Applied Ocean Research 72 (2018) 92–109

Contents lists available at ScienceDirect

Applied Ocean Research

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

Enhancement of Navier–Stokes solver based on an improved volume-of-fluid method for complex interfacial-flow simulations

Van-Tu Nguyen, Warn-Gyu Park*

School of Mechanical Engineering, Pusan National University, Busan, 46241, South Korea

A R T I C L E I N F O

Article history:
Received 29 June 2017
Received in revised form
28 December 2017
Accepted 16 January 2018

Keywords:
Volume of fluid
Chimera grid
6DOF
Dam-break
Free surface
Water entry
Breaking wave

A B S T R A C T

In this study, an extended validation of an improved volume-of-fluid (VOF)-based method for 3D highly nonlinear, complex, and breaking free-surface problems in engineering is presented. The VOF interface-tracking scheme is implemented with a Navier–Stokes solver in a curvilinear coordinate system. The numerical procedure is advanced by implementing the scheme on moving Chimera (overset) grids for application in complex engineering problems. The overset grids are used to facilitate the flow simulations of complex geometries and arbitrary motions of objects and improve computational productivity by taking advantage of their flexibility. To solve the six degrees of freedom (6DOF) motions of a rigid body simultaneously, the nonlinear 6DOF motion equations in a vector form system are strongly combined with a flow solver. A collection of problems is carefully selected such that a large density ratio and complex cases under a wide variety of Froude numbers are presented with the aim of demonstrating the capabilities of the new enhanced method on a complicated moving overset grid system. The method is validated for a wide range of parameters. The results of different free-surface problems are presented. The numerical results are compared with numerical alternatives and experimental measurements, and accurate approximations and good agreement are obtained for complex flows.

1. Introduction

The simulation of two-fluid flow problems involving 3D complex interfacial structures and large deformations of a fluid free surface in the presence of fully or partially submerged bodies and the interaction between them is considerably challenging in computational fluid dynamics (CFD) methods. A few important examples of free-surface hydrodynamics are observed in environmental, naval, and ocean engineering, such as the evaluation of potential risks of the rupture of levees, dams, and reservoirs; interaction of extreme waves with floating structures, moving ships, and green water on decks; and problems in harbors and coastal areas. In such flows, the dynamics of the interface between fluids and its impact on structures play a dominant role; therefore, advances in numerical methods for achieving accurate simulations of highly nonlinear free-surface flows with complex geometries is an issue in practical applications, which is receiving increasing attention. Examples of such flows include sloshing in a tank [1–3], breaking waves and wave impact on offshore structures [4–8], free-surface flow around hydrofoils [9,10], dam-break flows [11–13], and water entry of objects [14–16]. Complex problems involving fluid kinematics and dynamics, interfacial structures, water impact loads, and trajectory predictions are 3D in nature. These applications are complex and highly nonlinear and are typically solved using several time steps; therefore, it is still difficult to obtain acceptable accuracy while using CFD methods for 3D complex flow simulations. Until now, only a limited number of numerical computations have achieved accurate predictions for violent free-surface 3D problems, and the CFD development for such complex flows is an area of ongoing research.

The authors' previous efforts have been focused on the development of numerical methods for the simulation of multiphase flows. The developed modeling methods include the homogeneous mixture incompressible and compressible flow models for cavitation, boiling, and water entry flows [17–20], multifluid models for compressible flows [21], and sharp interface methods for free-surface flows [22–25]. Each method is validated and adequate for several regimes of flows and has advantages and disadvantages. For example, the homogeneous mixture flow models are stable and simple to implement; however, large diffusion errors may occur in their long-term solutions. Multifluid models can consider the nonequilibrium dynamics between the phases at an interface; however, they are complex and expensive, and the correlations

* Corresponding author.
E-mail addresses: nguyenvantu@live.com (V.-T. Nguyen), wgpark@pusan.ac.kr (W.-G. Park).

https://doi.org/10.1016/j.apor.2018.01.007
0141-1187/

between the phases at an interface have not been generalized for several regimes. Several approaches have been developed in the case of sharp interface methods, such as the volume-of-fluid (VOF) method, level-set method, and interface sharpening approaches. With specific interface treatment in each method, the interface between phases is kept sharp, and diffusion errors are significantly reduced. Among these approaches, the VOF method is used widely as it exhibits good mass conservation. This study extends our previous work [24], wherein a numerical model was developed for free-surface flows. The dual-time pseudo-compressibility Navier–Stokes model combined with the VOF interface-tracking algorithm (NS/VOF) was implemented in a generalized curvilinear coordinate system, and it was used to simulate complex 3D problems. A numerical modeling approach with high resolution is required to obtain useful information on a flow field, particularly for water impact loads and interfacial structures. Therefore, a high-resolution shock-capturing monotonic upstream centered scheme for conservation laws (MUSCL) upwind scheme, which provides a high degree of accuracy, was used to prevent spurious oscillation at discontinuities. An interface-treatment method with good mass-conservation accuracy, referred to as the VOF/piecewise linear interface calculation (PLIC) algorithm for multiphase flows, was developed in a body-fitted, structured, and curvilinear grid.

