Ocean Engineering 130 (2017) 531–541

Contents lists available at ScienceDirect

Ocean Engineering

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

Simulation of the Kelvin–Helmholtz instability using a multi-liquid moving particle semi-implicit method

Kim, Kyung Sung [a], Kim, Moo-Hyun [b,*]

[a] Tongmyong University, 428 Shinsun-Ro, Namgu, Busan, South Korea
[b] Texas A & M University, 3136 TAMU, College Station, TX 77843, USA

ARTICLE INFO

Keywords:
Kelvin-Helmholtz instability
MPS
Particle method
Multiphase flow

ABSTRACT

In general, multi-fluid multi-interface problems are more complicated than single-phase problems and therefore require sophisticated and robust algorithm development. In this study, a moving particle semi-implicit (MPS) method based on the Lagrangian approach was extended to a multi-phase system with multiple interfaces. Accordingly, several new particle interaction models, including self-buoyancy-correction, surface tension, and interface boundary condition models, were introduced. Representative multi-liquid MPS sloshing simulation results were compared with corresponding experimental results with three liquids. The results agree well for both global behaviors and the much smaller-scale interfacial instability phenomenon called the Kelvin–Helmholtz instability (KHI). The validated multi-phase MPS method was subsequently applied to the classical Poiseuille problem of two fluids flowing with different velocities between two parallel plates The KHI is triggered when the destabilizing effect of a shear across an interface with a sufficient relative velocity overcomes the stabilizing effect of gravity and surface tension. Both viscosity and surface tension tend to limit the growth of KHIs when they are over certain values. For many possible combinations of density ratios and surface tensions producing the same Richardson-number, the body-force component plays a much more important role than the surface-tension component if the interfacial wave number is not very high.

1. Introduction

In general, in the case of multiple-liquid layers, more complicated interfacial phenomena can exist than in the single-liquid problem. Examples include internal waves, fluid mixing, and instabilities. To simulate the multi-liquid interaction problem numerically, several methodologies and special treatments have been introduced to both the Eulerian and Lagrangian approaches.

In the Eulerian approach with a grid system, the most representative approach for free-surface tracing is the volume-of-fluid (VOF) method (e.g., Chen, 2010). VOF is an advection scheme for tracking and locating the free surface and interface. However, it may not be robust when interfaces have fragmentations and large and/or sharp deformations. In this regard, a hybrid Eulerian–Lagrangian method, such as the particle-in-cell (PIC) method, has been suggested to use the Lagrangian frame for interface tracing on the Eulerian grid system. This is a well-structured concept to maintain interface sharpness with low-level computing resources; however, the coupling of the Eulerian and Lagrangian frames can still pose minor problems.

Conversely, a fully Lagrangian approach called the particle method can more straightforwardly handle interfaces with large and/or sharp deformations and fragmentations because the Lagrangian approach can trace every particle with physical properties. There have been two major developments in Lagrangian particle methods: smoothed particle hydrodynamics (SPH); (e.g., Monaghan, 1994) and moving particle semi-implicit (MPS) methods. The MPS method was introduced by Koshizuka and Oka (1996) for free-surface hydrodynamic problems and was subsequently developed with improved algorithms by Gotoh (2009) and Lee et al. (2011). The MPS method has also been developed to tackle multi-fluid problems. For example, Khayyer and Gotoh (2013) introduced a density averaging scheme to relieve numerical problems caused when the density ratio of the two fluids is very large. The surface tension model in the MPS method was introduced by Nomura et al. (2001) and the buoyancy correction model was introduced by Shirakawa et al. (2001) for multi-liquid systems. The surface-tension and buoyancy-correction models were further improved by Kim et al. (2014). There have also been developments with SPH to solve multiple-liquid interactions (e.g., Shadloo and Yildiz, 2011). In this paper, a newly developed MPS method for multi-liquid systems with more robust algorithms, including interface searching, buoyancy correction,

* Corresponding author.
E-mail address: m-kim3@tamu.edu (M.-H. Kim).

http://dx.doi.org/10.1016/j.oceaneng.2016.11.071
Received 20 May 2016; Received in revised form 21 September 2016; Accepted 30 November 2016

and surface tension models, is applied to investigate interfacial interactions and instability phenomena.

The newly developed MPS method for multi-liquid systems is first applied to a three-liquid-sloshing problem using forced oscillations of a rectangular tank at various frequencies. Using the Fourier transform, the measured interface elevations at the left wall of the tank were transferred to the frequency domain to obtain the response amplitude operator (RAO). The calculated free-surface and interface time histories and the RAOs for various oscillation frequencies and the experimental data of Molin et al. (2012) matched well. In both the multi-phase sloshing experiment and the presented simulations, saw-tooth-shaped small-scale interfacial waves were observed, which are related to the Kelvin–Helmholtz instability (KHI). The KHI is one form of interface instability phenomena in multi-phase flows. Barnea and Taitel (1993) studied KHIs for viscous and inviscid flows to find the criteria for KHI generation. Funada and Joseph (2001) investigated KHIs in a channel flow using a viscous potential flow analysis. Roediger et al. (2011) experimentally and numerically studied KHIs in gas sloshing. Scardovelli and Zaleski (1999) introduced a simulation of KHIs with multiple fixed-grid methods, including VOF, marker-and-cell (MAC), and level-set. Poiseulli's flow with interfacial KHIs of multi-phase fluids was investigated by Shakibaeinia and Jin (2012). Price (2008) and Shadloo and Yildiz (2011) further investigated the corresponding KHIs using SPH methods.

