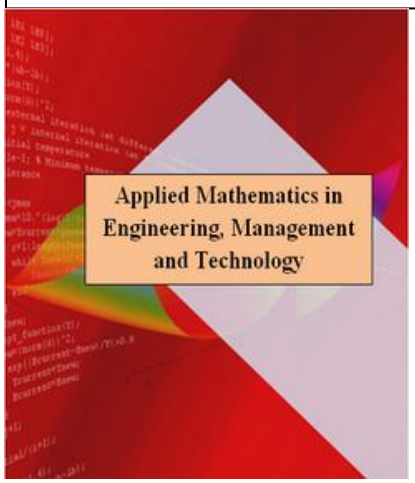
Applied mathematics in Engineering, Management and Technology 3(4) 2015:277-287
www.amiemt-journal.com

Numerical Simulation of Multiphase Flows Using SPH Projection Method

Massoud Rezavand
Department of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran
mrezavand@srbiau.ac.ir
Mohammad Taeibi-Rahni
Department of Aerospace Engineering, Sharif University of Technology, Tehran, Iran
taeibi@sharif.edu



Abstract:

The Lagrangian nature of smoothed particle hydrodynamics (SPH) lets track the interface between two different fluids without any need to an interface capturing algorithm. This characteristic has nominated SPH as an attractive method for simulation of complex flows. In present study a multiphase SPH method is presented for Newtonian multiphase flows with a special consideration of surface tension effects at the interface. Performance of the proposed method is verified against exact solution of Couette flow which is a well-known test case. Finally, two challenging two-phase problems; namely, Rayleigh-Taylor instability and bubble rising in a viscous fluid are widely investigated. Results are compared with analytical and other numerical data available in literature and good agreement is achieved.

Keywords: Smoothed Particle Hydrodynamics, Multiphase flows, Projection method

1. Introduction

Multiphase flows are problems in which different fluid phases are simultaneously present. These flows happen frequently in many natural and industrial processes. Because of their important role in prediction of natural phenomena and design of industrial processes, multiphase flows have been widely studied in many experimental and numerical researches. Most numerical investigations in this field have been performed by Eulerian mesh-based methods. Although these methods have been proved to be accurate and have had reliable results in many problems, they need to be more investigated in cases dealing with highly disordered interfaces. Heterogeneity in density and other hydrodynamic properties at the interface make it hard to handle multiphase flows. Hence, in order to accurate numerical simulations, special attention about the evolution of the interfaces is required. In the field of multiphase flows, mesh-dependent numerical methods require a specific algorithm to track the interface accurately (in addition to governing equations of fluid flow). Volume of fluid (VOF) [1] and level set (LS) methods [2] are the well-known algorithms for interface tracking for simulation of multiphase flows.

Lagrangian particle-based methods are alternatives to Eulerian mesh-based methods for simulation of interfacial flows. One of the more advanced methods of this category is smoothed particle hydrodynamics (SPH), which is a fully Lagrangian meshless method, which does not need to any computational grid. It replaces fluid by a finite number of fluid particles and the flow is described by evolution of these particles. Consequently, the Lagrangian nature of SPH lets to track the interface without any need for a specific algorithm. This advantage nominates SPH as an important candidate for simulation of multiphase flows.

Monaghan and Kocharyan [3] simulated multiphase problems with low density ratios using SPH for the first time. When Colagrossi and Landrini [4] used the method proposed in [3] for air-water problems (density ratio about 1:1000) they found that the method was unstable and gave very poor results. The reason for these instabilities is that when the integral interpolant is performed over two particles with severe density gradients, singularities occur in the solution. To remedy these singularities Colagrossi and Landrini [4] proposed a new form of particle evolution equation which enhanced the stability of the method. They also used a periodic re-initialization of the density according to Moving-Least-Square (MLS) density correction [5]. MLS is a first-order accurate interpolation method for irregularly distributed particles which requires computation of a special MLS kernel and thus, enforces additional computational costs. Furthermore, a general correction to the standard

form of the artificial viscosity used in SPH and a smoothing scheme for velocity field based on XSPH formalism [6] has been used in [4]. XSPH is a velocity correction method in which, velocities of neighboring particles are used in order to smooth the velocity field.

Hu and Adams [7, 8] with the concept of particle number density represented the spatial derivatives of the field variables as a smoothing function in which the neighboring particles participate just in specific volume, but not in density. In this way, the influence of severe density gradients is alleviated. Grenier et al. [9] presented a Hamiltonian formulation as an extension of the one proposed in [4] and similar to one presented in [7]. Their method also can be adopted for free surface problems. The proposed method is based on use of a Shepard kernel for density calculation, which enhances the accuracy of the method and requires knowing a spatial distribution of the particle volumes (enforcing computational costs). They have also used a kind of repulsive force at the interface of different phases in problems where surface tension effects were negligible.

