Numerical Simulation of Turbulent Structures under Breaking Waves

Zhihua Xie[1,3], **Maurizio Brocchini**[2], **Pengzhi Lin**[3]
[1]School of Engineering, Cardiff University, Cardiff, CF24 3AA, UK
[2]Department of Civil and Building Engineering, and Architecture, Universit a Politecnica delle Marche, 60131 Ancona, Italy
[3]State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University, China
E-mail Address of the presenting author: zxie@cardiff.ac.uk

HIGHLIGHTS

- A large-eddy simulation model is developed to simulate turbulent two-phase flows under breaking waves with pre- and post-breaking processes.
- The 3D Piece-wise Linear Interface Calculation Volume-of-Fluid (PLIC-VOF) is implemented for interface capturing.
- The surface tension effect is considered for the bubble and droplet break-up and coalescence.

1 INTRODUCTION

Wave breaking plays an important role in marine hydrodynamics, wave-structure interaction, air-sea interaction, surf zone dynamics, and nearshore sediment transport [1]. With the development of the numerical methods for the Navier–Stokes equations and free surface flows, several numerical studies have been performed to further our understanding of breaking waves, such as periodic waves in the surf zone [2, 3, 4], focusing wave [5], deep-water breaking waves in a periodic domain [6, 7, 8, 9, 10, 11], and shallow-water breaking waves over complex topography [12], which have provided much insight into the kinematics and dynamics of breaking waves, including the overturning jet and the subsequent splash-up process. In this study, a 3D large-eddy simulation based two-phase flow model is developed to investigate the turbulent vortical structures under the breaking waves, with the surface tension effect being taken into account for the small-scale bubble and droplet break-up and coalescence process. In addition, a Cartesian cut-cell method is developed to deal with complex topography and wave-structure interaction, which can provide detailed information on the impact force and energy dissipation during wave breaking.

2 MATHEMATICAL MODEL

The large-eddy simulation (LES) approach is adopted in this study, for which the large-scale turbulence is resolved and a subgrid-scale model is employed to compute the unresolved scales of turbulence. The governing equations used for incompressible two-phase flow are based on the spatially filtered Navier–Stokes equations, given as:

$$\boldsymbol{\nabla} \cdot \bar{\boldsymbol{u}} = 0, \tag{1}$$

$$\frac{\partial(\rho\bar{\boldsymbol{u}})}{\partial t}+\boldsymbol{\nabla}\cdot(\rho\bar{\boldsymbol{u}}\otimes\bar{\boldsymbol{u}})=-\boldsymbol{\nabla}\bar{p}+\boldsymbol{\nabla}\cdot[\mu(\boldsymbol{\nabla}\bar{\boldsymbol{u}}+\boldsymbol{\nabla}^T\bar{\boldsymbol{u}})]+\rho\boldsymbol{g}+\sigma\kappa\tilde{\mathbf{n}}\delta+\boldsymbol{\nabla}\boldsymbol{\tau}^{\mathrm{sgs}}, \tag{2}$$

where the overbar $\bar{\cdot}$ denotes the spatial filtering over the grid in Cartesian coordinates (x, y, z), $\bar{\boldsymbol{u}} = (\bar{u}, \bar{v}, \bar{w})$ is the filtered velocity vector and $\bar{p}$ is the filtered pressure. t is the time, $\boldsymbol{g}$ is the gravitational acceleration vector, ρ and μ are the density and dynamic viscosity of the fluid, σ is the surface tension coefficient, κ is the interfacial curvature, $\tilde{\mathbf{n}}$ is the interface unit normal, δ is the Dirac delta function.

