Computers & Fluids 89 (2014) 66–77

Contents lists available at ScienceDirect

Computers & Fluids

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

Simulations of free-surface flows with an embedded object by a coupling partitioned approach

C.S. Wu, D.L. Young *

Department of Civil Engineering and Hydrotech Research Institute, National Taiwan University, Taipei 10617, Taiwan

ARTICLE INFO

Article history:
Received 5 December 2012
Received in revised form 30 July 2013
Accepted 23 October 2013
Available online 29 October 2013

Keywords:
Free surface
Hybrid Cartesian/immersed boundary method
Modified height function
Navier–Stokes equations
Moving object

ABSTRACT

This article describes numerical investigations of the flow and wave patterns under stationary or moving submerged objects in a viscous fluid. To incorporate the effects of the free motion objects as well as the free surface, the modified height function scheme is implemented to accurately capture the configurations of free surfaces. For dealing with complex submerged objects in the fluid, a hybrid Cartesian/immersed boundary method is adopted to allow imposition of the solid boundary conditions with a linear interpolation approach. The considered physical model is developed for incompressible, unsteady free-surface flows to satisfy the condition of volume conservation based on the staggered finite-difference spatial discretization. Possible free-surface configurations are described by a high-order flux corrected transport model to maintain the sharp interface and in the mean time to eliminate the surface numerical oscillations. Finally several examples are provided to assess the performance of the developed numerical model. Four numerical validated examples are used to respectively demonstrate the proposed schemes. They are (1) the flow past a circular cylinder, (2) in-line oscillating circular cylinder in a fluid, (3) liquid sloshing in a partially filled rectangular tank and (4) the oscillatory sloshing tank over a shaking table to test the total volume preservation. In addition, two more numerical experiments are carried out to simulate (5) the free-surface flows with the embedded solid body subject to stationary horizontal cylinder and (6) free-surface simulations induced by an oscillating moving object. Both tested cases show encouraging results as well by the present algorithm.

1. Introduction

Multi-dimensional numerical simulations of free-surface flows such as earthquake excited fluid–structure interactions of dam reservoir systems, tsunami propagation in sea water, and water wave generated due to landslides are the common events for risk analysis in engineering applications. To capture both the features of flow and wave patterns with satisfactory resolution in space and time domains, in general the Navier–Stokes equations are solved numerically using a suitable discretization technique such as the finite difference (FD), finite volume (FV), or finite element (FE) methods. To develop a numerical model in relatively easy implementation, the FD discretization method is employed in this study based on a staggered grid system. However the conservation of physical quantities will also be involved in our numerical computations.

Free-surface flows and moving boundary problems pose a challengeable issue for both theoretical and experimental studies [1,2]. It is one kind of fluid–structure interaction problems. The interactions are nonlinear multi-physics phenomenon applied to a wide range of engineering discipline. The hydrodynamic instabilities of the wave motion as well as strong interface tearing and stretching are rather difficult issues to analyze in the theoretical studies. Hence the alternative capability of solution procedure is employed to a significant extent by the numerical treatments. Accurate preservation of discontinuities at the free surface is particularly important to the overall accuracy of the flow solver. Concerned developments over decades have yielded two general categories. They are the interface tracking methods and the interface capturing methods. The representative approach of interface tracking methods is the marker-and-cell (MAC) concept [3,4]. As far as the interface capturing methods are concerned, there are two main approaches used, namely the volume-of-fluid (VOF) method [5] and the level set method (LSM) [6], which are among the most commonly used schemes. The detailed literature reviews of these two commonly used interface tracking [7] and interface capturing methods [8,9] can be found in the references. To preserve the advantages of the above-mentioned methods, this proposed numerical model aims to satisfy the mass conservation possibly and to overcome the discontinuous physical quantities near the interface. However, since the target in this study is also to maintain relatively easy implementation and good computational efficiency on the treatment of free surface, the kinematic condition at the free

* Corresponding author. Fax: +886 2 2362 6114.
E-mail address: dlyoung@ntu.edu.tw (D.L. Young).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.10.030

surface is modified and advanced, namely the modified height function (MHF) method. This method is one of the interface tracking approaches in which the height values at a set of horizontal locations are recorded in a single value function during the transients. Memory requirements for a numerical solution are extremely small. The method in this paper is implemented to predict the configurations of the free surface without considering the liquid-breaking effects.

To describe the immersed body in the fluids, the immersed boundary method (IBM) was widely utilized to handle the flow field with complex geometries in the computational fluid dynamics (CFD). One of the numerical challenges in CFD is to deal with moving boundaries in the fluid. The technique was then proposed to cope with the geometries undergoing arbitrary complex motion and deformation [10–12], so that we can simplify grid generation to fit a complex moving boundary. The main idea of the IBM is to represent the effect of an embedded body within the fluid domain by adding an equivalent body force in the governing equations. However there is flow penetration often existing in the conventional IBM results, especially for the non-slip boundary condition on the immersed body. To avoid the problem, Wu et al. [13] developed a local domain-free discretization-immersed boundary method to accurately capture the immersed body. Until now, the improvements of the IBM are consistently making progress [12]. In most instances, since the IBM did not coincide with the regular fluid grids, a considerable number of contributions were derived to improve the interpolation schemes for imposing the desired velocity with boundary conditions of immersed body.