Monaghan and Rafiee [10] presented a simple method for multiphase problems with high density ratios, using a repulsive force between the particles of different phases. In their method, when interacting particles are of two different phases, pressure is increased by a repulsive force which acts on the interface. This method is also applicable for surface tension problems and has been validated against various test cases. Szewc et al. [11] simulated the motion of an air bubble in a viscous fluid with a WCSPH formulation similar to the one proposed in [7]. They have used a repulsive force at interface similar to one used in [9] and [10]. Zainali et al. [12] presented a new formulation in ISPH approach for multiphase flows with high density ratios based on corrective SPH gradient formulation which they have introduced in a previous work [12]. In their formulation, the concept of particle number density is used in discretization of the functions and their derivatives. In their discretized formulations, density and viscosity are smoothed by a Harmonic mean interpolation. More recently, Chen et al. [13] have presented a new multiphase WCSPH model with the assumption that pressure and space are continuous across the interface. In their method, when the concerned particle is near the interface, all neighboring particles within the support domain belonging to the other phase are regarded as particles of the same phase and only the information of position, velocity, volume and pressure of these particles are involved in solution of acceleration and density of the concerned particle. They have also presented a new corrected density re-initialization algorithm in order to avoid the pressure instability caused by free movement of SPH fluid particles. This new algorithm leads to better computational accuracy with smoother pressure field. In their simulations, a cut-off value of particles density is also used in order to avoid negative pressure.

In this work, a SPH multiphase model is employed based on the projection method described in [14]. Details of the projection method and the CSF model for surface tension are described. The developed algorithm is verified using exact solution of a well-known test case. Two multiphase test cases are investigated numerically and results are compared against available data in literature.

2. SPH Formulations

2.1. Basics

The main concept of SPH method is an interpolation method which represents any arbitrary function in terms of its values at a number of points (Particles) [15]. The integral representation of an arbitrary function *f(x)* is defined as:

$$f(x) = \int_{\Omega} f(x')W(x - x', h)dx', \tag{1}$$

where, Ω is the entire computational domain, W is interpolating kernel function, and h is the effective length of the kernel function. The kernel function must satisfy the following two properties:

$$\int_{W} W(x - x')dx' = 1, \tag{2}$$

$$\lim_{h\to 0} W(x - x', h) = \delta(x - x'), \tag{3}$$

where, δ is the Dirac delta function.

In SPH method, fluid is represented by a finite number of particles which are the interpolating points (Fig. 1). In order to approximate the properties of the fluid on these points, the integral interpolation presented in Eq. (1), is replaced by following summation:

$$f(x) = \sum_{b=1}^{N} \frac{m(x_b)}{\rho(x_b)} f(x_b)W(x - x_b, h), \tag{4}$$

where, m is mass, ρ is density, and subscript b is the label of neighboring particles. This summation is restricted to N particles lieing within a circle with a radius of kh and is centered at x, as shown in Fig. 1. In addition, k is a coefficient that depends on the order of kernel and for a 3th, 4th, and 5th order kernel, its value is 2, 2.5, and 3, respectively. Many examples of various kernels are presented in literature and the kernel function used here is a quintic spline [16] which has the form:

$$W(x-x_b,h)=\begin{cases} n_d\left[(3-q)^5-6(2-q)^5+15(1-q)^5\right], & q<1 \\ n_d\left[(3-q)^5-6(2-q)^5\right], & 1\le q<2 \\ n_d(3-q)^5, & 2\le q<3 \\ 0. & q\ge 3 \end{cases} \tag{5}$$

where, $q=|x-x_b|/h$ and $n_d=\dfrac{7}{478\pi h^2}$.

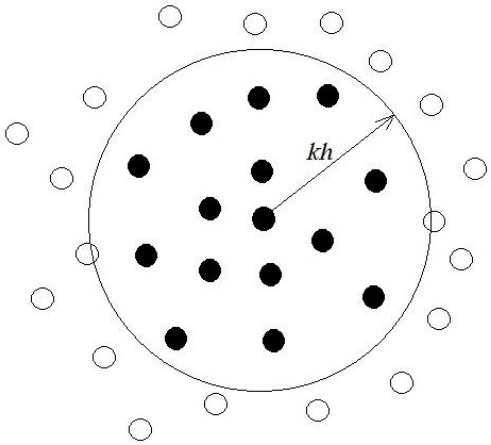


Fig. 1. Representation of fluid by Particles in SPH method and the kernel support for a particle.

