NUMERICAL SIMULATION OF OBLIQUE WAVE IN WATER BASIN

Yu Chen, Yali Zhang, Mimi Gao, Johan Gullman-Strand
Lloyd's Register GTC
1 Fusionopolis Way
Connexis (North Tower) #17-14
Singapore 138632

Rajeev Kumar Jaiman
Department of Mechanical Engineering
National University of Singapore
Singapore 117575

MyHa Dao, Lup Wai Chew
Institute of High Performance Computing
A*STAR, Singapore
1 Fusionopolis Way
Connexis (North Tower) #16-16
Singapore 138632

ABSTRACT
Simulation of more realistic ocean conditions in wave basins is becoming important for offshore industry. As spreading wave has become more desirable, the capability of reproducing oblique planar wave train is critical for a wave basin’s performance. In this study, the oblique waves have been simulated in a numerical wave basin based on the volume of fluid (VOF) method and Navier-Stokes solver. Multi-element bottom hinged flap motion using the conventional snake principle has been simulated by moving boundary dynamic mesh in OpenFOAM. Finite length of the entire wave maker and finite width of each paddle caused considerable spatial variations in wave height and wave propagating direction. Beaches with slope have been implemented to minimize the sidewall reflection and improve the uniformity of the oblique wave field. The generated linear oblique waves have been compared with the analytical solutions for validation in terms of uniformity and wave height. The time history of the surface elevation at different locations have been computed to investigate the uniformities of general wave fields. In addition, the free surface along the wave propagating direction has also been investigated to show the quality of the generated wave.

INTRODUCTION
In the past three decades, the offshore oil exploitation of natural resources and the demand for offshore platforms have continued to increase. However, the ocean environment can pose many dangers to offshore structures, risk assessment process is required for these offshore working units in order to ensure the safety under extreme wind and wave conditions. Presently within the offshore industry, analytical methods and scaled model tests by laboratory experiments are the main methods for establishing wave-in-deck loads [1–3]. Generally, the analytical methods are based on simplified fluid dynamics theory and although they are easy to implement they have low reliability and accuracy. Thus, experimental tests in small wave basins are quite common in the field of marine engineering. These basins have currents, wind and waves generators to try to simulate the marine environment. However, testing in basins also has its disadvantages, e.g. it is relatively expensive and scaling effects may influence the accu-
racy especially when there is air entrainment [4–6]. As an alternative approach, numerical simulation based on computational fluid dynamics (CFD) has attracted a lot of attention recently due to advances in numerical methods and the availability of high performance computers. CFD can provide a useful tool to predict impact forces with high accuracy for the full scale deck, as well as the loading produced by breaking waves. Detailed insight into the flow and resultant loadings can also be resolved by CFD which is not possible with analytical methods and very difficult with experimental methods. CFD simulations can also be treated as a set of preliminary tests of experiments in wave basin since configuration and design of platforms can be easily changed inside, and the wave basin experiments can be used as a final verification.

Recently, CFD has been widely employed to investigate environmental loads for offshore structures, i.e. wave-in-deck problems [7–9]. A new CFD tool, ComFLOW, based on local grid refinement, VOF and a blended turbulence model has been utilized by [10] to study extreme wave impact in offshore and coastal engineering. However, most of these works are restricted to unidirectional regular waves, e.g. 5th order Stokes wave or other similar kind of waves, which are significantly different from the directionally spreading waves in real ocean conditions. O’dea and Newman [11] conducted a numerical study of directionally-spread waves in WAMIT [12]. However, WAMIT is completely based on the boundary element method and potential flow theory, so the effect of viscosity and air entrapment cannot be. Thus, it is important to generate the spreading waves in the numerical wave basin based with CFD based on Navier-Stokes equations. As known, spreading waves are composed of group of wave components which travels in different directions, thus generating single oblique wave precisely is essential for generating spreading wave. Actually generating oblique waves to adjust angle between current and wave propagation direction has been widely implemented in some physical wave basins in the world, so validating the oblique wave numerically generated can be seen as a preliminary stage of numerical spreading wave and also can guide the design and operation of experimental wave basins.

