfactory, while for the positive impact forces, the numerical results are smaller than the experimental results. This could be demonstrated by the trapped air effect in the experiments during wave impingement, while it could not be considered in the present incompressible solver. To prove this conjecture, Figure 29 compares the integrated forces for the FP model under the case of SW1 with an incompressible solver against a compressible solver. The average impact forces obtained from the incompressible and compressible solvers are 66.9 and 83.2 N, respectively. The average experimental force is

![Figure 25](image1.png)

**Figure 25.** Experimental distribution of the 1/2 maximum wave impact pressure along the underside of the models in the case of SW1: (a) FP model and (b) CP model (unit: kPa).

![Figure 26](image2.png)

**Figure 26.** Time history of integrated wave impact forces for the two models: numerical results and experimental measurements (SW1).
89.3 N; thus, the relative error for the incompressible results against experiments is 25.0%, while it is 6.8% for the compressible data. The fact that the compressible peak pressures are somewhat larger than those of the incompressible peak pressures can be attributed to the additional compressibility of the air phase, which will lead to the prediction of higher peak pressure (Dias & Ghidaglia, 2018). This clarifies the need to appropriately reproduce such physical phenomena by using compressible numerical models.

In general, the integrated wave impact forces predicted by the numerical model in DUT-FOAM have good agreement with the experimental results for both the FP and CP models. The upward forces of the CP model are generally larger than those of the FP model. However, considering that the slamming impact is highly random, the existence of singularity is reasonable and inevitable. To study this phenomenon, we use the average value of the peak impact forces as the characteristic value of the statistical results. Taking the experimental results when

![Figure 27](image1.png)  
**Figure 27.** Time history of integrated wave impact forces for the two models: numerical results and experimental measurements (SW2).

![Figure 28](image2.png)  
**Figure 28.** Time history of integrated wave impact forces for the two models: numerical results and experimental measurements (SW3).

![Figure 29](image3.png)  
**Figure 29.** Time history of integrated wave impact forces for FP: incompressible results and compressible results (SW1).
$T = 1.8\,\text{s}$ as an example, the average impact forces for the FP model and CP model are 89.3 and 98.7 N, respectively. The existence of columns has a significant effect on the force magnitude, and the growth rate is approximately 10.5%, which is probably a slightly larger value when considering the uplift forces under the bottom surface of columns in the CP model. Therefore, during the process of long-term wave impact on the deck, the stress-increasing effects of columns on the deck could be a concern during the design of the structure.

It should be noted that the negative impact forces in the FP model seem greater than those of the CP model (see Figures 26 and 27 (a)). The possible reason could be that during the water dropping process, the existence of columns increases the damping of falling and indirectly reduces the dragging forces under the deck. However, this process is quite complex, as it is related to wave breaking and trapped air and still needs more research and proof.

5. Conclusions and perspectives

The insights given in this study provide motivation and a foundation for attaching importance to the influences of columns on semi-submerged platforms and for developing a sophisticated model to predict the wave impact load. The wave impact loads due to regular waves on a semi-submerged platform are investigated by experiments and 3D numerical simulations. Based on the statistical analysis of the data and the discussion, the following conclusions are drawn:

- The slamming event can be divided into the following two periods as pressure varies: wave upwelling and wave exiting, corresponding to positive and negative pressures. The wave impact load of the FP model is evenly distributed along the transverse direction, but the symmetrical distribution is destroyed in the CP model due to the blocking effect of columns on the water body. As the wave climbs the columns, the peak value of slamming pressure is concentrated in the narrow region of the front and rear columns and closer to the rear columns. The junction between the deck and the columns is the danger area of the structure, and the run-up along the columns can cause increased local pressure on the deck.
- The existence of columns greatly increases the slamming pressure amplitude under the deck. In the region between the front and rear columns, the growth rate reaches 2–7 times. In addition, pressure differences between PT#12 and PT#14 harm the stability of the structure by causing a large overturning moment.
- The numerical model based on the modified mass source method is demonstrated to be able to simulate target waves. In the analysis of numerical simulation convection field velocity, tight counterclockwise vortices have been found close to the rear columns in the CP model, driving waves with high speed to approach or leave the bottom surface of the deck. Further analysis on the relationship between velocity and pressure is conducted by extracting the vertical velocity component $v_z$. It is proven that a positive relationship exists between the magnitude of $v_z$ and the pressure, where a positive $v_z$ corresponds to positive pressure and vice versa.
- A roughly good agreement is obtained for the integrated impact forces between the numerical and experimental results. By comparing the CP and FP models, the presence of columns increases the overall impact forces under the deck as a consequence of the increased pressure. There is a large increase of 10.5% calculated from the experimental results for SW1 in the integrated impact forces for the CP model.
- that the findings show that the incompressible 3D numerical model underestimates the impact forces on the deck compared to the experimental results. A relatively good match was obtained with the compressible model. The results suggest that the compression and expansion of trapped air cavities or bubbles have an effect on pressure forces, which occur in reality but cannot be captured by an incompressible model.