The term $\boldsymbol{\tau}^{\mathrm{sgs}} = \rho(\bar{\boldsymbol{u}}\bar{\boldsymbol{u}} - \overline{\boldsymbol{u}\boldsymbol{u}})$ is the subgrid-scale (SGS) stress tensor and the anisotropic part of the SGS term is modelled by an eddy-viscosity model of the form:

$$\boldsymbol{\tau}^{\mathrm{sgs}}-\frac{1}{3}\mathrm{trace}(\boldsymbol{\tau}^{\mathrm{sgs}})\boldsymbol{I}=2\mu_t\bar{\boldsymbol{S}}, \tag{3}$$

where $\boldsymbol{I}$ is the unit tensor and $\bar{\boldsymbol{S}}$ is the strain rate tensor of the resolved field. μ_t is the turbulent eddy viscosity defined as:

$$\mu_t=\rho C_d\bar{\Delta}^2\left|\bar{\boldsymbol{S}}\right|, \text{and} \left|\bar{\boldsymbol{S}}\right|=\sqrt{2\bar{\boldsymbol{S}}\bar{\boldsymbol{S}}}, \tag{4}$$

with the cut-off length scale $\bar{\Delta} = (\Delta x\Delta y\Delta z)^{1/3}$ and the coefficient C_d is calculated by the dynamic SGS model in the present study.

The momentum equation is closed with the constitutive relations for the density and dynamic viscosity of the fluid

$$\rho=F\rho_{\mathrm{w}}+(1-F)\rho_{\mathrm{a}}, \tag{5}$$

$$\mu=F\mu_{\mathrm{w}}+(1-F)\mu_{\mathrm{a}}, \tag{6}$$

where the subscripts w and a denote fluid water and air, respectively. F is the volume fraction of water and the air-water interface is governed by

$$\frac{\mathrm{d}F}{\mathrm{d}t}=\frac{\partial F}{\partial t}+\bar{\boldsymbol{u}}\cdot\boldsymbol{\nabla}F=0. \tag{7}$$

These equations complete the mathematical description of the two-phase flow model.

3 NUMERICAL METHODS

A Cartesian grid multiphase flow solver (Xdolphin3D), based on the 3D LES approach, is used for the computations. The governing equations are discretised using the finite volume method on a staggered Cartesian grid. The advection terms are discretised using a high-resolution scheme, combining high order accuracy with monotonicity, whereas the gradients in the pressure and diffusion terms are obtained using a central difference scheme. A Cartesian cut-cell method [13] is developed to deal with the complex topography. The PISO algorithm is employed in the present study for the pressure-velocity coupling and a second-order backward Euler method is used for the time derivative. In this study, the model has been further developed to capture the air-water interface by a geometric PLIC-VOF (Piece-wise Linear Interface Calculation Volume-of-Fluid) method [14], and the continuum surface force model is implemented for the surface tension effect. The multiphase flow code Xdolphin3D has already been extensively verified and validated through numerous test cases for breaking waves [12], wave-structure interaction [15], and LES studies of free surface flows over rough beds [16].

4 RESULTS AND DISCUSSION

The third-order Stokes wave in a periodic domain is considered here, similar to previous 2D [6, 7] and 3D [8, 9, 10, 11] studies. The simulation is initialised with a modified potential solution of water waves, the third-order Stokes wave solution for the air-water interface and velocity potential [17] in the water. The velocity in the air is initialised as zero as little is known in the air. The computational domain of $L \times L \times 0.5L$ (where L is the wave length) is discretised by a uniform mesh $512 \times 512 \times 256$ in the streamwise, vertical, and spanwise direction. Periodic boundary conditions are used in both streamwise and spanwise direction. The corresponding Reynolds number and Weber number are $Re = 10^5$ and $We = 100$, respectively.