Basically, the wave generator consists of the actuator and the power system to move it. The key of successfully generating a correct oblique wave in a basin is the transfer function between wave height and flap stroke and proper delay among the flaps. Thus it requires a mathematical formulation for the water waves propagation and numerical models simulating multiphase problems with free surface, which determine the aim of the current paper, e.g. validating the capability of numerical models and transfer functions in generating oblique waves, in terms of wave height, oblique angle and uniformity in the test region. So, the theory of transfer functions are briefly introduced firstly in Section “Mathematical Model”, then followed by the numerical methodology implemented (i.e. Volume of Fluid, VOF). The configuration of the numerical wave basin and the results are then presented and discussed.

**MATHEMATICAL MODEL**

In this section, the mathematical connection between the desired waves and the motion of wave maker (e.g. transfer function) is briefly introduced. The transfer function is normally determined based on the potential flow theory for water waves, and has been studied in the last century [13–15], which is introduced firstly. A detailed account of oblique wave generation has been presented by Frigaard et al. [16].

**Velocity Potential**

“Les Appareils Générateurs de Houle en Laboratoire” [13] presented by Biésel and Suquet discussed and solved the analytical problems concerning a number of different wave generator types. For each wave maker type the paper presented the transfer function between wave maker displacement and wave amplitude in these cases where the analytical problem could be solved. The article therefore represented a giant step in wave generation techniques and found the basis for today’s wave generation in hydraulics laboratories.

The simplest velocity potential is that obtained assuming small amplitude waves or very small wave slope since linear theory can be applied. Under such assumptions the so-called velocity potential (\(\phi\)) should be determined because it allows the derivation of all the desired wave characteristics. Once the velocity potential is known the transfer function and hydrodynamic reaction can be calculated. The configuration of the wave basin and properties of waves are presented in Figure 1.

**FIGURE 1.** SCHEME OF WAVE MAKER, ADOPTED FROM [17].

The displacement of paddles is described in [16]

\[
x = e(z) \sin(\omega t) = \frac{S(z)}{2} \sin(\omega t).
\] (1)
where \( e(z) \) is the maximum displacement of different parts of paddles, and \( S(z) \) is distance between two extreme position of different part of paddles (see Figure 1 and Eq. 2). For bottom hinged (flap) type wave maker, the movement of the paddle can be expressed as

\[
S(z) = S_0 \frac{h+z}{h}
\]

(2)

and the transfer function is

\[
H = \frac{2\sinh(kh)(1 - \cosh(kh) + kh \sinh(kh))}{kh(\sinh(kh) \cosh(kh) + kh)}
\]

(3)

where \( k \) is the wave number of the generated sinusoidal wave, and it is the solution to the dispersion relation:

\[
\omega^2 = kg \tanh(kh),
\]

(4)

and \( H \) is the height of wave generated.

**Oblique Wave**

In this section, techniques to generate oblique waves will be discussed. Considering a wave generating system where the generator front consists of a number of very small paddles. An oblique regular wave can then be generated using Huygen’s principle [16], by introducing a suitable delay between the wave paddles as illustrated in Figure 2. It is evident that the required delay of the individual wave paddles will lead to a sinusoidal shape of the front of the wave generator.

**Phase Delay in Neighbouring Paddle**

If the wave length of the generated wave is \( \lambda \), and the wave length from maximum to maximum of the sinusoidal front of the wave generator is \( l = \lambda / \sin \theta \). Thus the delay, \( \varphi_p \), between neighbouring wave paddles of width \( l_p \) is \( l_p \cdot 2\pi / \lambda \) or

\[
\varphi_p = l_p \frac{2\pi \sin \theta}{\lambda}
\]

(5)

If the wave paddle displacement is calculated for a regular wave with the wave length \( \lambda \) travelling in the x-axis direction, the delay between the ith and the 0th wave paddle when generating the same regular wave travelling in the \( \theta \)-direction, is

\[
\varphi_{pi} = i \cdot l_p \frac{2\pi \sin \theta}{\lambda}
\]

(6)

**Transfer Function for Oblique Waves**

Considering the energy propagation, the transfer function of oblique wave should be corrected as

\[
F_3 = F_2 / \cos \theta
\]

(7)

where \( F_2 \) is the transfer functions defined in equation 3.

