MODELING COMPLEX FLOW INDUCED BY WATER WAVES PROPAGATION OVER SUBMERGED SQUARE OBSTACLES

MONA A. TAWAB¹, TAMER HESHMAT², and ANDREAS SCHLENKHOFF³

¹Smart Engineering Systems Research Center (SESC), Nile University, Giza, Egypt,
²Smart Engineering Systems Research Center (SESC), Nile University, Giza, Egypt,
³Hydraulic Engineering Section, School of Architecture and Civil Engineering, Bergische Universität Wuppertal, Wuppertal, Germany

Submerged breakwaters are efficient structures used for shore protection. Many design features of these structures are captured upon modeling wave propagation over submerged square obstacles. The presence of separation vortices and large free surface deformations complicates the problem. A multiphase turbulent numerical model is developed using ANSYS commercial package. Careful domain discretization is done employing suitable mesh clustering to capture high gradients. Various numerical model parameters are provided, including grid size and time step. Special attention is directed towards clarifying turbulence initial conditions. Stable simulation results are obtained within acceptable computational time. Numerical results are validated quantitatively using subsurface measurements. Comparison along continuous horizontal and vertical velocity profiles is provided. Temporal and spatial model resolutions are illustrated for three test cases. The effect of wave period and height is well focused. The unsteady vortical structure is visualized. The incident wave energy is calculated and validated against theoretical values. The wave energy dissipation characteristics are briefly explained.

Keywords: CFD, Turbulence, Vortex, Multiphase, Breakwater, PIV.

1 INTRODUCTION

Accurate simulation of breakwaters is crucial to ensure safety during hostile sea conditions. Towards this goal modeling water waves possess the utmost importance (Goda 2000). Specifically, submerged breakwaters provide considerable shore protection while providing a clear sea view (Johnson et al. 2005). Analysis of wave propagation over submerged obstacles captures many features of real submerged breakwaters. However, two main challenges of the problem are: separation vortices, and strong free surface deformation. Both phenomena occur near relatively large-sized obstacles (Kasem et al. 2010). Accurate modeling of the turbulent rotational flow field is necessary to quantify wave energy dissipation.

Hence, turbulent, and multiphase models are needed to capture the problem physics. Developing these models is a nontrivial task. The challenges include fine discretization needed to achieve accuracy, avoiding the effects of reflected waves, and suitable turbulence initial conditions. Suitable grid clustering is crucial to perform transient simulations within acceptable durations. Specifically, clustering is needed near high gradients regions.
These difficulties are addressed by adopting a numerical model based on ANSYS commercial package (ANSYS-FLUENT 2013). Model details are well described. Special effort is dedicated to quantitative validation. This step is usually performed using measurements of free surface level and pressure (Cooker et al. 1990, Yuan et al. 2003, Kasem and Sasaki 2010). Measurements needed to verify the vortex dynamics are rather scarce. The relatively recent velocity measurements provided using the Particle Image Velocimetry (PIV) method (Kasem et al. 2014) are employed. These data extended over a two-dimensional window to capture the vortical flow field. Spatial and temporal numerical accuracy will be elucidated. Finally, a brief study of the influence of wave parameters on energy dissipation is provided.

2 GOVERNING EQUATIONS

The model assumptions are incompressible, multiphase, and turbulent flow. The governing equations are provided below.

\[
\begin{align*}
\frac{\partial u}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0 \quad (1) \\
\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) &= -\frac{\partial p}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2) \\
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) &= -\frac{\partial p}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3) \\
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x} (\rho k u) + \frac{\partial}{\partial y} (\rho k v) &= \frac{\partial}{\partial x} \left( \Gamma_k \frac{\partial k}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_k \frac{\partial k}{\partial y} \right) + G_k - Y_k + S_k \quad (4) \\
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x} (\rho \omega u) + \frac{\partial}{\partial y} (\rho \omega v) &= \frac{\partial}{\partial x} \left( \Gamma_\omega \frac{\partial \omega}{\partial x} \right) + \frac{\partial}{\partial y} \left( \Gamma_\omega \frac{\partial \omega}{\partial y} \right) + G_\omega - Y_\omega + S_\omega \quad (5) \\
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x} (\rho \omega u) + \frac{\partial}{\partial y} (\rho \omega v) &= 0 \quad (6)
\end{align*}
\]