To systematically explore the compressibility effect in this particular wave slamming scenario, research based on various wave conditions will be conducted in future studies, especially in irregular waves and shallow water conditions. In addition, the possible relationship in the case of the CP model should also be further verified.

Disclosure statement

No potential conflict of interest was reported by the author(s).

Funding

The authors want to acknowledge the financial support from the Natural Science Foundation of Liaoning Province, China (Grant No. 2021-MS-109).

References

Abdussamie, N., Amin, W., Ojeda, R., & Thomas, G. (2014, June 15). Vertical wave-in-deck loading and pressure distribution on fixed horizontal decks of offshore platforms. Proceedings of the 24th International Ocean and Polar Engineering conference, Busan, Korea.

Abdussamie, N., Ojeda, R., Drobyshhevski, Y., Thomas, G., & Amin, W. (2017). The impact of extreme wave events

on a fixed multicolumn offshore platform. *International Journal of Offshore and Polar Engineering*, 27(3), 293–300. https://doi.org/10.17736/ijoipe.2017.sh18

Bredmose, H., Peregrine, D. H., & Bullock, G. N. (2009). Violent breaking wave impacts. Part 2: Modelling the effect of air. *Journal of Fluid Mechanics*, 641, 389–430. https://doi.org/10.1017/S0022112009999157

Cao, H., & Wan, D. (2017). Benchmark computations of wave run-up on single cylinder and four cylinders by nse-FOAM-SJTU solver. *Applied Ocean Research*, 65, 327–337. https://doi.org/10.1016/j.apor.2016.10.011

Chella, M. A., Torum, A., & Myrhaug, D. (2012). An overview of wave impact forces on offshore wind turbine substructures, in: Tranell, G. (Ed.). *Energy Procedia*, Tranell, G. ed, 217–226. https://doi.org/10.1016/j.egypro.2012.03.022

Chen, Y. L., & Hsiao, S. C. (2016). Generation of 3D water waves using mass source wavemaker applied to navier–Stokes model. *Coastal Engineering*, 109, 76–95. https://doi.org/10.1016/j.coastaleng.2014.11.011

Dias, F., & Ghidaglia, J.-M. (2018). Slamming: Recent progress in the evaluation of impact pressures. *Annual Review of Fluid Mechanics*, 50(1), 243–273. https://doi.org/10.1146/annurev-fluid-010816-060121

Elhimer, M., Jacques, N., Alaoui, A. E. M., & Gabillet, C. (2017). The influence of aeration and compressibility on slamming loads during cone water entry. *Journal of Fluids and Structures*, 70, 24–46. https://doi.org/10.1016/j.jfluidstructs.2016.12.012

Faltinsen, O. M., Landrini, M., & Greco, M. (2004). Slamming in marine applications. *Journal of Engineering Mathematics*, 48(3–4), 187–217. https://doi.org/10.1023/B:BENG.0000018188.683804e

Ferrer, P. J. M., Causon, D. M., Qian, L., Mingham, C. G., & Ma, Z. H. (2016). A multi-region coupling scheme for compressible and incompressible flow solvers for two-phase flow in a numerical wave tank. *Computers & Fluids*, 125, 116–129. https://doi.org/10.1016/j.compfluid.2015.11.005

French, J. A. (1979 July). Wave uplift pressure on horizontal platforms. Proceedings of Civil Engineering in the oceans, ASCE, San Francisco, CA, 187–202.