**NUMERICAL MODEL**

The open-source CFD code OpenFOAM, which employs a nonlinear Navier-Stokes equation solver, has been used to conduct the numerical simulation of the wave basin. Specifically, the volume of fluid (VOF) method was employed to simulate the water-air interface. This includes the viscous dissipation during wave breaking and slamming which cannot be simulated by theoretical methods or potential flow based methods.

“interFoam”, one of the multi-phase solvers in OpenFoam, has been used in the current numerical study. It solves the 3D N-S equations for two incompressible phases using a finite volume discretization and the volume of fluid (VOF) method.

In a two-phase flow simulation with fluid densities \( \rho_l \) and \( \rho_g \), viscosities \( \mu_l \) and \( \mu_g \) and surface tension coefficient \( \sigma \), the flow is governed by the following momentum equations [18]:

\[
\frac{\partial \rho U}{\partial t} + \nabla \cdot (\rho U \otimes U) = - \nabla p + \left[ \nabla \cdot (\mu \nabla U) + \nabla U \cdot \nabla \mu \right] + \rho g + \int_{\Gamma} \sigma k \delta(x - x_s) n d\Gamma(x_s)
\]

(8)

where \( \Gamma \) is the air-water interface, \( \delta(x - x_s) \) is the three-dimensional (3D) Dirac delta function and the viscous term \( \nabla \cdot \left[ \mu (\nabla U + \nabla U^T) \right] \) has been rewritten as \( \nabla \cdot (\mu \nabla U) + \nabla U \cdot \nabla \mu \). In interFoam, the continuum surface force (CSF) model of Brackbill et al. [19] has been employed. Readers are referred to [19]
for detailed discussion of the integration of surface tension term and approximation of the local interfacial curvature $\kappa$.

The solver employs a modified pressure ($p_d$) rather than solving for total pressure $p$ and their relationship is given by

$$p_d = p - \rho g \cdot x \nabla p_d = \nabla p - \rho g \cdot x \nabla p$$  \hspace{1cm} (9)

with both the surface tension term and the substitution of $p_d$ for pressure, the volume integral of equation (8) over an arbitrary cell is solved in interFoam by constructing a predicted velocity field and then correcting it using the Pressure Implicit with Splitting of Operators (PISO) [20] implicit pressure correction procedure to time advance the pressure ($p_d$) and velocity fields. If the cell in question is identified by subscript $P$, and the PISO iteration procedure is indexed by $m$, with $m = 0$ corresponding to the initial step and pertaining to the present time level $t^n$, then, we may consider the discrete version of equations (8) neglecting for the moment pressure term, which yields an explicit expression for the predicted velocity field $U_f'$, namely

$$\frac{\rho_p^{n+1} U_f' - (\rho U)_f^n}{\Delta t} [\Omega_f] + \sum_{f \in \partial \Omega_i} \left( r_f \Phi_f \right) U_f'$$

$$= \sum_{f \in \partial \Omega_i} \left[ \mu_f^{n+1} \left( \nabla_f U' \right)' S_f \right] + \nabla U_p' \cdot \nabla \mu_p^{n+1} [\Omega_f] \hspace{1cm} (10)$$

where the variables with subscription $f$ are interpolated values at cell faces. There is wide range of interpolation schemes with or without limiters implemented in OpenFOAM for this purpose (centred, upwind, TVD/NVD). The associated volume flux $\Phi_f = (U_p')_f \cdot S_f \cdot |S_f|$ and $|\Omega_f|$ are magnitude of face surface area and cell volume respectively. In this expression, the fluid density $\rho$ and viscosity $\mu$ are obtained by

$$\rho = \gamma \rho_l + (1 - \gamma) \rho_g, \hspace{0.5cm} \mu = \gamma \mu_l + (1 - \gamma) \mu_g \hspace{1cm} (11)$$

and volume fraction function $\gamma$ is solved by a separate equation and is known at time level $t^n$ and $t^{n+1}$ when solving momentum equations, the density and viscosity fields in equation (11) are also known. By employing the techniques described in [21] to approximate the velocity face value $U_f'$ and face normal gradient $\left( \nabla_f U \right)'$, equation (10) can be re-arranged to a linear system