Here \( t \) stands for time, \( x \) and \( y \) stand for horizontal and vertical space coordinates, respectively; \( u \) and \( v \) stand for horizontal and vertical velocity components, respectively; \( \rho \) and \( \mu \) stand for the fluid density and viscosity, respectively; \( k \) and \( \omega \) stand for turbulence kinetic energy and specific dissipation, respectively. The eddy viscosity \( \mu_t \) is calculated as \( \mu_t = \frac{k}{\omega} \). In the above equations, \( G_k \) and \( G_\omega \) represent generation terms, \( \Gamma_k \) and \( \Gamma_\omega \) represent effective diffusivity, \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \), respectively. The SST k-\( \omega \) turbulence model is adopted since it generally yields accurate predictions of pressure-induced separation vortices. The reader is referred to Menter (1994) for more details. The value of the volume fraction \( \alpha \in [0, 1] \) indicates whether a cell is occupied by air (\( \alpha = 0 \)) or water (\( \alpha = 1 \)). Eq. (1) is the continuity equation, Eq. (2) and (3) are the momentum equations, Eq. (4) and (5) represent the turbulence model, and Eq. (6) models the volume fraction.

Fluid properties were selected corresponding to water: \( \rho_2 = 998.2 \text{ kg m}^{-3} \) and \( \mu_2 = 1.003 \times 10^{-3} \text{ kg m}^{-1} \text{s}^{-1} \), and air: \( \rho_1 = 1.225 \text{ kg m}^{-3} \) and \( \mu_1 = 1.7894 \times 10^{-5} \text{ kg m}^{-1} \text{s}^{-1} \).

3 BOUNDARY AND INITIAL CONDITIONS

Stokes second-order waves are imposed at the inlet boundary. Slip velocity conditions are imposed on lower walls. No-slip conditions are imposed at the obstacle walls where enough grid clustering is employed. Turbulence wall boundary conditions are imposed using wall functions imposed on energy dissipation.
defined by White and Christoph (1971). A numerical beach is implemented on the last fifth part adjacent to the right boundary, to damp reflected waves. Damping terms include linear and quadratic terms whose values where $100 \text{ s}^{-1}$ and $100 \text{ m}^{-1}$, respectively.

Proper assignment of initial turbulence values is crucial to avoid excessive dissipation. Elhanafi, et al. (2017) emphasized two requirements: The initial values of $k$ and $\omega$ should be high enough to avoid instability and the kinematic eddy viscosity $v_t = \frac{k}{\omega}$ should be low enough to avoid extra damping. Based on numerical experiments the following initial values are adopted: $k = 1 \times 10^{-3} \text{ m}^2\text{s}^{-2}$ and $\omega = 500 \text{ s}^{-1}$.

### 4 NUMERICAL GRID AND EXPERIMENTAL SETUP

In the Figure 1 the dimensions of the square obstacle numerical domain are represented, with a measurement window of height $0.12m \times$ width $0.1m$. The window is located directly behind the obstacle as shown. The experiments are conducted using the $24m$ length physical wave flume located at Bergische Universität Wuppertal. The numerical domain is shorter than the physical wave flume, to reduce computational requirements. This numerical setup is suitable along with adequate incoming wave and numerical beach conditions. The water depth $h$ was fixed at $0.3m$ for all runs. The measurement window was populated with neutrally buoyant particles, whose density is close to that of water. Hence these particles should move with almost the same water velocity. Using suitable lighting, videos are recorded at a high speed of 120. Grid clustering is employed in the vicinity of the obstacle.

![Figure 1. Square obstacle numerical domain.](image)

Details of various mesh setups are omitted due to space limitation. However, the main features of the suitable used grid are provided. A structured rectangular grid is used away from the obstacle. For this grid, vertical stretching is adopted to yield a maximum and minimum spacing of $2.5 \times 10^{-2} \text{m}$ near slip walls (lower boundary), and $4.5 \times 10^{-3} \text{m}$ at free surface, respectively. Grid aspect ratio was maintained below 19 to preserve mesh quality. Horizontal spacing is enlarged six times at the numerical beach zone, to avoid unnecessary computations. A fine unstructured grid of average spacing of $4 \times 10^{-3} \text{m}$ is adopted near no-slip boundaries (obstacle walls), to well capture high gradients. Total number of grid points was about 20,400.

