Computers & Fluids 103 (2014) 18–33

Contents lists available at ScienceDirect

Computers & Fluids

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

Numerical analysis of water impact forces using a dual-time pseudo-compressibility method and volume-of-fluid interface tracking algorithm

Van-Tu Nguyen [a], Duc-Thanh Vu [a], Warn-Gyu Park [a,*], Young-Rae Jung [b]

[a] School of Mechanical Engineering, Pusan National University, Busan 609-735, Republic of Korea
[b] Advanced Naval Technology Center, NSRDI, ADD, Jinhae P.O. Box 18, Changwon, Gyeongnam 645-600, Republic of Korea

ARTICLE INFO

Article history:
Received 26 March 2014
Received in revised form 18 June 2014
Accepted 8 July 2014
Available online 18 July 2014

Keywords:
Navier–Stokes
Pseudo-compressibility
VOF
Water impact
Free surface

ABSTRACT

An implicit algorithm based on a dual-time pseudo-compressibility method is developed to compute water impact forces on bodies. Flow fields of incompressible viscous fluids are solved using unsteady Reynolds-averaged Navier–Stokes equations. Pseudo-time derivatives are introduced into the equations to improve computational efficiency. A second-order volume-of-fluid interface tracking algorithm is developed in a generalized curvilinear coordinate system to track the interface between the two phases in the computational domain. A grid refinement study of the dam-break flow is performed as a validation, and the obtained solutions agreed well with the experimental data and with the results of other numerical simulations. Numerical analysis of water impact forces on a hemisphere, two cones, and a wedge through free falling in one degree of freedom is then performed. Free surface deformation, pressure coefficients, impact velocities, and vertical accelerations during impact are compared with available experimental data and theoretical results. Good agreement with these results is obtained.

1. Introduction

The water impact problem has been a topic of interest for many decades, especially in naval applications. A solid understanding of the physical characteristics of this phenomenon is essential for the effective design and usability of air-to-sea projectiles and torpedoes, as well as for numerous other applications. Early experimental studies on water-entry into different objects were performed and a series of splash photographs were reported by Worthington [1]. Later studies that addressed the characteristics of this phenomenon employed a variety of experimental, analytical, and numerical approaches. May [2] conducted experiments on water-entry to discuss the effect of different parameters on cavity formation. A study of the impact loads on seaplane floats during water landings, performed using the added mass concept and linearized free surface boundary conditions, was published by Von Karman [3]; in that study, a physical picture of the water-entry phenomenon was successfully captured. Analytical expressions based on several mathematical and physical assumptions were obtained by Miloh [4] in calculations of the slamming coefficient and wetting factor of a rigid spherical shape in a vertical and oblique water-entry. Numerical calculations were performed by Greenhow [5] by considering the deformations of a free surface caused by the forced motion of horizontal circular cylinders. A source panel method that assumes inviscid potential flow around water impact bodies was developed by Park et al. [6]. A pilot study that used this method reported the analysis of impact and ricochet behavior for bodies entering water at an arbitrary entry angle.

Numerical methods for multiphase flows in water impact problems have received increasing attention in recent years. However, a number of difficult challenges associated with computational fluid dynamics (CFD) methods remain for flows that involve strong fluid-body interactions, a large constituent density ratio, presence of discrete interfaces, and non-equilibrium interfacial dynamics. General methods for the flows base on the Reynolds-averaged Navier–Stokes equations [7–9] and good numerical simulations to describe the behavior of incompressible viscous fluids with a free surface can be obtained. Various methods have been developed to treat free surface deformation. The level-set method (LSM) and the volume-of-fluid method (VOF) are the most popular and are frequently used by researchers.

The LSM method was originally proposed by Osher and Sethian [10]. The interface is represented as a smooth function of the distance from the interface, and highly deformed interfaces can be treated. The method has been developed and applied to a wide

* Corresponding author. Tel.: +82 515102457.
E-mail address: wgpark@pusan.ac.kr (W.-G. Park).

http://dx.doi.org/10.1016/j.compfluid.2014.07.007
0045-7930/

range of problems, such as Rayleigh–Taylor instability [11], vortex motion [12], bubbles and drops [13], free surface motion, and interaction between free surface flow and rigid bodies [14].

The VOF technique is a good tool for numerical simulations of free surface flows. Different VOF interface tracking algorithms, such as the simple line interface calculation algorithm (SLIC) that was originally introduced by Noh and Woodward [15], the piecewise linear interface calculation algorithm (PLIC) that was proposed by Youngs [16], and the second-order volume-of-fluid method interface tracking algorithm known as the least squares volume-of-fluid interface reconstruction algorithm (LVIRA) and the efficient least squares VOF interface reconstruction algorithm (ELVIRA) of Puckett [17,18], have been developed. In the SLIC algorithm, the interface is reconstructed using a piecewise constant scheme and an advection algorithm, which are accurate to the first-order. The interface is parallel to one of the coordinate axes. Based on the reconstructed interface and the velocity field during each step of physical time, the values of the volume fractions are updated by solving the advection equation for either liquid or gas phase. Owing to the stair-stepped reconstruction, the SLIC computation breaks up the interface and a solution obtained by this method is commonly characterized by many small droplets disconnecting from the free surface; these artifacts are called "flotsam and jetsam". The volume fraction is rounded if it is less than zero or greater than one, which may result in some liquid loss during computation. The PLIC algorithm was significantly advanced by introducing a piecewise linear scheme. In this scheme, the interface is linearly approximated in each multi-fluid cell. A smoother and more realistic interface can be obtained and the flotsam and jetsam are completely eliminated when the SLIC is replaced by this scheme [16,18]. Notwithstanding, interface reconstruction algorithms yield an approximation, that is accurate only to the first-order. More advanced VOF interface reconstruction methods, e.g., the LVIRA and ELVIRA methods, use a second-order piecewise linear approximation instead of a first-order piecewise linear approximation for more realistic reconstruction of interface lines. The method has been used widely to compute shock refraction [19] and multiphase flows of incompressible fluids [20,21].