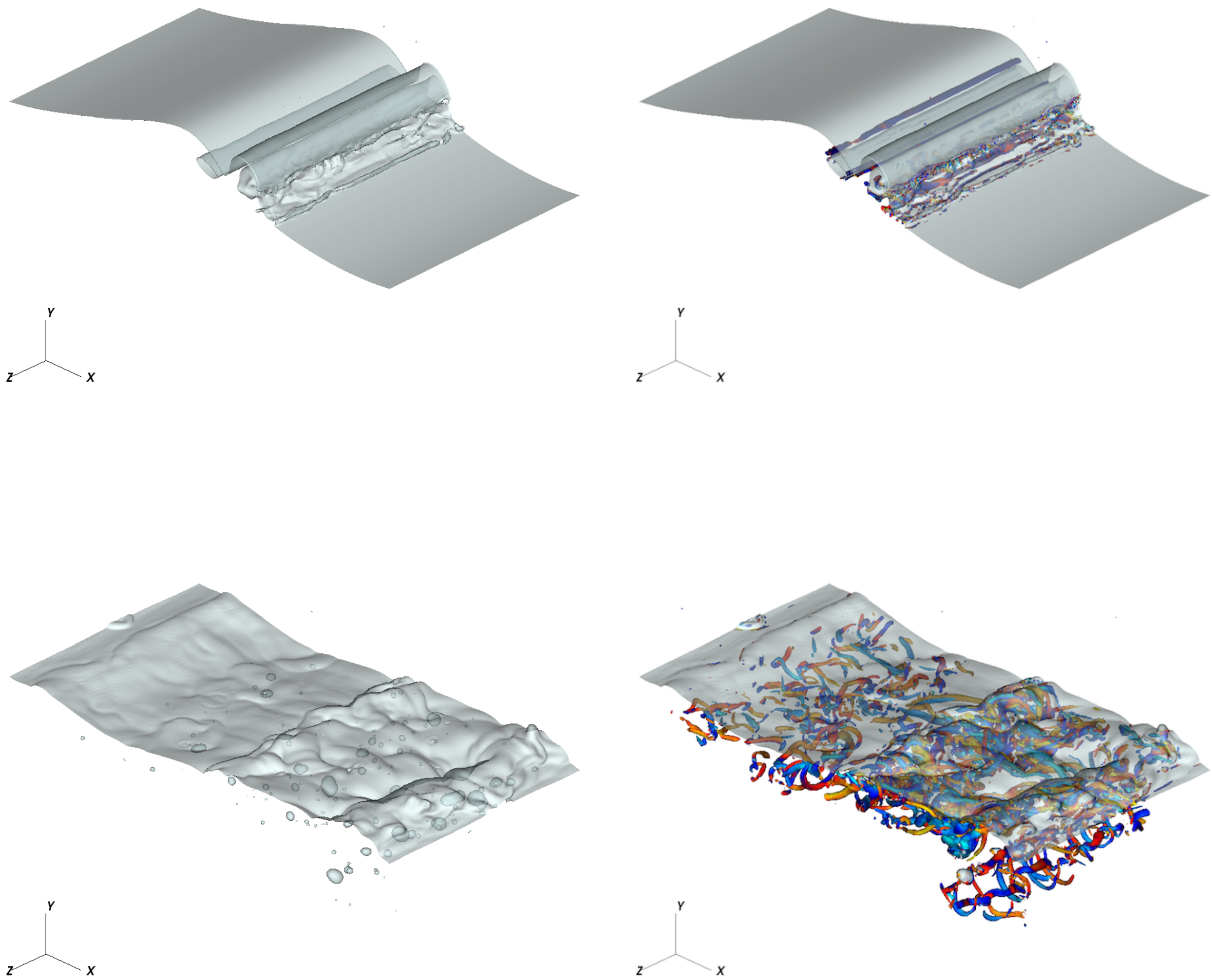

Figure 1: Snapshots of air-water interface (left panel) together with their vortex structures identified by the λ_2 method (right panel) at two instants during wave-breaking process.

Figure 1 shows some snapshots of the air-water interface and the vortex structures during wave breaking processes. It can be seen that the splash-up is generated when the plunging jet hits the water surface ahead. Small-scale bubbles and droplets are generated during the wave-breaking process. There is complex vortex structure-bubble interaction during the coalescence.

More results of the kinematics and dynamics of 3D breaking waves interaction with fixed and moving bodies will be presented at the workshop.