Here, the newly developed multi-phase MPS method is applied to simulate KHIs for the Poiseulli's flow case with two different fluids flowing in opposite directions. In this case, saw-tooth-shaped interfacial disturbances are generated due to the KHI, and either they grow further with strong roll-up motions or the growth is limited without roll-up, depending on the given fluid properties and conditions. In particular, the roles of the Richardson number (Ri), surface tension, and viscosity for the growth or limitation of the KHI are extensively investigated for various combinations of two-fluid properties.

2. Moving particle simulation

2.1. Governing equations

The continuity and Navier–Stoke's equations are employed as governing equations to solve the incompressible viscous fluid dynamics.

$$\frac{D\rho}{Dt} = 0 \tag{1}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla P + \upsilon\nabla^2\mathbf{u} + \sigma\kappa n + \mathbf{F} \tag{2}$$

where ρ is the density of the fluid, $\mathbf{u}$ is the particle velocity, P is the pressure, υ is the kinematic viscosity, ∇ is the gradient, t is time, σ is the surface tension coefficient, κ is the curvature of the interface, n is the normal vector to the interface, and $\mathbf{F}$ includes other external forces such as the gravitational force.

To simulate incompressible viscous flow, all terms of the differential operators need to be replaced with particle interaction models, which include gradient, divergence, and Laplacian models. Because the MPS method is based on the Lagrangian approach, the effect of particle interactions needs to be considered. Therefore, a kernel function, which represents the effect of neighboring particles with respect to their distance from the center particle, was suggested by Koshizuka and Oka (1996). In this study, the following kernel function was employed:

$$w(r) = \begin{cases} \left(1 - \frac{r}{r_e}\right)^3\left(1 + \frac{r}{r_e}\right)^3 & (0 \le r < r_e) \\ 0 & (r_e < r) \end{cases}, \tag{3}$$

where r is the distance between the particles and r_e is the effective range of the particle interaction. According to Eq. (3), if the particle distance, r, exceeds the effective range, the value of the kernel function becomes zero. In this study, the effective range was set to $2.1 \times l_o$, where l_0 is the initial particle distance.

2.2. Divergence, gradient, and Laplacian terms

To solve the governing equations, all differential operators, including the gradient, divergence, and Laplacian, need to be replaced with particle interaction models.

The divergence model represents the divergence of a physical quantity between two particles and is expressed as follows:

$$\langle\nabla\cdot\phi\rangle_i = \frac{d}{n^0}\sum_{j\neq i}\frac{(\phi_j - \phi_i)\cdot(\vec{r_j} - \vec{r_i})}{|\vec{r_j} - \vec{r_i}|^2} w(|\vec{r_j} - \vec{r_i}|). \tag{4}$$

The local weighted average of the gradient vectors between the center and neighboring particles can be represented by the gradient model. The gradient model can be expressed as follows:

$$\langle\nabla\phi\rangle_i = \frac{d}{n^0}\sum_{j\neq i}\left[\frac{(\phi_j - \phi_i)}{|\vec{r_j} - \vec{r_i}|^2}(\vec{r_j} - \vec{r_i}) w(|\vec{r_j} - \vec{r_i}|)\right]. \tag{5}$$

The Laplacian model, which represents the diffusion of a fluid, can be expressed as

$$\langle\nabla^2\phi\rangle_i = \frac{2d}{\lambda n^0}\sum_{j\neq i}[(\phi_j - \phi_i) w(|\vec{r_j} - \vec{r_i}|)], \tag{6}$$

where $\langle\rangle$ denotes the particle interaction model, ϕ is an arbitrary value for a physical quantity, d is the number of dimensions, $\vec{r}$ is the position vector of a particle, and n^0 is the initial particle number density.

The symbol λ in Eq. (6) is a parameter to ensure that the increase in the variance measured by the distribution of the particles is equal to the increase in the variance from the unsteady diffusion equation. The Laplacian model is used to calculate the viscous effects. It is measured from the viscosity of the center particle, which means that the weighted/averaged viscosity of the neighboring particles inside the effective radius is used. The parameter, λ, can be obtained from the following equation:

$$\lambda = \frac{\sum_{j\neq i}|\vec{r_j} - \vec{r_i}|^2 w(|\vec{r_j} - \vec{r_i}|)}{\sum_{j\neq i} w(|\vec{r_j} - \vec{r_i}|)}. \tag{7}$$

The particle number density can be calculated using a summation of the kernel function for particle i, shown in Eq. (5), which corresponds to the density of the fluid.

$$n_i = \sum_{j\neq i} w(|\vec{r_j} - \vec{r_i}|). \tag{8}$$

Since particle number density is the summation of neighboring particle with respect to kernel function, no special treatment is necessary even if particles are moved from original media zone to the different media zone. The density of each particle is applied for the calculation of pressure gradient.

2.3. Incompressibility algorithm

The incompressibility model in the MPS method is an adopting algorithm similar to the simplified marker-and-cell (SMAC) method in a grid-based CFD system. It consists of two stages, explicit and implicit. In the explicit stage, the intermediate velocity and position of the particle are predicted from the viscous and gravitational forces. In the MPS method, the particle number density is used as the density of the fluid and can be varied by the re-arrangement of the particles with the intermediate velocity. This changed particle number density may violate the continuity equation, which is one of the governing equations; therefore, an adjustment in the particle arrangement is required to satisfy the governing equations. This problem can be solved by