The new contributions of this work include the developments realized in the numerical solver by integrating the Chimera grid scheme and the six degree of freedom (6DOF) motion equations of objects into the solver for extended applications to more complicated structures of interfaces and breaking waves around complex geometries in engineering. A grid system comprising sub-grids of various resolutions and a global grid are introduced to ensure grid communication within a framework suitable for multiblocks and parallel computations. The Chimera grids can be used to overcome the difficulty of grid generation in simulating complex configurations using a body-fitted general structured grid, which can be used to easily implement boundary conditions because of the body-aligned nature of the grid.

In our previous works, several numerical cases including dam break and simple water entry flow were simulated and validated [23,24]. In this work, we simulated several numerical cases to demonstrate that our framework is good at interface capturing and impact loading prediction on an overset system of multiple structured grids. The ability of the proposed method to handle problems involving complex geometries was confirmed by simulating a sphere and an oblique cylinder with 6DOF motions under unsteady viscous transient flow conditions using moving Chimera grids comprising a body-fitted grid and a curvilinear background grid. The results were in good agreement with the experimental data provided in literature. Furthermore, the proposed method was validated using free-surface flow and subsequently applied to breaking waves around a vertical hydrofoil and a submerged vehicle. This study demonstrates that the proposed method is suitable for two-fluid flow problems including free-surface flows, water-impact loads, breaking and collapsing of water waves, flow separations, and contact phenomena.

2. Incompressible flow model

The incompressible Navier–Stokes equations are solved based on a dual-time pseudo-compressibility method. The artificial compressibility (or pseudo-compressibility) method was originally introduced by Chorin for computing steady-state incompressible flows [26]. The extension of the pseudo-compressibility method to the calculation of unsteady incompressible flows based on pseudo-time marching compressible flow solvers has been used in several studies [23,24,27–29]. This method is a combination of the pseudo-compressibility and dual time-stepping procedures; it involves an iteration in pseudo-time within each physical time step, and it can obtain convergence toward the solution of incompressible flow equations. In this work, the pseudo-compressibility method is based on the basic principle of the classical methods; however, pseudo-time derivative terms are constructed by replacing true density by pseudo-density and pressure is calculated as a given function (referred to as the pseudo-law of state) of pseudo-density. The governing equations can be expressed as follows [23,24,27]:

$$\frac{\partial \tilde{\rho}}{\partial \tau} + \frac{\partial \rho u_j}{\partial x_j} = 0, \tag{1}$$

$$\frac{\partial \tilde{\rho} u_i}{\partial \tau} + \frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j}\right)\right] + \rho g_i, \tag{2}$$

where t is physical time, τ is pseudo-time, u is flow velocity, and p is pressure.

The pseudo-time derivatives use pseudo-density $\tilde{\rho}$, which is introduced in Eqs. (1) and (2), and the pressure field is calculated based on an additional pseudo-state equation [23],

$$p = \rho U_0^2 \ln\left(\frac{\tilde{\rho}}{\rho_\infty}\right) + p_\infty, \tag{3}$$

where the parameters are set in accordance with $U_0 = U_\infty$ or $U_0 = \sqrt{u^2 + v^2 + w^2}$, in which u, v, and w are the local values of the respective velocities obtained at a previous iteration step in pseudo-time.

In order to take the effects of turbulence into account, the filter-based model for turbulent closure is used. This turbulence model is a modified form of the $\mathrm{k}-\varepsilon$ turbulence model to improve the predictive capability of this RAN based engineering turbulence closure [17].

3. Numerical method

3.1. Numerical discretization scheme

Governing Eqs. (1) and (2) can be expressed in a vector form and discretized in a general structured grid using a class of the lower–upper symmetric Gauss–Seidel method [24]. An upwind difference scheme based on the characteristic information of the governing equations is used to compute convective flux derivatives. Hence, the flux Jacobian matrix is divided into two sub-vectors that are associated with non-negative and non-positive eigenvalues. The explicit part of the convective flux vector is discretized using a cell-centered finite-volume procedure, wherein extrapolated Riemann variables are obtained using the MUSCL procedure. For time discretization, physical time derivatives are approximated based on a second-order-accurate backward difference scheme, and the Euler's implicit finite-difference formula is used for pseudo-time derivatives. The pseudo-time step is determined in accordance with the local pseudo-time step, which is defined based on the largest eigenvalues of the system. At each step of physical time, a pseudo-time iterative procedure is applied such that pseudo terms approach zero upon convergence.

To proceed with the numerical procedure, the advection equation is solved using a known velocity field to update the density and viscosity fields in the next time step.

Scalar field α is passively advected with flow and satisfies the advection equation,

$$\frac{\partial \alpha}{\partial t} + u\frac{\partial \alpha}{\partial x} + v\frac{\partial \alpha}{\partial y} + w\frac{\partial \alpha}{\partial z} = 0. \tag{4}$$

Thus, the VOF tracking-interface method is employed to determine the volume fractions of the liquid and gas phases. The method