Computers and Fluids 131 (2016) 56–65

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A coupled volume-of-fluid/immersed-boundary method for the study of propagating waves over complex-shaped bottom: Application to the solitary wave

Simon Gsell*, Thomas Bonometti, Dominique Astruc

Institut de Mécanique des Fluides de Toulouse, UMR No 5502 CNRS-INPT-UPS, Allée du Prof. Camille Soula, F-31400 Toulouse, France

ARTICLE INFO

Article history:
Received 12 August 2015
Revised 9 March 2016
Accepted 10 March 2016
Available online 15 March 2016

Keywords:
VOF
IBM
Solitary wave
Wave-bottom interactions
Wave shoaling
Wave breaking

ABSTRACT

We report about a numerical approach based on the direct numerical simulation of the Navier–Stokes equations for the study of wave-bottom interaction problems. A Volume of Fluid (VOF) method is coupled with an Immersed Boundary Method (IBM) and applied to the simulation of propagating waves over complex shaped bottoms. We first investigate the flow induced by a solitary wave over generic bottoms (i.e. a semi-circular cylinder and a sloping beach). We show that the method is able to describe various important features of wave-bottom interactions, including flow separation, vortex shedding and wave breaking, while keeping a reasonable computational effort. Then we demonstrate the capability of the present approach to model arbitrary shaped bottoms by simulating the run-up of a breaking solitary wave over a natural beach profile.

1. Introduction

After being generated in the open sea by wind or geophysical events, gravity waves propagate toward the coast, carrying a considerable amount of energy. Close to the shore, the wave dynamics changes as a result of the interaction with the bottom. This is the shoaling process, which may include wave breaking. Knowing the mechanisms that take place during the wave shoaling is a key issue for many engineering applications involving sediment transport, civil engineering, shore protection and energy extraction. During the last decades, numerical simulation has proved to be a promising tool for the study of wave-body and wave-bottom interaction problems, as demonstrated by the wide variety of numerical methods applied to the study of waves interacting with a submerged body in the literature (e.g. [1–3]).

Many studies of surface waves are based on potential flow theory, under the assumption of irrotational and inviscid flow. The fluid equations are usually reduced to a Laplace equation for the velocity potential and a set of non-linear boundary conditions. Two families of numerical techniques using potential flow theory are the Boundary Element Methods [4–7] and the Spectral Methods [1,8–11] which are based on pertubative expansions and are able to model both constant- and variable-depth problems [12,13]. When viscous effects such as vortex shedding and energy dissipation in the boundary layers have to be locally taken into account to describe the wave dynamics, potential flow models can be coupled with a Navier–Stokes solver through a domain decomposition approach [14–16]. Note that Lin and Huang [2,17] also proposed a vortex method to take into account the generation and shedding of vorticity due to the presence of a solid body.

Another class of numerical techniques is based on the long wave theory using in particular the Boussinesq equations (e.g. [18,19]) and the non-linear shallow-water equations [20–22]. The similarity between shallow-water and gas dynamics equations allows to apply efficient shock-capturing schemes, initially developed for Euler equations, to investigate bore dynamics resulting from breaking waves [23,24]. However, as both the non-linear shallow-water and the Boussinesq equations are based on hydrostatic or almost hydrostatic approximation, they fail to predict the complex interaction between a wave and a bottom of arbitrary shape.

In order to capture all the flow characteristics resulting from a wave-body interaction problem, one may rather choose to solve the full incompressible Navier–Stokes equations. A major challenge raised by this numerical strategy deals with the treatment of the free surface dynamics.

Various techniques using boundary-fitted moving grids have been used [3,25–27]. These methods allow an accurate interface tracking as the mesh fits the shape of the free surface. However, it can hardly be applied to complex interface deformations as wave breaking. Eulerian methods allow instead to use a fixed

* Corresponding author. Tel.: +33534322912.
E-mail address: simon.gsell@imft.fr (S. Gsell).

http://dx.doi.org/10.1016/j.compfluid.2016.03.013

grid on which the interface is free to deform. The capture of the interface is ensured by the convective transport of either a finite number of markers or a continuous function. In particular, the Volume of Fluid (VOF) method has been extensively used for the study of wave-breaking and wave run-up [28–31] as well as wave-body interaction [32–34]. This method has been shown to be able to deal with complex phenomenon as wave breaking and air entrainment. When the shape of the bottom remains simple, it is convenient to use a boundary-fitted grid associated with a no-slip condition. However, with such a method the computational effort rapidly increases when increasing the geometrical complexity of the boundaries. In order to treat wave dynamics problems with increasing complexity while keeping a reasonable computational cost, VOF-type methods can be coupled with Immersed Boundary Methods (IBM). The IBM technique enables to place bodies of arbitrary shape in a computational domain discretized with a structured Cartesian grid. This numerical approach has already been applied to the interaction between surface waves and submerged obstacles [35–37]. In this type of configuration, the water/air interface does not cross the water/body interface. More recently, physical configurations involving partially immersed bodies (water entry of a sphere and dam break with an obstacle) have been investigated by Zhang et al. [38] and Zhao et al. [39].