Gemrrich, J. R., & Farmer, D. M. (2004). Near-surface turbulence in the presence of breaking waves. *Journal of Physical Oceanography*, 34(5), 1067–1086. https://doi.org/10.1175/1520-0485(2004)034<1067:NTTPO>2.0.CO;2

Ghalandari, M., Bornass, S., Shamshirband, S., Mosavi, A., & Chau, K. W. (2019). Investigation of submerged structures' flexibility on sloshing frequency using a boundary element method and finite element analysis. *Engineering Applications of Computational Fluid Mechanics*, 13(1), 519–528. https://doi.org/10.1080/19942060.2019.1619197

Hafisa, Z., Ben Haj, M., Lamlouni, H., & Maalel, K. (2009a). Comparison between moving paddle and mass source methods for solitary wave generation and propagation over a steep sloping beach. *Engineering Applications of Computational Fluid Mechanics*, 3(3), 355–368. https://doi.org/10.1080/19942060.2009.11015276

Hafisa, Z., Hadji, M. B., Lamlouni, H., & Maalel, K. (2009b). Internal inlet for wave generation and absorption treatment. *Coastal Engineering*, 56(9), 951–959. https://doi.org/10.1016/j.coastaleng.2009.05.001

Hayatdavoodi, M., Seiffert, B., & Ertekin, R. C. (2014). Experiments and computations of solitary-wave forces on a coastal-bridge deck. Part II: Deck with girders. *Coastal Engineering*, 88, 210–228. https://doi.org/10.1016/j.coastaleng.2014.02.007

Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39(1), 201–225. https://doi.org/10.1016/0021-9991(81)90145-5

Hsiang, W. (1970). Water wave pressure on horizontal plate. *Journal of the Hydraulics Division*, 96(10), 1997–2017. https://doi.org/10.1061/(JUCCEA)0002724

Kaplan, P. (1992, May 04). Wave impact forces on offshore structures: Re-examination and New interpretations. Proc offshore technology conference, Houston, Texas, USA.

Kaplan, P., Murray, J. J., & Yu, W. C. (1995, December 31). Theoretical analysis of wave impact forces on platform deck structures. Proc offshore technology conference, Houston, Texas, USA.

Khait, A., & Shemer, L. (2018). On the kinematic criterion for the inception of breaking in surface gravity waves: Fully nonlinear numerical simulations and experimental verification. *Physics of Fluids*, 30(5), 0571035. https://doi.org/10.1063/1.5026394

Leng, X., & Chanson, H. (2019). Air-water interaction and characteristics in breaking bores. *International Journal of Multiphase Flow*, 120, 103101. https://doi.org/10.1016/j.ijmultiphaseflow.2019.103101

Lin, P. Z., & Liu, P. L. F. (1999). Internal wave-maker for navier-Stokes equation models. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 125(4), 207–215. https://doi.org/10.1061/(ASCE)0733-950X(1999)125:4(207)

Ma, L., & Swan, C. (2020). An experimental study of wave-in-deck loading and its dependence on the properties of the incident waves. *Journal of Fluids and Structures*, 92, 102784. https://doi.org/10.1016/j.jfluidstructs.2019.102784

Ma, Z., Zhou, T., Ren, N., & Zhai, G. (2019). A comprehensive study of the wave impact loads on an inclined plate. *Journal of Marine Science and Engineering*, 7(4), 103. https://doi.org/10.3390/jmse7040103

Ma, Z. H., Causon, D. M., Qian, L., Mingham, C. G., & Martinez Ferrer, P. (2016). Numerical investigation of air enclosed wave impacts in a depressurised tank. *Ocean Engineering*, 123, 15–27. https://doi.org/10.1016/j.oceaneng.2016.06.044

Park, H., Do, T., Tomiczek, T., Cox, D. T., & van de Lindt, J. W. (2018). Numerical modeling of non-breaking, impulsive breaking, and broken wave interaction with elevated coastal structures: Laboratory validation and inter-model comparisons. *Ocean Engineering*, 158, 78–98. https://doi.org/10.1016/j.oceaneng.2018.03.088

Perić, R., & Abdel-Maksoud, M. (2015). Generation of free-surface waves by localized source terms in the continuity equation. *Ocean Engineering*, 109, 567–579. https://doi.org/10.1016/j.oceaneng.2015.08.030