Table 1. The different parameters for the three cases.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>$H (\text{m})$</th>
<th>$T (\text{sec})$</th>
<th>$L (\text{m})$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.04</td>
<td>0.8</td>
<td>0.96</td>
</tr>
<tr>
<td>2</td>
<td>0.03</td>
<td>1.0</td>
<td>1.3719</td>
</tr>
<tr>
<td>3</td>
<td>0.04</td>
<td>1.0</td>
<td>1.3719</td>
</tr>
</tbody>
</table>
5 NUMERICAL AND EXPERIMENTAL RESULTS

Three different cases are adopted whose wave parameters are shown in Table 1. The symbols $H, T$, and $L$, stand for the wave height, wave period, and wavelength, respectively. The simulation time needed for each test case was about 180 minutes. The numerical time step was assigned a constant value of $1 \times 10^{-3}$ seconds. The minimum grid spacing, and simulated velocity were $0.0053 \, \text{m}$ and $0.35 \, \text{m/s}$, respectively. This corresponds to a maximum CFL of $0.06 \ll 1$, which lies within acceptable limits.

The CFD results for the three cases are compared with the wave-induced flow Particle Image Velocimetry (PIV) measurements by Kasem et al. (2014). Flow streamlines are shown in Figure 2. Two vortices adjacent to the square obstacle can be noticed, on upstream and downstream sides. The figure is zoomed at the downstream vortex for clarification.

![Figure 2. Streamlines around the square obstacle for case 3 at one instant.](image)

![Figure 3. $u$ (m/sec) versus $y$ (m) Case 1-Experimental (crosses) and Numerical (solid line).](image)

![Figure 4. $u$ (m/sec) versus $y$ (m), Case 2-Experimental (crosses) and Numerical (solid line).](image)

Numerical and experimental values are plotted for vertical and horizontal profiles of $u$ and $v$, respectively. The vertical (Figures 3, 4, and 5) and horizontal profiles (Figures 6, 7, and 8) are drawn on lines $x = 0.01 \, \text{m}$ and $y = 0.08 \, \text{m}$, respectively. Positions are defined as referring to the lower-left corner of the measurement window. The results are provided at six consequent instants (phases) on one period to illustrate temporal variations. Referring to Figures 3, 4, and 5, three stages can be observed. Initially, the flow speed increases near the obstacle top (phase $60^0$ for cases 2 and 3). Next, a vortex starts to develop depicted by coexisting positive and negative
velocities (phase $150^\circ$ for all cases). Finally, the vortex is dissipated as the flow direction is reversed (phase $360^\circ$ for cases 1 and 2 and $330^\circ$ for case 3). The same features can be observed in Figures 6, 7, and 8.

The incident wave average kinetic energy for nine various values of $H$ and $T$, is shown in Figure 8. The average kinetic energy per unit surface area is calculated theoretically as $\overline{KE}_1 = \frac{1}{16} \rho g H^2$ (Dean and Dalrymple 1991). The agreement between numerical and theoretical results is clear. The energy dissipation is clarified by considering the ratio $\overline{KE}_2/\overline{KE}_1$, where the subscripts 1 and 2 stand for the incident and transmitted waves, respectively. Numerical results for $\overline{KE}_2/\overline{KE}_1$ are provided in Figure 9 and kinetic energy ratio is given in Figure 10. For the nine cases $\overline{KE}_2/\overline{KE}_1$ is lower than one as expected, due to flow dissipation already discussed. A lower value of $\overline{KE}_2/\overline{KE}_1$ indicates higher energy dissipation and vice versa. A proportional relation between $T$ and $\overline{KE}_2/\overline{KE}_1$ can be noticed. On the contrary, the effect of $H$ is rather negligible.
6 CONCLUSION

Accurate numerical modeling of wave propagation over submerged square obstacles is performed. The numerical wave tank incorporates multiphase, turbulent, and unsteady flow. Various model parameters including boundary conditions, and grid clustering are clarified. Unlike many existing works, numerical results are verified using subsurface vertical and horizontal velocities profiles. Spatial and temporal model resolutions of induced flow field are verified. A novel outcome is the detailed visualization and explanation of the multiple vortices generated. In addition, the effect of wave parameters on energy dissipation is illustrated. The developed model will be employed in future works for detailed design of breakwaters.

![Figure 9](image1.png)  
Figure 9. Incident wave KE (Joule/m²) versus T (s). Numerical (discrete markers) and theoretical (lines).

![Figure 10](image2.png)  
Figure 10. Ratio (dimensionless) of the incident to transmitted KE versus T (s).

References