In this paper, a coupled VOF-IBM method is used to simulate the flow induced by a solitary wave interacting with a complex shaped bottom. The numerical method is applied to geometries of increasing complexity. Both submerged and partially immersed obstacles are considered. We first investigate the flow induced by a solitary wave interacting with a submerged semi-circular cylinder, showing that our results are in good agreement with those of Klettner and Eames [3] which were obtained with a boundary-fitted approach. Then, the run-up of non-breaking and breaking solitary waves on a sloping beach is investigated and compared to detailed experiments of Synolakis [40]. Finally, we present simulations of the run-up of a breaking solitary wave on a natural beach of complex topography, the shape of which being the result of wave-induced sediment transport observed in laboratory experiments.

2. Numerical method

2.1. Governing equations and assumptions

Let us consider two immiscible fluids, i.e. air and water, of density ρ_a and ρ_w, and dynamic viscosity μ_a and μ_w, respectively. We assume the two fluids to be Newtonian and incompressible. Considering relatively large amplitude gravity waves, we neglect the surface tension effects in the following. The evolution of the two-phase flow is then described by the one-fluid formulation of the Navier–Stokes equations [41], namely

$$\frac{\partial \mathbf{V}}{\partial t} + \nabla \cdot (\mathbf{V} \otimes \mathbf{V}) = \mathbf{g} - \frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla \cdot [\mu(\nabla \mathbf{V} + \nabla \mathbf{V}^T)] + \mathbf{f}, \quad (1)$$

$$\nabla \cdot \mathbf{V} = 0, \quad (2)$$

where $\mathbf{V}$, P, ρ and μ denote the local velocity, pressure, density and viscosity in the flow, respectively, $\mathbf{g}$ denotes gravity and $\mathbf{f}$ is a volume force term used to take into account solid-fluid interaction. The local volume fraction C of the air obeys

$$\frac{\partial C}{\partial t} + (\mathbf{V} \cdot \nabla) C = 0. \quad (3)$$

This volume fraction equals one (resp. zero) in cells filled with air (resp water) while values of volume fraction lying between 0 and 1 indicate the presence of an air-water interface. The local density and viscosity are computed from the volume fraction as $\rho = C\rho_a + (1-C)\rho_w$ and $\mu = C\mu_a + (1-C)\mu_w$, respectively. In the present method, no interface reconstruction step is employed so the numerical thickness of the interface is not strictly zero, but typically spreads over three grid cells [42]. Eqs. (1)-(3) are solved throughout the entire computational domain, including the actual fluid domain and the space occupied by the solid.

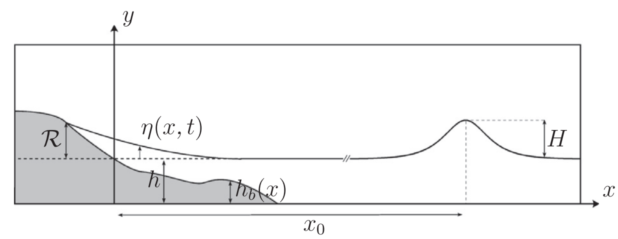

Fig. 1. Schematic view of the set-up. The solitary wave of amplitude H and depth h is initially placed at x_0. The vertical position of the air-water interface is $h+\eta$ while that of the intersection point between the air-water interface and the immersed boundary is $y = h + \mathcal{R}$, $\mathcal{R}$ being the run-up elevation.

2.2. Modeling of the immersed solid

Following the IBM method [41], the interaction between the fluid and the immersed body is carried out by the addition of a volume force term $\mathbf{f}$ in (1). We first define a solid volume fraction $\alpha(\mathbf{x})$ accounting for the presence of the immersed solid. The value of the parameter α is set to 1 in the solid region and 0 in the fluid region. A transition region is introduced, where α values are laying between 0 and 1. Here we consider the particular case of fixed bottom or bottom-seated obstacles. Assuming that the bottom shape is described by a function $h_b(x)$ (see Fig. 1), we define the solid volume fraction as [41]

$$\alpha(\mathbf{x}) = \frac{1}{2}\left\{1 - tanh\left(\frac{y - h_b(x)}{\lambda \eta_i \Delta}\right)\right\} \quad (4)$$

$$\lambda = |n_x| + |n_y| + |n_z| \quad (5)$$

$$\eta_i = 0.065(1 - \lambda^2) + 0.39 \quad (6)$$

where $\mathbf{n} = (n_x, n_y, n_z)$ is the normal outward unit vector at the surface, η_i is a parameter controlling the thickness of the transition region and Δ is a characteristic grid size ($\Delta = \sqrt{2}\Delta x$ for a 2D uniform grid). The force $\mathbf{f}$ is then defined as

$$\mathbf{f} = \alpha \frac{\mathbf{V}_s - \widetilde{\mathbf{V}}}{\Delta t}, \quad (7)$$

where Δt is the time step used for the time-advancement, $\mathbf{V}_s$ is the local velocity imposed in the solid object ($\mathbf{V}_s = 0$ here), and $\widetilde{\mathbf{V}}$ is an intermediate velocity field without considering the immersed object. For more precision, see Section 2.4.

2.3. Time stepping and spatial discretization

Eqs. (1) –(3) are solved on a staggered Cartesian grid following a finite-volume approach [43]. The time integration of (1) and (2) is performed via a third-order Runge–Kutta method for all terms except the viscous term for which a second-order semi-implicit Crank–Nicolson scheme is used [44]. The incompressibility condition (2) is satisfied at the end of each time step through a projection method. The transport equation of volume fraction (3) is solved by using a modified version of the flux-corrected transport scheme proposed by Zalesak [45]. Domain decomposition and Message-Passing-Interface (MPI) parallelization is performed to facilitate simulation of large number of computational cells.