The objective of this paper is to develop an implicit algorithm for simulating the water impact problem. We focus particularly on modeling variable density flows with sharp interfaces and strong pressure gradients to avoid spurious oscillations. The dual-time pseudo-compressibility model [22,23] based on the unsteady Reynolds-averaged Navier–Stokes equations is used to solve the flow fields. The method uses pseudo-time iterations that are performed until the non-linear equations are satisfied at each physical time-step and has a number of advantages, such as direct coupling of the continuity and momentum equations in incompressible flow equations and the ability to use pseudo-time, preconditioned iterative methods for more efficient convergence of the subiterations. The incompressible flow equations are discretized on a general, structured grid using an upwind difference scheme and an implicit upwind non-MUSCL Total Variation Diminishing (TVD) algorithm [24] together with an appropriate limiter function to prevent the generation of spurious solutions near strong gradients. These approaches require deriving an eigensystem for Jacobian matrices. The LVIRA and the operator split advection algorithms [17,18] are implemented in a generalized curvilinear coordinate system to reconstruct the water–air interface from volume fractions in the computational domain. The performance of the proposed numerical procedure is examined by performing a grid refinement study of the dam-break flow and by comparing the present results with the experimental data as well as with results of other numerical simulations. The multiphase solver is then applied to simulations of the water impact of a hemisphere, two cones with different deadrise angles, and a symmetric wedge through free falling in one degree of freedom. The calculations focus on the water impact forces and motions of the rigid bodies in the presence of a free surface. Comparisons of the calculated results with the available experimental data and with other published results were performed and indicate good agreement between these different data sets.

2. Governing equations

The dual-time, pseudo-compressibility, homogeneous flow model is developed on the basis of the unsteady incompressible Reynolds-averaged Navier–Stokes equations. All dependent variables are non-dimensionalized using the free-stream conditions, such as free-stream velocity U_∞, free-stream density ρ_∞, free-stream viscosity μ_∞, and characteristic length of the body D. The governing equations can be expressed in generalized curvilinear coordinates as follows [25,26]:

$$\Gamma_e \frac{\partial \widehat{Q}}{\partial t} + \frac{\partial \widehat{Q}}{\partial \tau} + \frac{\partial \widehat{E}_j}{\partial \xi_j} - \frac{\partial \widehat{E}_j^v}{\partial \xi_j} = \widehat{S} \tag{1}$$

where the primitive solution variable, convective, viscous flux, and source vectors are written as follows:

$$\widehat{Q} = \frac{Q}{J} = \frac{1}{J}[\tilde{\rho}, \tilde{\rho}u, \tilde{\rho}v, \tilde{\rho}w]^T, \tag{2}$$

$$\widehat{E}_j = \frac{1}{J}\left[\rho_m U_j, \rho_m u U_j + \xi_{j,x}p, \rho_m v U_j + \xi_{j,y}p, \rho_m w U_j + \xi_{j,z}p\right]^T, \tag{3}$$

$$\widehat{E}_j^v = \frac{1}{J}\left[0, \xi_{j,k}\tau_{xk}, \xi_{j,k}\tau_{yk}, \xi_{j,k}\tau_{zk}\right]^T, \tag{4}$$

$$\widehat{S} = \frac{1}{J}\left[0, \rho_m\bar{g}_x, \rho_m\bar{g}_y, \rho_m\bar{g}_z\right]^T, \tag{5}$$

and t is the physical time, τ is the pseudo-time, and $\bar{g}_x$, $\bar{g}_y$ and $\bar{g}_z$ are the accelerations due to gravity and hydrodynamic forces in the x, y and z directions, respectively. The accelerations can be determined using the dynamic equation of the body.

The pseudo-time derivatives involve a pseudo-density $\tilde{\rho}$, and the pressure field is calculated on the basis of an additional pseudo-state equation [22]:

$$p = \rho_m U_0 \ln\left(\frac{\tilde{\rho}}{\rho_\infty}\right) + p_\infty \tag{6}$$

The mixture density and mixture viscosity are defined as follows:

$$\begin{aligned} \rho_m &= \alpha_l\rho_l + (1-\alpha_l)\rho_g \\ \mu_m &= \alpha_l\mu_l + (1-\alpha_l)\mu_g \end{aligned} \tag{7}$$

where α is the volume fraction, and the subscripts l and g denote liquid and gas phases, respectively.

The contravariant velocities are given as follows:

$$U_j = \xi_{j,t} + \xi_{j,x}u + \xi_{j,y}v + \xi_{j,z}w \tag{8}$$

The non-dimensionalized Reynolds number is defined as follows:

$$\mathrm{Re} = \frac{U_\infty D \rho_\infty}{\mu_\infty} \tag{9}$$

The matrix Γ_e is given as

$$\Gamma_e = \frac{\rho_m}{\tilde{\rho}}\begin{bmatrix} 0 & 0 & 0 & 0 \\ -u & 1 & 0 & 0 \\ -v & 0 & 1 & 0 \\ -w & 0 & 0 & 1 \end{bmatrix} \tag{10}$$