Computers & Fluids 100 (2014) 347–355

Contents lists available at ScienceDirect

Computers & Fluids

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

A two-way coupled simulation of moving solids in free-surface flows

Tso-Ren Wu [a], Chia-Ren Chu [b,*], Chih-Jung Huang [b], Chung-Yue Wang [b], Ssu-Ying Chien [b], Meng-Zhi Chen [a]

[a] Institute of Hydrological and Oceanic Sciences, National Central University, Taiwan, ROC
[b] Department of Civil Engineering, National Central University, Taiwan, ROC

ARTICLE INFO

Article history:
Received 17 October 2013
Received in revised form 31 March 2014
Accepted 10 May 2014
Available online 15 May 2014

Keywords:
Large Eddy Simulation
Fluid structure interaction
Moving solid
Discrete Element Method
Volume of Fluid
Free surface flow

ABSTRACT

This study developed a novel two-way dynamic coupled numerical model to simulate moving solids in free surface flows. The fluid flows and hydrodynamic pressures are simulated by a Large Eddy Simulation model, and the free surface is tracked by the Volume-of-Fluid (VOF) method. The fluid response from the solid motion is modeled through specifying the cell-face velocity with partial-cell treatment (PCT). The displacement and rotation of the solids are calculated by the Discrete Element Method (DEM). In order to verify the present model, two laboratory experiments of rectangular blocks floating and sinking in a water tank are conducted. The numerical simulations compare favorably with the experimental results on the trajectory of the moving blocks. The numerical scheme presented in this paper can be used as a design tool for practical problems involved moving objects in free-surface flows.

1. Introduction

Dynamic coupled simulation of fluid/structure interaction, due to its wide application in engineering problems, has gained a lot of interests in recent years [1–3]. The interaction between incompressible fluids and moving (rigid or flexible) objects occurs in civil engineering [4,5], aeroelasticity [6] and hemodynamics [3,7]. Among them, coastal engineering [8] has the problems of floating breakwater [9] and the crane vessels and barge operating on the water surface, involved the interaction of free surface flows and moving objects. Numerical simulations are useful to obtain detailed information on the interaction process and to assist engineering design.

According to O'Brien et al. [10], there are three types of coupling to calculate the interaction between fluid and solids: (1) one-way solid-to-fluid coupling; (2) one-way fluid-to-solid coupling and (3) two-way dynamic coupling. The first type calculates the fluid flow with the prescribed motion of the solids. For example, Chen and da Vitoria Lobo [11] simulated the fluid flow with moving obstacles by solving the Navier–Stokes equations. They treated the solids as massless marker particles and these particles could move unrestrainedly on the surface of the water. They demonstrated both types of one-way coupling problems. Since then, many researchers have developed different one-way coupling models.

O'Brien and Hodgins [12] first described a numerical method to simulate the dynamic behavior of splashing fluids when an object impacts on the water surface, but the accuracy of their simulation is not very good. Foster and Metaxas [13,14], Stam [15] and Fedkiw et al. [16] developed couple numerical models in which the rigid bodies were treated as boundary conditions with known velocities. So, the boundary condition can be applied directly on the cell faces with the prescribed solid motion. This method can be categorized as the Cell-Face Velocity (CFV) method because the velocity of the solid is explicitly set on the cell faces. However, the above studies focused on the real time animation rather than the accuracy of the simulation results.

Liu et al. [17] developed a one-way coupled model to simulate landslide generated water waves on a fixed grid. They modified the Partial Cell method by adding a source function at the region where the solids move. Compared with the Cell-Face Velocity method, the Partial Cell Treatment has better accuracy in terms of mass conservation. However, adding a source term in the Pressure Poisson Equation often causes fluctuations on the pressure field which generates instability in the calculation of two-way coupling.

The second type of numerical model is one-way fluid-to-solid coupling. The fluid forces the solids to move without the solids affecting the fluid. Foster and Metaxas [13] demonstrated this type of coupling by animating a tin can floating on the surface of swelling waters. However, the volume change of the tin can will not affect the fluid flow in this type of one-way coupling. In the

* Corresponding author. Tel.: +886 3 4227151; fax: +886 3 425 2960.
E-mail address: crchu@cc.ncu.edu.tw (C.-R. Chu).

http://dx.doi.org/10.1016/j.compfluid.2014.05.010
0045-7930/

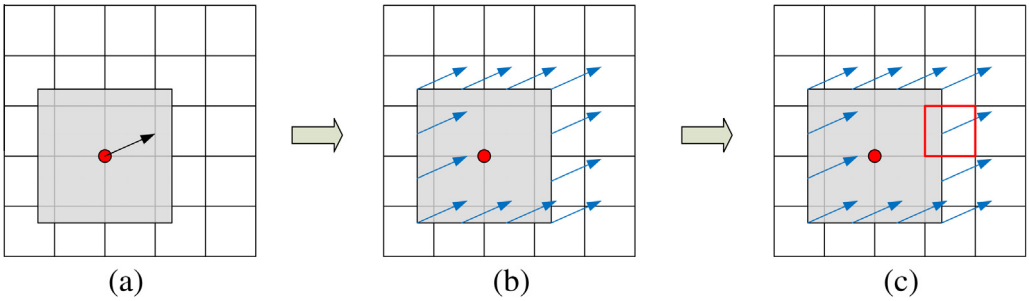

Fig. 1. Schematic diagram to illustrate no-slip boundary condition on a moving solid. The grid is the fluid grid. The gray area represents a solid block. The red dot is the centroid of the block. The black and blue arrows are the moving vector of the block. (a) A moving block in the fluid domain. (b) The velocity vectors on the surfaces of the block. (c) A typical fluid cell (boundary in red) locates on the interface of the solid block and fluid. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