Raad, P. E., & Bidoae, R. (2005). The three-dimensional eulerian-lagrangian marker and micro cell method for the simulation of free surface flows. *Journal of Computational Physics*, 203(2), 668–699. https://doi.org/10.1016/j.jcp.2004.09.013

Ren, B., & Wang, Y. (2004). Numerical simulation of random wave slamming on structures in the splash zone. *Ocean Engineering*, 31(5–6), 547–560. https://doi.org/10.1016/j.oceaneng.2003.10.006
Renzi, E., Wei, Y., & Dias, F. (2018). The pressure impulse of wave slamming on an oscillating wave energy converter. *Journal of Fluids and Structures, 82*, 258–271. https://doi.org/10.1016/j.jfluidstructs.2018.07.007

Salih, S. Q., Aldlemy, M. S., Rasani, M. R., Ariffin, A. K., Ya, T. M. Y. S., Al-Ansari, N., Yaseen, Z. M., & Chau, K. (2019). Thin and sharp edges bodies-fluid interaction simulation using cut-cell immersed boundary method. *Engineering Applications of Computational Fluid Mechanics, 13*(1), 860–877. https://doi.org/10.1080/19942060.2019.1652209

Sarfraz, M., & Pak, A. (2017). SPH numerical simulation of tsunami wave forces impinged on bridge superstructures. *Coastal Engineering, 121*, 145–157. https://doi.org/10.1016/j.coastaleng.2016.12.005

Scharnke, J., & Hennig, J. (2015, October 21). Vertical wave impact loading on a fixed platform deck. Proceedings of the ASME 2015 34th International Conference on ocean, offshore and Arctic engineering, St. John’s Newfoundland, Canada.

Seiffert, B., Hayatdavoodi, M., & Ertekin, R. C. (2014). Experiments and computations of solitary-waves forces on a coastal-bridge deck. Part I: Flat plate. *Coastal Engineering, 88*, 194–209. https://doi.org/10.1016/j.coastaleng.2014.01.005

Shorter, P. J. (2004). Wave propagation and damping in linear viscoelastic laminates. *Journal of the Acoustical Society of America, 115*(51), 1917–1925. https://doi.org/10.1121/1.1689342

Sun, H., Sun, Z., Liang, S., & Zhao, X. (2019a). Numerical study of air compressibility effects in breaking wave impacts using a CIP-based model. *Ocean Engineering, 174*, 159–168. https://doi.org/10.1016/j.oceaneng.2019.01.050

Sun, L., Zang, J., Chen, L., Eatock Taylor, R., & Taylor, P. H. (2016). Regular waves onto a truncated circular column: A comparison of experiments and simulations. *Applied Ocean Research, 59*, 650–662. https://doi.org/10.1016/j.apor.2016.03.011

Sun, P., Luo, M., Le Touzé, D., & Zhang, A. (2019b). The suction effect during freak wave slamming on a fixed platform deck: Smoothed particle hydrodynamics simulation and experimental study. *Physics of Fluids, 31*(11), 117108. https://doi.org/10.1063/1.5124613

Sun, S., Wu, G., & Xu, G. (2018). Breaking wave impact on a floating body with air bubble effect. *Journal of Fluids and Structures, 82*, 16–34. https://doi.org/10.1016/j.jfluidstructs.2018.06.016

Von Karman, T. (1929). The impact on seaplane floats during landing. Technical Report Archive & Image Library. 321.

Wang, D., Sun, J., Gui, J., Ma, Z., & Ning, D. (2017). A numerical piston-type wave-maker toolbox for the open-source library OpenFOAM. *Journal of Hydrodynamics, 5*(29), 774–781.

Wang, D., Sun, J., Gui, J., Ma, Z., Ning, D., & Fang, K. (2019). A numerical piston-type wave-maker toolbox for the open-source library OpenFOAM. *Journal of Hydrodynamics, 31*(4), 800–813. https://doi.org/10.1007/s42241-018-0116-4

Weller, H. G. (2008). A new approach to VOF-based interface capturing methods for incompressible and compressible flow. OpenCFD Ltd Report TR/HGW/04.

Yan, B., Luo, M., & Bai, W. (2019). An experimental and numerical study of plunging wave impact on a box-shape structure. *Marine Structures, 66*, 272–287. doi:10.1016/j.marstruc.2019.05.003