Comput. Methods Appl. Mech. Engrg. 229–232 (2012) 13–26

Contents lists available at SciVerse ScienceDirect

Comput. Methods Appl. Mech. Engrg.

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

# MPS mesh-free particle method for multiphase flows

Ahmad Shakibaeinia, Yee-Chung Jin *

*Faculty of Engineering and Applied Science, University of Regina, Regina, Canada*

ARTICLE INFO

*Article history:*
Received 19 May 2011
Received in revised form 21 February 2012
Accepted 19 March 2012
Available online 28 March 2012

*Keywords:*
Mesh-free particle method
MPS
Multiphase flow
Multi-viscosity flow
Multi density flow

ABSTRACT

By treating the multiphase system as a multi-density multi-viscosity fluid, a straightforward model has been proposed in this paper based on the Moving Particle Semi-implicit (MPS) mesh-free particle method for incompressible multiphase flow. The weakly-compressible MPS (WC-MPS) formulation (developed by the authors) is used to solve a single set of equations for all of the phases. In the model, the multiphase forces are introduced in a straightforward way. Dealing with multi-viscosity systems, different methods for defining the viscosity, by which particles of different phases interact, is examined. To evaluate the accuracy of each of these methods, a stratified multi-viscosity Poiseuille flow test case is used and model results are compared with the analytical solution. The results show that selection of this interaction viscosity has an important role in the accuracy of the results. The model is then validated and applied to two basic hydrodynamic instability cases (Rayleigh–Taylor and Kelvin–Helmholtz instabilities). The results are then compared to a mesh-based model (with volume of fluid interface tracing method) to examine the ability of the model to deal with the multi-density systems. Comparisons show the model has reasonable accuracy. The results of this work offer the potential of modeling of multiphase incompressible immiscible systems in an extensive range of conditions using MPS.



## 1. Introduction

From the point of view of the computational frame, the numerical methods for handling multiphase systems can be generally classified into three main groups. The first group is the fully Eulerian approaches such as Volume-Of-Fluid (VOF) method [1] which is characterized by a mesh that is either stationary or adaptive to the interfaces. The second group is the hybrid Eulerian–Lagrangian approaches such as Particle-In-Cell (PIC) method [2] in which a Lagrangian frame is used for interface tracing where the field variables are computed on a Eulerian frame. The last group is the fully Lagrangian approaches such as Smooth Particle Hydrodynamics (SPH) [3–5], Dissipative Particle Dynamics (DPD) [6], and Moving Particle Semi-implicit (MPS) [7,8] methods.

Eulerian methods with a stationary mesh have problems with maintaining a sharp interface between two phases while those with a deforming mesh have problems with adaptation of the mesh to follow the highly deformed and fragmented interfaces [9]. Hybrid Eulerian–Lagrangian methods have difficulties with coupling of the Eulerian and Lagrangian frames, as well as numerical inaccuracy that can occur due to the coupling process.

The mesh-free particle (Lagrangian) methods belong to the group of fully Lagrangian approaches and are known to be capable of dealing with any interface deformations and fragmentations. One of the most useful characteristics of the particle methods in multiphase systems is that the behavior of phases arises from interaction of the particles alone and is irrelevant to the position of interface. The Smoothed-Particle Hydrodynamics (SPH) method was originally developed for astrophysical applications and was later expanded for applications in solid and fluid mechanics. It works by dividing the fluid into a set of discrete elements, referred as particles and is based on integral representation of quantities and spatial derivatives. A number of SPH models have been developed for multiphase systems (e.g. [10–14]). Moving Particle Semi-implicit method (MPS), which is based on Taylor series expansion, was originally developed for fluid mechanics applications by Koshizuka et al. [7] and Koshizuka and Oka [8]. The MPS method is similar to the SPH method. The fundamental difference of MPS and SPH is that the MPS method of spatial derivative approximation is solely based on a local weighted averaging process without incorporating the gradient of a kernel function.

Since its development, MPS has been applied to a wide range of fluid mechanics applications, including modeling of dam breaks [7], breaking waves [15], jet breakup [16], hydraulic jump formation [17], and flow over the sills and in trenches [18]. The sub particle-scale (SPS) turbulence model was proposed by Gotoh et al. [19] for simulation of turbulent flow using MPS. Khayyer and Gotoh [20] and Shakibainia and Jin [17] showed that the original MPS formula does not guarantee the conservation of linear and angular momentum and proposed some modification of the MPS pressure gradient calculation as well as the deployed kernel function. The artificial

* Corresponding author.
*E-mail address:* yee-chung.jin@uregina.ca (Y.-C. Jin).

0045-7825/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cma.2012.03.013

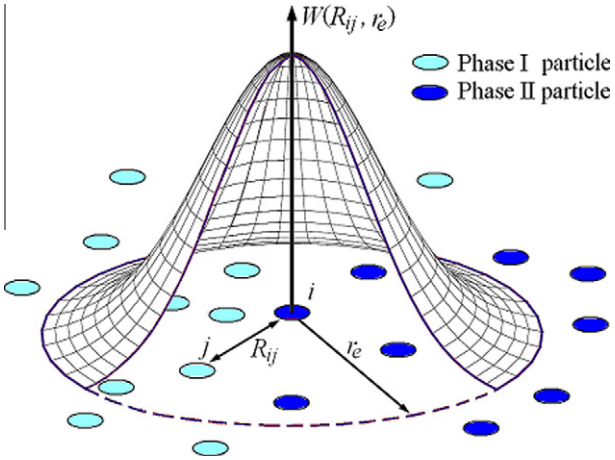


**Fig. 1.** Particle interaction conceptual model.

pressure fluctuation in the MPS was addressed by Khayyer and Gotoh [20] and Shakibaeinia and Jin [17]. A multiphase MPS method was introduced by Gotoh and Fredsøe [21] for solid–liquid and by Ikari et al. [22] for gas–liquid two phase flow by solving two sets of equations for different phases and adding the interaction force to the momentum equation. A hybrid particle-mesh method was proposed by Liu et al. [9] for incompressible viscous multiphase flow, in which one phase is represented by the moving particles and the other one is defined by a stationary mesh. Shakibaeinia and Jin [17] proposed the weakly compressible MPS method (WC-MPS) for modeling of the incompressible fluids and showed that this method not only improves the MPS artificial fluctuations but also slightly increases the model efficiency compare to standard MPS (fully compressible MPS).