third type of model, two-way dynamic coupling, the fluid pressure will push the solids, and the motions of the solids impose forces back on the fluids. For example, Takahashi et al. [18] combined the Volume of Solid (VOS) and Cubic Interpolated Propagation (CIP) method to simulate solid motion in incompressible fluids. Takahashi et al. [19] used their model to simulate the liquid motion with splashes and foams. However, the dynamic forces and torques were neglected in their model.

Singh et al. [20] used the Arbitrary Lagrangian–Eulerian (ALE) method, introduced by Hirt et al. [21], to two-way couple the solids and fluid. However, this method has two main drawbacks. First, the computational grid has to re-mesh when the displacements of the solids are moving. Second, the method needs two layers of meshes in the gap between solids when they approach one another. Génevaux et al. [22] used a mass/spring system to model the interaction between incompressible fluids and deformable bodies. This method mainly relied on the definition of the force between solids and fluids to bridge the gap between the traditional Eulerian fluid models and Lagrangian solid models. However this technique is difficult to use for solids with complex geometry and the discretization of the force field is imprecise.

Muller et al. [23,24] developed the Smoothed Particle Hydrodynamics (SPH) to simulate the free surface flow around solids. Since then, the Smoothed Particle Hydrodynamics is widely used for computer animation [25,26] of fluid/solid interactions. However, the numerical error and particle inconsistency accumulates after several interactions between the solid boundary and particles [27].

In this study, a two-way coupled Moving Solid Algorithm (MSA) was proposed to calculate the motion of the solids in water flows with free surface. The water flow was solved by a Large Eddy Simulation model, and the water surface was tracked by the Volume of Fluid (VOF) method using a fixed grid. The motion of the solids was determined by the Discrete Element Method, and does not need to be prescribed beforehand. Instead of adding a source term in the Poisson Pressure Equation (PPE), we specified the face velocity on the cells intersected with the solids. The Partial-Cell Treatment was adopted to satisfy the velocity on the solid faces. This new method also enforces the conservation of fluid mass. The accuracy of the simulation results was verified by comparison with laboratory experiments of falling and floating solids in water.

2. Numerical model

2.1. Turbulence model

This study used a three-dimensional Large Eddy Simulation (LES) model to compute the flow field around the solids. The fluid motion was simulated by solving the continuity equation and the Navier–Stokes equations with the Volume of Fluid (VOF) free surface tracking algorithm. The governing equations can be expressed as:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{\rho u_i}}{\partial t} + \frac{\partial \rho \overline{u_i u_j}}{\partial x_j} = -\frac{\partial \overline{P}}{\partial x_i} + \rho g \delta_{i3} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right)\right] \tag{2}$$

where the subscripts $i, j = 1, 2, 3$ represent the x, y and z directions, respectively; t is the time, $\bar{u}$ and $\overline{P}$ are the filtered velocity and pressure [28], ρ is the density of the water, g is the gravitational acceleration and μ_{eff} is the effective viscosity, defined as:

$$\mu_{\text{eff}} = \mu + \mu_{\text{SGS}} \tag{3}$$

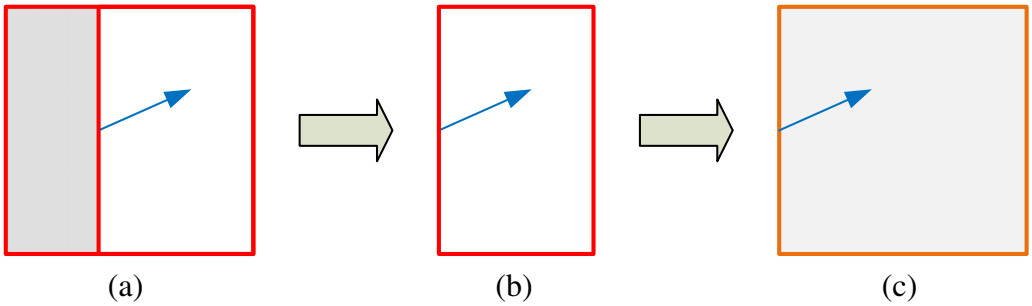

Fig. 2. Schematical diagram to depict the conceptural steps of the moving solid algorithm. (a) A numerical cell is partially occupied by a solid material (in gray) with a moving speed (blue arrow). (b) The effective volume of the fluid cell is θV. (c) The partial-cell method expands the cell back to the original size with the same effective cell volume θV. The speed of the moving solid therefore can be applied on the cell face. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)