2.2. Governing Equations

Mass and momentum conservation equations are considered as governing equations. For an incompressible flow, these equations can be written in Lagrangian form respectively as:

$$\nabla.\boldsymbol{u}=0 \tag{6}$$

$$\rho\frac{D\boldsymbol{u}}{Dt}=-\nabla p+\mu\nabla^2\boldsymbol{u}+\rho\boldsymbol{g}+\boldsymbol{F}_s \tag{7}$$

where, $\boldsymbol{u}$ is velocity, p is pressure, μ is dynamic viscosity, ρ is density, $\boldsymbol{g}$ is gravitational acceleration and $\boldsymbol{F}_s$ is volumetric surface tension. In order to non-dimensionalize the momentum conservation equation, following dimensionless parameters are introduced:

$$t^*=\frac{t}{D^{\frac{1}{2}}g^{\frac{-1}{2}}},\qquad x^*=\frac{x}{D},\qquad u^*=\frac{u}{(gD)^{\frac{1}{2}}},\qquad p^*=\frac{p}{\rho_H gD},\qquad \rho^*=\frac{\rho}{\rho_H}, \tag{8}$$

where, D is diameter of bubble, κ is curvature of the interface between two different fluids, and subscript H denotes the heavier fluid. Introducing these dimensionless parameters into Eq. (7), dimensionless Reynolds and Bond numbers are obtained as:

$$Re=\frac{\rho_H g^{\frac{1}{2}}D^{\frac{3}{2}}}{\mu_H},\qquad Bo=\frac{\rho_H gD^2}{\sigma}. \tag{9}$$

2.3. Incompressibility Constraint in SPH

In SPH, there are two general approaches to impose the incompressibility constraint: weakly compressible SPH (WCSPH) and incompressible SPH (ISPH). WCSPH has been more common and was first proposed by Monaghan [17]. In this approach, the fluid is assumed to be a little compressible and pressure is calculated from an appropriate equation of state. One of the commonly used equations of state in WCSPH approach is [17]:

$$P_a = \frac{\rho_{0_a} c_a^2}{\gamma}\left[\left(\frac{\rho_a}{\rho_{0a}}\right)^{\gamma} - 1\right], \tag{10}$$

where, ρ_0 is the reference density, c_a is the speed of sound and γ is the polytropic coefficient which is 1.4 for air and is 7 for liquid in an air-water multiphase problem.

In order to ensure the incompressibility constraint (Much number, Ma≈0.1) in the liquid phase, the value of speed of sound in Eq. (10) must be at least 10 times greater than the maximum velocity of the fluid and therefore the density fluctuations will be less than 1%. On the other hand, since the CFL condition for stability of the method is based on the speed of sound, consequently numerical stability will be achieved in smaller time steps leading to more computational costs.

ISPH is the more recent approach and has been proposed by Cummins and Rudman [18] for the first time. In this approach, the incompressibility constraint is enforced by projection method which has been widely used in Eulerian mesh-based systems [19, 20]. In ISPH approach, based on projection method, there are different algorithms to impose the incompressibility condition in which the incompressibility criteria are different. From a theoretical point of view, in an incompressible flow, both variations of density and divergence of velocity field are zero, but in numerical methods satisfying one of these conditions does not guarantee satisfaction of the other one and this is the reason for proposing different methods based on incompressibility criteria.

In the original projection method proposed by Cummins and Rudman [18], the divergence of velocity field is considered as the source term for Poisson pressure equation and the satisfaction of the continuity equation is achieved by variations of velocity field being zero; Therefore, it is called divergence-free method. This algorithm whose detailed steps are presented in [14] is used for all simulations of the present work.

2.4. The SPH Projection Algorithm

First, the intermediate position of the particles are predicted using the current time-step velocities

$$\boldsymbol{r}_a^* = \boldsymbol{r}_a^n + \Delta t(\boldsymbol{u}_a^n), \tag{11}$$

where, $\boldsymbol{r}$ is position, $\boldsymbol{u}$ is velocity field, Δt is time-step, and the superscripts * and *n* denote intermediate time-step and current time-step, respectively. At this intermediate position, the intermediate velocities are calculated by solving the momentum equation excluding the pressure gradient, which is:

$$\boldsymbol{u}_a^* = \boldsymbol{u}_a^n + \left(\boldsymbol{g} + \mu\nabla^2\boldsymbol{u} + \boldsymbol{F}_s\right)\Delta t, \tag{12}$$

where, $\boldsymbol{g}$ is the gravitational acceleration, μ is the dynamic viscosity coefficient, and $\boldsymbol{F}_s$ is the surface tension force. In order to discretize the viscous term in Eq. (12), the formulation originally proposed by Cleary and Monaghan [21] is used, which conserves both linear and angular momentum as:

$$\left(\frac{\mu}{\rho}\nabla^2\boldsymbol{u}_a\right) = \sum_b m_b\left(\frac{1}{\rho_b\rho_a}\frac{4\mu_a\mu_a}{\mu_a+\mu_b}\frac{\boldsymbol{u}_{ab}\boldsymbol{r}_{ab}}{|\boldsymbol{r}_{ab}|^2+\eta^2}\right)\nabla_a W_{ab}, \tag{13}$$

where, $\boldsymbol{u}_{ab}$ is the difference between the velocity of particle *a* and the neighboring particle *b* $(\boldsymbol{u}_{ab} = \boldsymbol{u}_a - \boldsymbol{u}_b)$, $\boldsymbol{r}_{ab}$ is the displacement vector between particle *a* and particle *b* $(\boldsymbol{r}_{ab} = \boldsymbol{r}_a - \boldsymbol{r}_b)$, and η is a small number for avoiding denominator being zero.