ACKNOWLEDGMENTS

This work was financially supported by the Engineering and Physical Sciences Research Council (EPSRC) through grant (EP/V040235/1), the Royal Society Newton Advanced Fellowship (NAF/R1/201156) and International Exchanges (IES/R2/202095), and the Open Fund of the State Key Laboratory of Hydraulics and Mountain River Engineering at Sichuan University (SKHL1904). The financial support from the MIUR PRIN 2017 Project, titled "FUNdamentals of BREAKing wave-induced boundary dynamics – FUNBREAK" (Grant Number 20172B7MY9) is also gratefully acknowledged.

REFERENCES

[1] Brocchini, M., and Peregrine, D. H. 2001. *The dynamics of strong turbulence at free surfaces. Part 1. Description. Journal of Fluid Mechanics 449*, 225–254.

[2] Lin, P., and Liu, P. L. F. 1998. *A numerical study of breaking waves in the surf zone. Journal of Fluid Mechanics 359*, 239–264.

[3] Watanabe, Y., Saeki, H., and Hosking, R. J. 2005. *Three-dimensional vortex structures under breaking waves. Journal of Fluid Mechanics 545*, 291–328.

[4] Lakehal, D., and Liovic, P. 2011. *Turbulence structure and interaction with steep breaking waves. Journal of Fluid Mechanics 674*, 522–577.

[5] Derakhti, M., and Kirby, J. T. 2016. *Breaking-onset, energy and momentum flux in unsteady focused wave packets. Journal of Fluid Mechanics 790*, 553–581.

[6] Chen, G., Kharif, C., Zaleski, S., and Li, J. 1999. *Two-dimensional Navier-Stokes simulation of breaking waves.* Physics of Fluids 11(1), 121–133.

[7] Iafrati, A. 2009. *Numerical study of the effects of the breaking intensity on wave breaking flows. Journal of Fluid Mechanics 622*, 371–411.

[8] Lubin, P., and Glockner, S. 2015. *Numerical simulations of three-dimensional plunging breaking waves: generation and evolution of aerated vortex filaments. Journal of Fluid Mechanics 767*, 364–393.

[9] Deike, L., Melville, W. K., and Popinet, S. 2016. *Air entrainment and bubble statistics in breaking waves. Journal of Fluid Mechanics 801*, 91–129.

[10] Wang, Z., Yang, J., and Stern, F. 2016. *High-fidelity simulations of bubble, droplet and spray formation in breaking waves. Journal of Fluid Mechanics 792*, 307–327.

[11] Chan, W. H. R., Johnson, P. L., Moin, P., and Urzay, J. 2021. *The turbulent bubble break-up cascade. Part 2. Numerical simulations of breaking waves. Journal of Fluid Mechanics 912.*

[12] Xie, Z. 2015. *A two-phase flow model for three-dimensional breaking waves over complex topography. Proceedings of the Royal Society A 471*, 20150101.

[13] Xie, Z., and Stoesser, T. 2020. *A three-dimensional Cartesian cut-cell/volume-of-fluid method for two-phase flows with moving bodies. Journal of Computational Physics 416*, 109536.

[14] Gueyffier, D., Li, J., Nadim, A., Scardovelli, R., and Zaleski, S. 1999. *Volume-of-fluid interface tracking with smoothed surface stress methods for three-dimensional flows.* Journal of Computational Physics 152(2), 423–456.

[15] Xie, Z., Stoesser, T., Yan, S., Ma, Q., and Lin, P. 2020. *A Cartesian cut-cell based multiphase flow model for large-eddy simulation of three-dimensional wave-structure interaction. Computers & Fluids 213*, 104747.

[16] Xie, Z., Lin, B., Falconer, R., Nichols, A.N. Tait, S., and Horoshenkov, K. 2021. *Large-eddy simulation of turbulent free surface flow over a gravel bed. Journal of Hydraulic Research.* , in press.

[17] Deike, L., Popinet, S., and Melville, W. 2015. *Capillary effects on wave breaking. Journal of Fluid Mechanics 769*, 541–569.