This paper then aims to develop a new model based on WC-MPS for multiphase incompressible systems. Here, a straightforward model in which the multiphase system is treated as a multi-viscosity multi-density fluid is employed, and only a single set of equations is solved for the whole of the flow field. In the present method, the multiphase forces are automatically calculated by retaining all terms of the governing equations, such as the viscosity gradients. A fractional steps algorithm is used for time splitting. The pressure is calculated using the WC-MPS method. Particle motion is modified using a pair-wise collision algorithm. For calculation of the shear stress between two different-phase particles, an interaction viscosity must be defined. In this study, different methods of defining of the interaction viscosity are evaluated by applying the model to the stratified single-density, multi-viscosity Poiseuille flow test case and comparing the results with the analytical solution. The final model is validated using two of the basic hydrodynamic instabilities, the Rayleigh–Taylor (multi-viscosity, multi-density) and Kelvin–Helmholtz (inviscid, multi-density) instability cases. The results are then compared to a mesh-based VOF model (Fluent6.3 software is used for this purpose).

## 2. Governing equations

The multiphase system is treated as a multi-density multi-viscosity fluid, hence a single set of equations can be used for all of the phases. The governing equation is expressed by the continuity and momentum equations. In the Lagrangian system, they can be written as:

Continuity:

$$\frac{D\rho}{Dt} + \rho(\nabla \cdot \mathbf{u}) = 0. \tag{1}$$

Momentum:

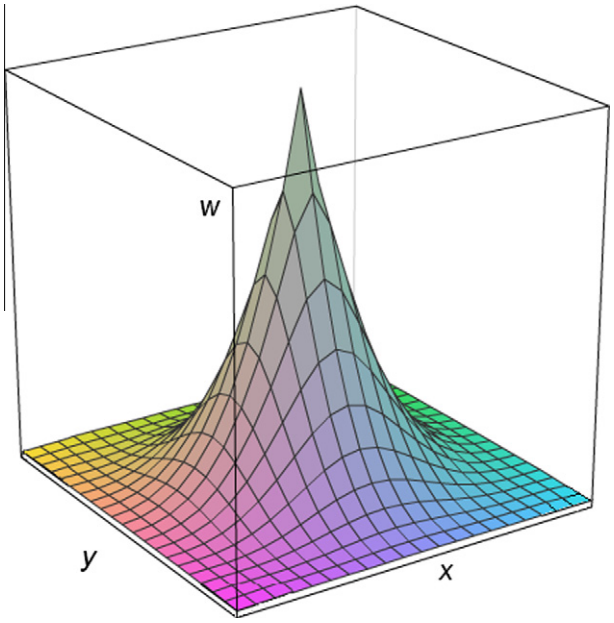


**Fig. 2.** Third order polynomial spiky kernel function.

$$\rho\frac{D\mathbf{u}}{Dt} = -\nabla p + \nabla(\mu\nabla\cdot\mathbf{u}) + \mathbf{f}, \tag{2}$$

where $\mathbf{u}$ is the flow velocity, $\rho$ is the density, $p$ is the pressure , $\mu$ is the dynamic viscosity, and $\mathbf{f}$ represents the body forces (per unit volume). Needless to say that, in Lagrangian systems, there is no convective acceleration term in the left hand side of the momentum equation. The motion of each particle is simply calculated by $D\mathbf{r}/Dt = \mathbf{u}$, ($\mathbf{r}$ being the position vector). Spatial derivatives in the governing equations need to be approximated on the particle domain. For calculation of spatial derivatives as well as interpolation of physical quantities, the MPS particle interaction model is used.

## 3. Numerical method

### 3.1. MPS appoximations

As a mesh-free method MPS interaction is built on a set of disordered points in a continuum without a grid or mesh. The particle $i$, interacts with other particles in its vicinity, $j$, covered with a kernel (weight) function $W(r_{ij}, r_e)$, where $r_{ij} = |\mathbf{r}_j - \mathbf{r}_i|$ is the distance between particle $i$ and $j$ (either in the same phase or different phases) and $r_e$ is the radius of the interaction area around each particle (Fig. 1). The kernel function is considered to be a smoothing function of physical quantities around particles. This article employs a third order polynomial spiky kernel function (Fig. 2) proposed by Shakibaeinia and Jin [17] and written as:

$$W(r, r_e) = \begin{cases} (1 - r/r_e)^3 & 0 \leqslant r/r_e < 1, \\ 0 & r/r_e \geqslant 1. \end{cases} \tag{3}$$

A particle interacts with a finite number of adjacent particles located within a distance of $r_e$. A dimensionless parameter, namely, particle number density, $n$, defined in Eq. (4) represents the density of particles around a specific particle. Real fluid density, $\rho$, is defined by Eq. (5).

$$\langle n\rangle_i = \sum_{j\neq i} W(r_{ij}, r_e), \tag{4}$$

$$\langle\rho\rangle_i = \frac{\sum_{j\neq i} m_j W(r_{ij}, r_e)}{\int_V W(r, r_e)dv}, \tag{5}$$