The pressure at time-step *n+1* can be then calculated by solving the following Poisson pressure equation:

$$\nabla\cdot\left(\frac{1}{\rho_a}\nabla p_a^{n+1}\right) = \frac{1}{\Delta t}\nabla\cdot(\boldsymbol{u}_a^*), \tag{14}$$

where *p* is pressure. The left hand side of Eq. (14) can be approximated by the formulation proposed by Cummins and Rudman [18], as:

$$\nabla\cdot\left(\frac{1}{\rho_a}\nabla p_a^{n+1}\right) = \sum_b \frac{m_b}{\rho_b}\left(\frac{4}{\rho_a+\rho_b}\right)\frac{(p_a-p_b)\boldsymbol{r}_{ab}\cdot\nabla_a W_{ab}}{|\boldsymbol{r}_{ab}|^2+\eta^2}. \tag{15}$$

The velocity field at time-step *n+1* can be obtained from correcting the predicted velocities by the pressure field calculated from Eq. (14), as:

$$\boldsymbol{u}_a^{n+1} = \boldsymbol{u}_a^{*} - \Delta t \frac{1}{\rho} \nabla p^{n+1}. \tag{16}$$

Finally, all particles are moved to their new position by the following scheme:

$$\boldsymbol{r}_a^{n+1} = \boldsymbol{r}_a^{n} + \frac{1}{2}(\boldsymbol{u}_a^{n+1} + \boldsymbol{u}_a^{n})\Delta t. \tag{17}$$

This algorithm will be unstable and leads to particle clustering in high Reynolds numbers. In order to enhance the stability features of the algorithm, Xu et al. [22] introduced a method in which particles are replaced slightly from streamlines at the end of each time step. Then, hydrodynamic properties are corrected by a Taylor expansion interpolation for new positions.

2.5. Surface Tension

Surface tension is caused by cohesive forces between liquid molecules and acts as a net force at the interface region toward the liquid. A special attention must be paid in order to accurately take surface tension force into account. In SPH, there are two general classes of methods used. The methods in the first class are microscopic and use the inter-particle interactions in order to implement the inter-particle cohesive forces. Nugent and Pasch [23] have used the van der Waals (vdW) inter-particle attraction potentials to simulate stable drop configurations. The cohesive vdW used in their work leads to an attractive central force acting between particles. These forces vanish outside fringes of the interface and they cancel each other at the interface band. In their method, the interaction range of attractive forces needs to be larger than the one used for all other forces in SPH formulations. Although implementation of microscopic inter-particle potentials is straightforward, some difficulties arise when using these methods. One difficulty is that the calculated surface tension requires being calibrated. In addition, surface tension depends on the resolution of the solution such that when resolution is increased, it does not converge to a fixed value [24]. On the other hand, Breinlinger et al. [25] pointed out that since the interaction between particles are not readily available and must be fitted, using a macroscopic method, in which the surface tension and contact angles are given as input parameters, is advantageous.

The second class belongs to the models which use a macroscopic point of view. These models are based on the continuum surface force (CSF) model originally proposed by Brackbill et al. [26]. In CSF model, a color function is attributed to each fluid and the interface is modeled as a transition region in which the color function varies smoothly from its value in one fluid to its value in the other one (Fig. 2).

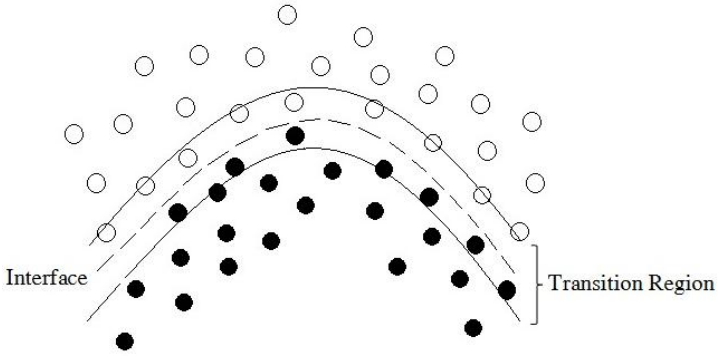


Fig. 2. The transition region between different fluids and the position of interface.

Morris [27] have used this approach in SPH for the first time. He presented various formulations in order to implement surface tension for multiphase problems with low density and viscosity ratios but without free surfaces. Adami et al. [24] pointed out that all formulations presented in [27] are based on a smoothed color function leading to incorrect curvatures for interface. One reason for the calculation of incorrect divergences for unit normals (the curvature) is that in the transition band, full support of kernel function is not satisfied. In order to remedy aforementioned problems, Hu and Adams [7] proposed the continuum surface stress (CSS) model. In this model a sharp color function with a discontinuity at the interface is used and the curvature of the interface is calculated by a surface stress tensor which depends on the gradient of the color function. In a recent work, Zainali et al. [12] have mentioned that in their ISPH simulations for multiphase problem, in contrary with Hu

and Adams [7], they have obtained more accurate results using CSF method (compared with CSS) for surface tension. In the present work, the CSF method proposed by Morris [27] is used.