$$A \cdot x = B \hspace{1cm} (12)$$

where $A$ is a large sparse matrix, $x$ the dependent variables vector and $B$ the constant source vector. Equation (12) can be solved using the preconditioned conjugate gradient (PCG) method. Besides PCG, OpenFOAM provides various other options [22] such as preconditioned bi-conjugate gradient, generalized geometric algebraic multi-grid and smooth Solver, which uses a smoother for convergence. Equations (10–12) completes the velocity predictor step, and now the pressure equation can be formulated by including the pressure contribution (which is not accounted in deriving Equation (11)) and enforcing the continuity (mass conservation) for an incompressible medium. This results in

$$\sum_{f \in \partial \Omega_i} \left[ \frac{1}{A_P} \left( \nabla_f U_f' \cdot p_d^{m+1} \right) S_f \right] = \sum_{f \in \partial \Omega} \Phi_f \hspace{1cm} (13)$$

where quantity $A_P$ is the computed coefficient before unknown $U_f'$ after rearranging Equation 10. Equation (13) again leads to a linear system for $p_d^{m+1}$, and can be solved using one of the techniques introduced above. Equation (13) completes the brief introduction to the solution of the incompressible Navier-Stokes system.

The mathematical formulations of the two phase system with free surface are closed by solving a separate convection equation for the evolution of volume fractions ($\gamma$). The present solver, interFoam, employs a modified approach similar to one proposed in [23], named High Resolution Surface Capturing Scheme (HRIC), relying on a two-fluid formulation of the conventional volume-of-fluid (VOF) model in the framework of finite volume method. In this model an additional convective term originating from modeling the velocity in terms of weighted average of the corresponding liquid and gas velocities is introduced into the transport equation for phase fraction, providing a sharper interface resolution. The model makes use of the two-fluid Eulerian model for two-phase flow, where phase fraction equations are solved separately for each individual phase. For details readers are referred to [21]. This completes the mathematical formulations for a two phase incompressible system and what remain are the formulations for focused extreme wave generator.

**RESULTS AND DISCUSSIONS**

The linear wave investigated in the current study was a small amplitude regular wave, whose properties are listed in Table 1. It can be described by the 1st order Stokes wave theory. There are a lot of different types of actuators but the flap is one of the most common types in the physical basins, so it was chosen herein. For simplicity but without any loss of generality, a small numerical wave basin was chosen in order to reduce the computational cost, since the aim of the study was to validate the capability of the numerical wave basin generating oblique waves. The oblique wave is expected to travel in the 60 degree direction to the x-axis as shown in Figure 4(a). Two side walls of the wave basin are installed with the moving paddles, and the other two are wave
TABLE 1. PARAMETERS OF WAVE IMPLEMENTED AND WAVE BASIN DIMENSIONS

<table>
<thead>
<tr>
<th>Wave properties</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water depth d (m)</td>
<td>0.7</td>
</tr>
<tr>
<td>Wave height H (m)</td>
<td>0.08</td>
</tr>
<tr>
<td>Wave period T (sec)</td>
<td>1.303</td>
</tr>
<tr>
<td>Wave crest elevation E (m)</td>
<td>0.04, 0.06</td>
</tr>
<tr>
<td>Wave length λ (m)</td>
<td>2.5</td>
</tr>
<tr>
<td>Basin dimension L×W×D (m)</td>
<td>12×12×0.85</td>
</tr>
<tr>
<td>Test section L×W (m)</td>
<td>8×8</td>
</tr>
<tr>
<td>Paddle width l_p (m)</td>
<td>0.5</td>
</tr>
<tr>
<td>Paddle range (m)</td>
<td>7×7</td>
</tr>
<tr>
<td>Beach slope</td>
<td>1:5</td>
</tr>
</tbody>
</table>

The mesh employed for the simulation are illustrated in Figure 3, which is refined near the interface between water and air.

FIGURE 3. MESH FOR THE SIMULATION OF WAVE BASIN.

Four wave gauges were defined inside the wave basin (see Table 2) to monitor the wave elevation and investigate the oblique angle and uniformity of the generated waves. These four wave gauges were arranged in a line orthogonal to the oblique wave’s propagation direction (also see Figure 4(a)).

Figure 4 presents the visualization of the wave basin together with the generated oblique wave’s free surface. It can be found that qualitatively the generated wave is travelling in the expected direction within the test region and then is absorbed in the beach absorbing beaches as shown in Figure 4(b). More detailed dimensions of the wave basin, paddles and beaches can be found in Table 1. The mesh employed for the simulation are illustrated in Figure 3, which is refined near the interface between water and air.

