Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(5):800-804
DOI: 10.1016/S1001-6058(16)60791-6
www.sciencedirect.com/
science/journal/l0016058

Numerical simulation of three-dimensional breaking waves and its interaction with a vertical circular cylinder *

Zhihua Xie[1,2,3,4], Lin Lu (吕林)[2], Thorsten Stoesser[1], Jian-guo Lin (林建国)[5], Dimitrios Pavlidis[3], Pablo Salinas[3], Christopher C. Pain[3], Omar K. Matar[4]
1. *School of Engineering, Cardiff University, Cardiff, CF24 3AA, UK, E-mail: zxie@cardiff.ac.uk*
2. *State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116023, China*
3. *Department of Earth Science and Engineering, Imperial College London, London, SW7 2AZ, UK*
4. *Department of Chemical Engineering, Imperial College London, London, SW7 2AZ, UK*
5. *College of Environmental Science and Engineering, Dalian Maritime University, Dalian 116026, China*

(Received June 6, 2017, Revised July 29, 2017)

Abstract: Wave breaking plays an important role in wave-structure interaction. A novel control volume finite element method with adaptive unstructured meshes is employed here to study 3-D breaking waves. The numerical framework consists of a "volume of fluid" type method for the interface capturing and adaptive unstructured meshes to improve computational efficiency. The numerical model is validated against experimental measurements of breaking wave over a sloping beach and is then used to study the breaking wave impact on a vertical circular cylinder on a slope. Detailed complex interfacial structures during wave impact, such as plunging jet formation and splash-up are captured in the simulation, demonstrating the capability of the present method.

Key words: Breaking waves, volume of fluid method, 3-D simulation, Navier-Stokes equation, adaptive unstructured mesh

Introduction

Wave breaking plays an important role in marine hydrodynamics, wave-structure interaction, air-sea interaction, surf zone dynamics, and nearshore sediment transport. Several comprehensive reviews of breaking waves and wave mechanics can be found in Refs.[1-3].

Wave-structure interaction is a key aspect in the safe and cost-effective design of coastal and offshore structures, and marine renewable devices. In order to roughly predict the hydrodynamic loads on structures, the Morison equation and potential flow theory have been widely used in the literatures. However, it is challenge to consider breaking wave impact on the structures by using these two approaches, especially when there are splash-up and air entrainment[4].

With developments of computational fluid dynamics (CFD) and increases in computer power, recent models for studying free surface flows, including breaking waves, solve the Navier-Stokes equations coupled with a free surface calculation (see Lin[5] for comprehensive modelling applications and methodologies for water waves and McSherry et al.[6] for large-eddy simulation of free surface flows). There are several numerical studies on breaking waves in the literature based on one-phase flow model in 2-D[7-10] and 3-D[11,12], in which only the flow in the water is considered in the computations. The pressure in the air is taken as a constant, and the boundary conditions are approximately specified at the free surface. In order to take the effect of air phase into account for wave breaking, several 2-D two-phase flow model, in which both flows in the air and water are solved, have been developed to study the details of breaking waves in the surf zone[13,14], overturning jet during wave breaking[15] and the effect of wind on breaking waves[16,17]. Several 3-D two-phase flow models have also been developed to understand 3-D breaking waves in a periodic space domain[18], over a plane

* **Biography:** Zhihua Xie (1980-), Male, Ph. D., Lecturer

beach[19] and over a complex topography[20], which have provided much insight into the kinematics and dynamics of breaking waves, including the overturning jet and the subsequent splash-up process. Previous investigations have greatly improved our knowledge of breaking waves, however, little attention has been given for the numerical study on the 3-D wave action on a vertical circular cylinder over a slope.