In CSF method, the surface tension is replaced by a volumetric force as:

$$\boldsymbol{F}_s = \boldsymbol{f}_s \delta_s, \tag{18}$$

where, δ_s is the surface delta function which reaches its maximum value at interface. $\boldsymbol{f}_s$ is a surface force which is defined as:

$$\boldsymbol{f}_s = \sigma\kappa\hat{n} + \nabla_s \sigma, \tag{19}$$

where, σ is surface tension coefficient, $\hat{n}$ is the unit normal vector to the interface, κ is the curvature of interface and ∇_s is the surface gradient. The second term in Eq. (19) acts tangentially on interface and forces fluid from the regions with low surface tension to the one with high surface tension. In [27], surface tension is considered constant and thus the second term of Eq. (19) is neglected. The unit normal vector to the interface is calculated as:

$$\hat{n} = \frac{\nabla C}{|\nabla C|}, \tag{20}$$

where, C is the color function as:

$$C = \begin{cases} 1, & \text{in heavier fluid,} \\ 0, & \text{in lighter fluid.} \end{cases} \tag{21}$$

The curvature of the interface is calculated as:

$$\kappa = -\nabla \cdot \hat{n}. \tag{22}$$

Various functions can be employed as δ_s which has to satisfy the normalization condition. In this work, this function is defined as:

$$\delta_s = |n| = |\nabla C|. \tag{23}$$

In order to accurate predictions for surface tension, the curvature of the interface must be calculated; hence the unit normal vectors to the interface and their divergence must be calculated. A discretized form for calculation of the unit normal is:

$$n_a = \sum_b \frac{m_b}{\rho_b} C_b \nabla_a W_{ab}, \tag{24}$$

where, C_b is the color index of the neighboring particle, b. In order to achieve more accurate calculations for unit normal vector, color function can be smoothed by the concept of particle number density as:

$$\tilde{C}_a = \sum_b \frac{m_b}{\rho_b} C_b W_{ab} = \frac{\sum_b C_b W_{ab}}{\Theta_a}, \tag{25}$$

where, Θ_a in the particle number density, which is approximately equal to the inverse of the volume of the corresponding particle a and is defined as:

$$\Theta_a = \sum_b W_{ab} = \frac{\rho_a}{m_a}. \tag{26}$$

Using Θ_a, a more accurate formulation can be written as:

$$n_a = \sum_b \frac{m_b}{\rho_b} (\tilde{C}_b - \tilde{C}_a) \nabla_a W_{ab}. \tag{27}$$

The curvature of interface can be obtained by calculation of the divergence of the unit normal vector as:

$$(\nabla \cdot \hat{n})_a = \sum_b \frac{m_b}{\rho_b} \hat{n}_b \cdot \nabla_a W_{ab}, \tag{28}$$

and according to [15], a more accurate calculation can be obtained using the difference of unit normal of particle a and unit normal of the neighboring particle b as:

$$(\nabla \cdot \hat{n})_a = \sum_b \frac{m_b}{\rho_b}(\hat{n}_b - \hat{n}_a) \cdot \nabla_a W_{ab}. \tag{29}$$

It is noteworthy that unit normal vectors at the fringes of interface might be incorrect and lead to incorrect calculations of the curvature of interface. Hence, a constraint is needed for unit normals to be reliable. Morris [16] has proposed a constraint as:

$$|n_a| > \frac{\varepsilon}{h}, \tag{30}$$

where, ε is a constant, which was set to 0.01 in [16] and 0.08 in [12]. Normals satisfying this condition are considered as reliable normal to participate in calculation of the curvature. Finally, the calculated surface tension can be imposed in momentum equation as:

$$(F_s)_a = -\frac{\sigma}{\rho_a}(\nabla \cdot \hat{n})_a n_a. \tag{31}$$

3. Algorithm Verification

To verify performance of the present algorithm, numerical results are compared with exact solution of the well-known Couette flow test case. The analytical solution of this test case is presented in [28] as:

$$U = -V(1-2y) + \frac{4V}{\pi} \sum_{j=2,4,6,\ldots}^{\infty} \sin(j\pi y) \exp(-j^2 \nu \pi^2 t) \tag{32}$$

where, U is the x component of velocity field, V is the constant horizontal velocity of the upper ($+V$) and lower ($-V$) walls, y is vertical component of position, ν is kinematic viscosity coefficient and t is time. According to [28] value of V and ν are set as V=±1 m/s and $\nu = 0.0321\ m^2.s^{-1}$. After a finite amount of time, the flow reaches a steady state condition. Results are compared in Fig. 3 for $t = 1.14\ s$ in transient condition. It is observed that there is good agreement between analytical results and present SPH simulations.