TABLE 2. WAVE GAUGES’ LOCATIONS INSIDE THE NUMERICAL WAVE BASIN

<table>
<thead>
<tr>
<th>Point</th>
<th>X (m)</th>
<th>Y (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.00</td>
<td>2.00</td>
<td>7.00</td>
</tr>
<tr>
<td>2.00</td>
<td>3.00</td>
<td>5.27</td>
</tr>
<tr>
<td>3.00</td>
<td>4.00</td>
<td>3.54</td>
</tr>
<tr>
<td>4.00</td>
<td>5.00</td>
<td>1.80</td>
</tr>
</tbody>
</table>

Figure 5 shows the time history of the surface elevation of the generated wave at different locations and it can be seen that the the phase of free surface elevation at different locations synchronized with each other very well, proving that the oblique wave was travelling in the prescribed angle. In addition, the magnitude of the free surface elevation match well with the expected value, 0.04m, thus the transfer function based on the linear potential flow theory has been correctly created. However, the surface elevation magnitude of point #1 was found to be slightly higher than the expected value, which is due to the fact that it is close to the paddles and beach zone on the side walls. It can also be observed that it takes about 5-6 cycles for the generated wave to become stable after the paddles start to move.

The spatial distribution of surface elevation along the oblique wave’s travelling direction is shown in Figure 6 (see line 1 in Figure 4(a)). For convenience of comparison, the theoretical results based on sinusoidal wave with the expected wave height and wave length are plotted together with the numerical results. It can be found that the numerical results matches well with the expected wave within the test region away from the beach zone (distance < 6m). However, the numerical surface elevation of wave deviate from the theoretical results when the location is close to the beach zone (distance > 6m), where the incident wave is significantly affected by the reflection from the beach. Especially, it is observed that free surface can rise up very high at the end of the beach (distance > 12m). Based on this observation it can be concluded that the test region with a wave of sufficient quality is small in the current study. However, it is acceptable because the aim of the current study was to validate the numerical wave basin’s capability to generate oblique waves. The area of the region affected by the reflection from beach is almost only determined the wave properties and the configuration of beach (slope, surface curve), so a larger test region can be expected through enlarging the dimension of wave basin in the future. It has not been done in the present study is due the fact that solv-
To further validate the capability of the numerical wave basin in generating oblique waves, a wave with larger height, 0.06m, was investigated. The same wave basin and wave gauges were utilized to monitor the quality of the wave generated. In Figure 7, the time history of surface elevation at four wave gauges along the same wave front plane is illustrated to validate the uniformity and propagating direction. Similar to the case with lower amplitude 0.04m, it is shown that the phase of wave at different gauges match with each other perfectly. Furthermore, the uniformity of the generated wave is also generally acceptable.
CONCLUSIONS

In this study, the technique of generating oblique waves in the numerical wave basin has been studied. The mathematical model and transfer function of generating oblique waves by paddles have been established and the numerical model based on the volume of fluid approach was utilized in this study. The numerical wave basin based on the CFD tool “interFoam” of OpenFOAM and the previously introduced wave theory has been tested for a linear wave travelling in the 60-degree direction. To minimize the influence of the reflection from the side wall, a kind of simple beach was employed at two sides.

The quality of the generated oblique wave was investigated in terms of the oblique angle and uniformity. The free surface geometry obtained in the numerical wave basin was compared with the theoretical results. All these findings indicated that the present numerical wave basin based on CFD tools and VOF can generate oblique waves as expected. It should be noted that the transfer function cannot only be employed in physical wave basins but also is applicable for numerical wave basin.

Due to the time constrains, the computational domain was kept small, which resulted in a small qualified test zone. In the future, a larger wave basin can be implemented to minimize the effect of reflection from beach and extend the test zone.

ACKNOWLEDGMENT

Financial support provided by Lloyd’s Register and A*STAR for this work is gratefully acknowledged.

DISCLAIMER

The views expressed in this paper are those of the authors and do not necessarily reflect those of their affiliated companies.

REFERENCES

[23] Berberović, E., van Hinsberg, N. P., Jakirlić, S., Roisman,