In this study, a 3-D two-phase flow model with adaptive unstructured meshes is developed to investigate 3-D breaking wave interaction with a vertical circular cylinder along a constant sloping beach, which can provide detailed information on the impact force during wave breaking. The usage of adaptive unstructured meshes is helpful to reduce the computational efforts in CFD simulations and is also easy to deal with the problems with irregular boundaries. The description of the mathematical model for the two-phase flow is described in next section. The numerical method is presented after that. Both 2-D overturning waves and 3-D breaking wave impacting on a cylinder are simulated in the results and discussion section. Detailed computational results of the water surface profiles associated with the adaptive unstructured meshes are shown and discussed. Finally, conclusions are drawn.

1. Mathmatical model

A multi-fluid modelling framework has been developed based on the multi-component modelling approach with information on interfaces embedded into the continuity equations. In two-phase flows, let α_i be the mass fraction of phase i, where $i=1,2$, the density and dynamic viscosity of phase i are ρ_i and μ_i, respectively. A constraint on the system is

$$\sum_{i=1}^{2}\alpha_i = 1 \qquad (1)$$

For each fluid component i, the conservation of mass may be denoted as

$$\frac{\partial}{\partial t}(\alpha_i) + \nabla\cdot(\alpha_i \boldsymbol{u}) = 0\,,\ \ i=1,2 \qquad (2)$$

and the equations of motion of an incompressible fluid can be written as

$$\frac{\partial(\rho\boldsymbol{u})}{\partial t} + \nabla\cdot(\rho\boldsymbol{u}\otimes\boldsymbol{u}) = -\nabla p + \nabla\cdot[\mu(\nabla\boldsymbol{u}+\nabla^{\mathrm{T}}\boldsymbol{u})] + \rho\boldsymbol{g} + \sigma\kappa\boldsymbol{n}\delta \qquad (3)$$

where t is the time, $\boldsymbol{u}$ is velocity vector, p is the pressure, the bulk density is $\rho = \rho_1\alpha_1 + \rho_2\alpha_2$, the bulk dynamic viscosity is $\mu = \mu_1\alpha_1 + \mu_2\alpha_2$, $\boldsymbol{g}$ is the gravitational acceleration vector, σ is the surface tension coefficient, $\kappa = \nabla\cdot\boldsymbol{n}$ is the interfacial curvature, $\boldsymbol{n}$ is the interface unit normal, and δ is the Dirac delta function.

2. Numerical method

In the present study, a transient, mixed, control-volume and finite-element formulation is used to discretise the governing equations (Eq.(2) and Eq.(3)). A finite volume discretisation of the continuity equations and a linear discontinuous Galerkin (DG)[21] discretisation of the momentum equations are employed with backward Euler time stepping. Within each time-step, the equations are iterated upon using a projection-based pressure determination method until all equations are simultaneously balanced. The main numerical framework includes a finite element type P1DG-P2 (linear discontinuous velocity between elements and quadratic continuous pressure between elements) for multi-fluid flow problems, which ensures exact balance between buoyancy force and pressure gradient. The framework also features a novel interface capturing scheme based on compressive control volume advection method[21], involving a high-order accurate finite element method to obtain fluxes on the control volume boundaries, where these fluxes are subject to flux-limiting using a normalised variable diagram approach to obtain bounded and compressive solutions for the interface. The implementation of capillary/surface tension force in the framework using an unstructured mesh minimises spurious velocities often found in interfacial flows[22]. Finally, use of anisotropic unstructured mesh adaptivity[23] allows the grid resolution to be concentrated in relatively important regions, such as the vicinity of interfaces, while lower resolution can be used in other regions; this leads to a significant gain in computational efficiency without sacrificing accuracy.

The numerical framework has been validated and employed to study various multiphase flow problems[24-27]. The detailed modelling of three-dimensional bubbles, droplet and liquid films can be found in Xie et al.[22,28].

3. Results and discussion

3.1 *2-D breaking wave over a sloping beach*

In this section, we simulate a 2-D overturning solitary wave and compare quantitatively with the experiment[29] for a breaking solitary wave splash-up