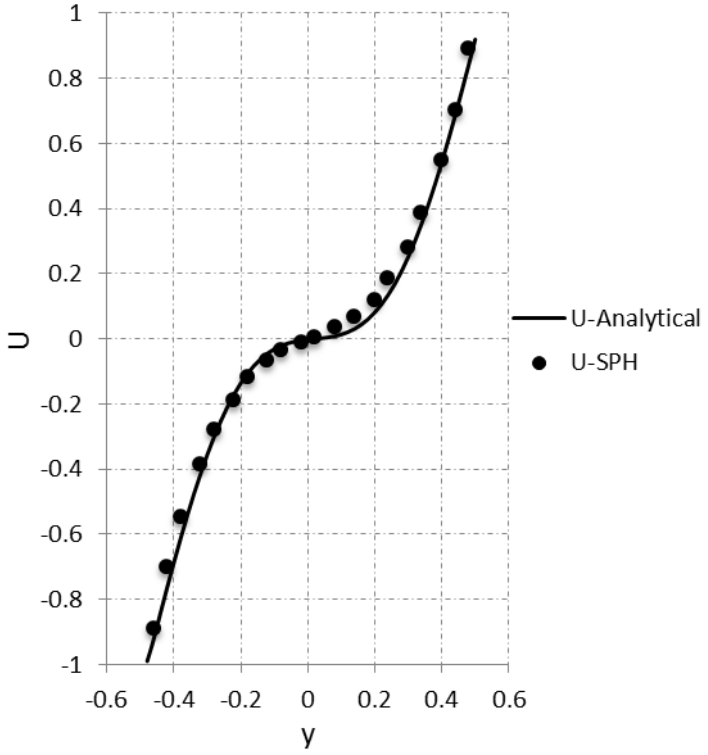


Fig. 3. Comparison of velocity profile for Couette flow at $t = 1.14\ s$ between present SPH results and exact solution [28].

4. Numerical Results

4.1. Rayleigh-Taylor Instability

One of the well-known examples of multiphase interfacial flows with complex interface evolution is Rayleigh-Taylor instability (RTI). This problem has been also investigated in several studies using SPH. For example Grenier et al. [9] and Monaghan and Rafiee [10] studied this problem by WCSPH approach and Cummins and Rudman [18], Hu and Adams [7], and Shadloo et al. [29] studied RTI using ISPH approach. In this problem, computational domain is a rectangular with dimensions of 1×2 (width×height). Initially, particles are set on regular positions and particle spacing is dx=0.005, thus, 200×400 fluid particles are created for simulations. The heavy fluid (ρ_H=1.8) is above the light fluid (ρ_L=1.0) and these two fluids are separated by an interface located at $y = 1 - 0.15\sin(2\pi x)$. Reynolds number is $\mathrm{Re} = \sqrt{H^3 g}/\nu = 420$, where, H is width of domain and, $\boldsymbol{g}$ is gravitational acceleration and ν is kinematic viscosity coefficient which is equal for two fluids. The method proposed by Morris et al. [16] is employed to enforce no-slip boundary condition on solid boundaries using ghost particles. In Fig. 4, position of interface in RTI problem at $t^*=t(g/H)^{1/2}$=5.0 is compared with results of Grenier et al. [9]. It is observed that the complex interface evolution is accurately tracked.

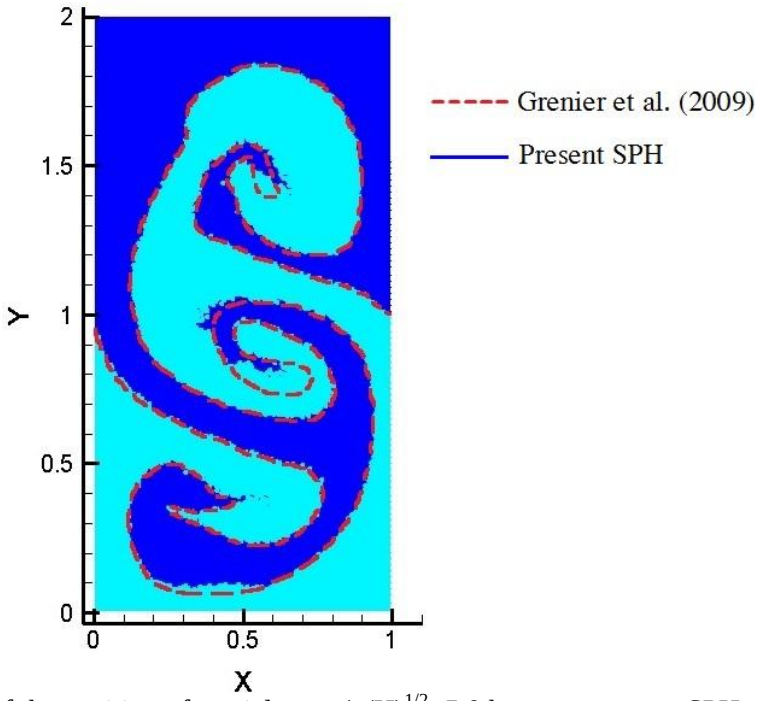


Fig. 4. Comparison of the position of particles at $t(g/H)^{1/2}$=5.0 between present SPH result and simulations of [9].