In 1977, Mohd-Yusof [14] first suggested an alternative direct forcing formulation for a specific case so that there was no need to couple the effect of the fluid to the solid. They applied the direct forcing only on the immersed boundary or inside the body, and the interpolation scheme were implemented in the B-spline direction. Fadlun et al. [15] reconstructed the velocity at the first grid point external to the immersed boundary and developed a fully three-dimensional immersed boundary model to treat the complex flow with a moving boundary. The desired velocity is obtained from the linear approximation of the velocity at the immersed boundary and the second grid point external to the immersed body. Balaras [16] proposed a simple interpolation scheme which the interpolated direction was normal to the immersed body over a fixed Cartesian grid. Gilmanov and Sotiropoulos [17] proposed a second-order accurate, dual-time-stepping artificial compressibility algorithm to investigate flow past an undulating fish-like body. Uhlmann [18] incorporated Peskin's regularized delta function scheme into a direct-forcing formulation for a smooth transfer between Eulerian and Lagrangian representations. The numerical simulation of the particle sedimentation was also carried out in that study. Kim and Choi [19] even proposed a non-inertial-reference-based HCIB method to simulate a free falling sphere under the gravitational field. A mixing linear and bi-linear interpolation technique was further introduced to estimate the momentum forcing on or inside an immersed boundary.

The issues still exist, however, the choice of interpolation direction is arbitrary and their use would be restricted to some specific situations. On the basis of the developed numerical model in a relatively easy implementation, the Balaras' concept [16] is followed in this paper. The employed technique combines the direct forcing formulation concept and the interpolated scheme in a fixed grid is known as the hybrid Cartesian/immersed boundary (HCIB) method. It provides the advantage of a simple mesh generation, as comparing to the other sophisticated boundary-fitted methods. The basic concept of the HCIB method is to modify the entries of the implicit matrix of the discretized momentum equations. This is achieved by imposing the boundary conditions at the solid surface at each time step. This method does not require a smaller computational time step to satisfy the stability of the discrete-time equation, and it holds regardless of free constants that makes the derivation of forcing independent on the Reynolds number. This concept also allows us to calculate flow around objects moving relative to the environments without additional difficulty. Recently, Young et al. [20,21] applied this concept to investigate the fluid and heat patterns in a moving two-roll mill model by using the HCIB method. Xu [22] also extended the similar concept to develop a pressure solver to handle two-fluid flow problem across the interfaces with second-order accuracy. Now, the HCIB method is employed in our study that the fluid grid is fixed throughout the computation for an arbitrary moving immersed boundary during the transients.

From the literature survey, it is investigated that the numerical model seldom treats the complicated fluid–structure interaction problem with moving boundaries over simple fixed grids. The novelty of this paper is to combine the HCIB technique and the MHF method together to simulate the free-surface flows interact with the embedded stationary or moving objects. The HCIB technique is applied to handle the moving boundaries in the fluids, while the MHF is used to capture the evolution of free surface. The capability of the numerical model will be demonstrated through several case studies.

2. Mathematical formulations of fluid motion with free-surface effects

2.1. Governing equations

Consider an incompressible, viscous fluid occupying at an instant time t and the domain Ω with a smooth boundary Γ. The dimensionless Navier–Stokes equations in terms of the primitive variables for the fluid flow are the mass conservation equation

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

and the momentum conservation equation

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}}\nabla^2 \mathbf{u} - \mathbf{g} + \mathbf{f}, \tag{2}$$

in which $\mathbf{u} = (u, v)$ and p are the dimensionless velocity vector and pressure, respectively; Re represents the Reynolds number; $\mathbf{g}$ is the gravitational force; and $\mathbf{f}$ is the forcing function used to make the fluid velocity to satisfy the desired boundary conditions.

In computations, to minimize the error from the convective term and overcome the numerical oscillation, the balanced tensor diffusivity term [23], $\frac{\Delta t}{2}(\mathbf{u}^n \cdot \nabla)[(\mathbf{u}^n \cdot \nabla)\mathbf{u}^n]$, is added and used to compensate the explicit discretization errors in time and thus the convection term is corrected. This term is also regarded as an upwind technique. As a consequence, the operator splitting procedures of the Navier–Stokes equations are summarized in the following steps [20,21,24].

Step 1: Intermediate velocity

$$\frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = -[\mathbf{u}^n \cdot \nabla]\mathbf{u}^n + \frac{\Delta t}{2}(\mathbf{u}^n \cdot \nabla)[(\mathbf{u}^n \cdot \nabla)\mathbf{u}^n] + \frac{1}{\mathrm{Re}}\nabla^2 \mathbf{u}^n - \mathbf{g} + \mathbf{f}. \tag{3}$$

Step 2: Pressure calculation

$$\nabla^2 p^{n+1} = 1/\Delta t(\nabla \cdot \mathbf{u}^*). \tag{4}$$

Step 3: Velocity correction

$$\mathbf{u}^{n+1} = \mathbf{u}^* - \Delta t(\nabla p^{n+1}). \tag{5}$$

In which $\mathbf{u}^*$ represents the intermediate velocity vector.