In order to further analysis of RTI results, similar to [10], the time variation of the highest point of the light fluid is compared with analytical theory of Layzer [30] in Fig. 5. Simulations have been performed with two different spatial resolutions (dx=0.005 and dx=0.01) and in both cases, there is good agreement with theoretical results. It can be observed that with higher resolution, better agreement is achieved.

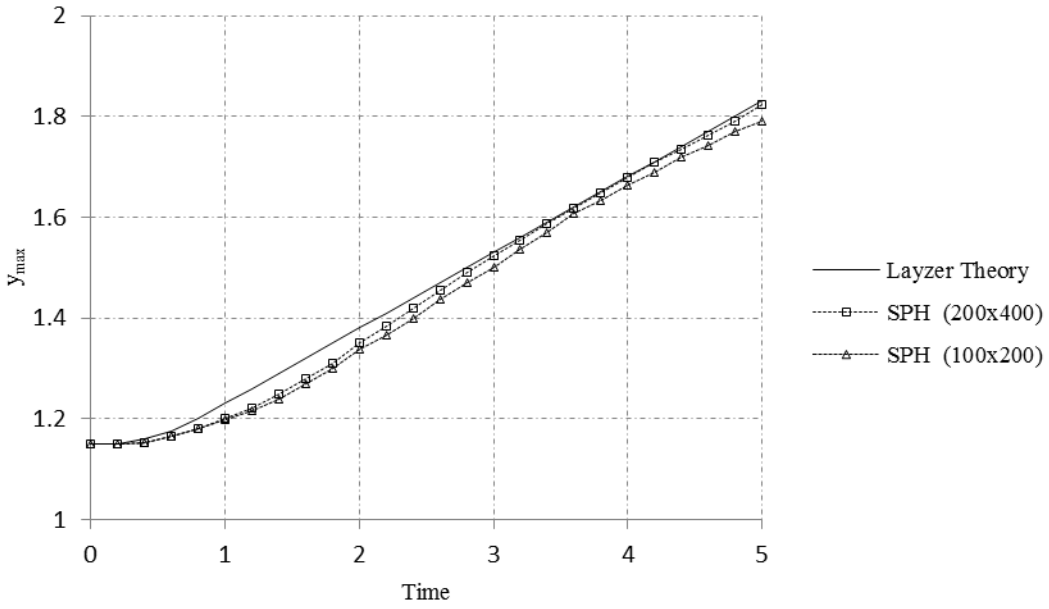


Fig. 5. The time variation of the highest point of the light fluid compared with Layzer theory [30].

4.2. Bubble Rising in a Viscous Fluid

Another example of multiphase flows which is commonly discussed in numerical and experimental studies is dynamics of a bubble rising in a viscous fluid. In addition to many Eulerian mash-based numerical studies, these problems are investigated in several papers using SPH. For example, Colagrossi and Landrini [4], Grenier et al. [9] and Szewc et al. [11] studied this problems by WCSPH approach and Zainali et al [12] studied bubble dynamics in three different cases using ISPH approach. A detailed description of geometry and physical properties of this problem is presented in [31]. In this problem a rectangular domain with dimensions of 1×2 (width×height) is considered. The bubble (ρ_B=0.1) is located initially a distance equal to its diameter above the bottom of the rectangular domain and is surrounded by water (ρ_L=1). Dynamic viscosity coefficient is 0.1 for bubble particles and is 1 for liquid particles. Reynolds number is 35 and Eötvös number is 10 for this problem. Bubble shapes at t=3 are shown in Fig. 6 for two different spatial resolutions (dx=0.00625 and dx=0.0125) compared with FEM results [31]. It can be observed that dynamics of bubble are accurately simulated compared with high resolution FEM results.

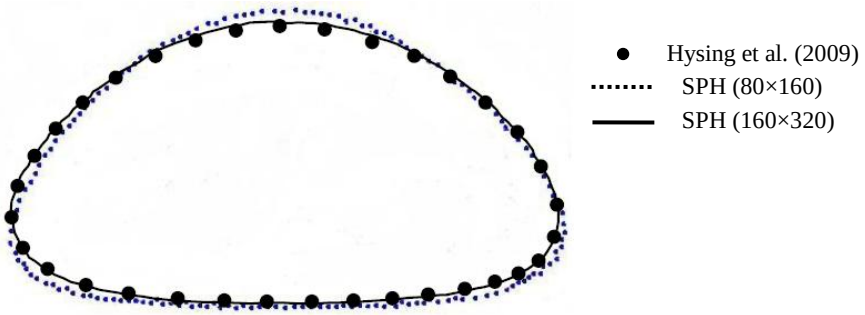


Fig. 6. The shape of the bubble at t=3 compared with FEM results of Hysing et al. [31].

The time variation of center of mass of the bubble is compared with FEM results [31] in Fig. 7. Simulations have been performed with two different spatial resolutions and in both cases good agreement is observed with FEM results. It can be apparently seen that in higher resolution, the agreement is better with high resolution FEM results.

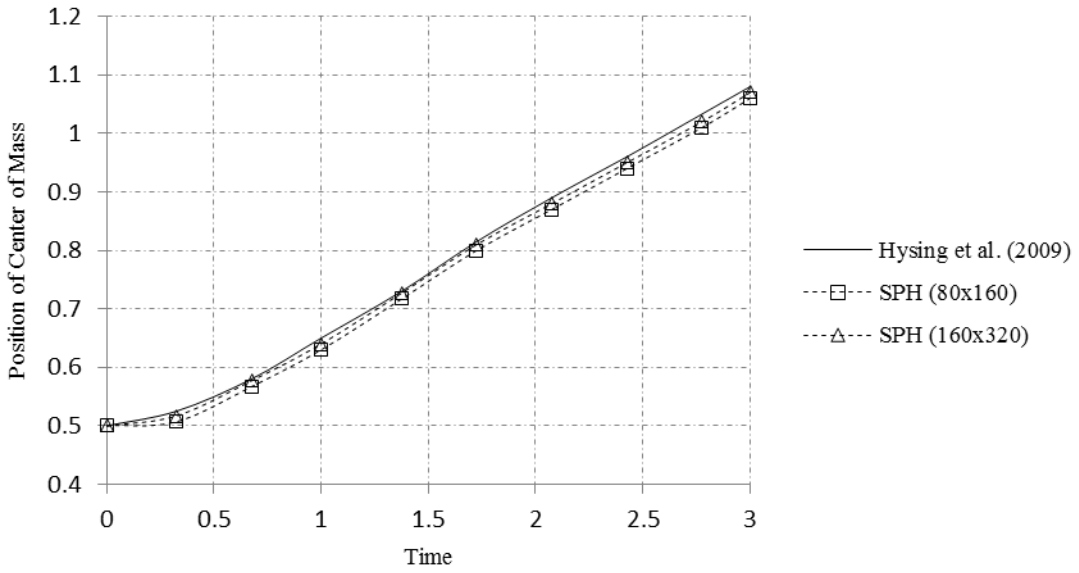


Fig. 7. The time variation of position of center of mass of bubble compared with FEM results [31].

5. Conclusion

Due to the Lagrangian nature of SPH method there is no need for a special algorithm to capture evolving interfaces and thus, in past decades, a great deal of attention has been paid to SPH in the field of numerical simulation of multiphase flows. In present study, a multiphase SPH method for Newtonian fluids has been presented. A special consideration is given to surface tension effects at the interface between two different fluids. Developed algorithm is verified against an analytical solution and good agreement is achieved. Thereafter, two multiphase test cases have been investigated. Performance of developed algorithm was very good in multiphase problem and interfaces were captured simply and accurately. It has been shown that in comparison with Eulerian numerical methods, SPH can handle multiphase problems in a more straightforward manner. Density ratio and viscosity ratio in two phase test cases were moderate. Further investigations are required for multiphase problems with higher density and viscosity ratios. Such studies are postponed to future works.

References:

[1]C. W. Hirt and B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries," Journal of Computational Physics, Vol. 39, pp. 201-225, 1981.
[2]S. Osher and J. A. Sethian, "Fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulations," Journal of Computational Physics, Vol. 79, pp. 12-49, 1988.
[3]J. Monaghan and A. Kocharyan, "SPH simulation of multi-phase flow," Computer Physics Communications, Vol. 87, pp. 225-235, 1995.
[4]A. Colagrossi and M. Landrini, "Numerical simulation of interfacial flows by smoothed particle hydrodynamics," Journal of Computational Physics, Vol. 191, pp. 448-475, 2003.
[5]G. A. Dilts, "Moving-least-squares-particle hydrodynamics—I. Consistency and stability," International Journal for Numerical Methods in Engineering, Vol. 44, pp. 1115-1155, 1999.
[6]J. Monaghan, "On the problem of penetration in particle methods," Journal of Computational physics, Vol. 82, pp. 1-15, 1989.
[7]X. Hu and N. A. Adams, "A multi-phase SPH method for macroscopic and mesoscopic flows," Journal of Computational Physics, Vol. 213, pp. 844-861, 2006.
[8]X. Hu and N. A. Adams, "An incompressible multi-phase SPH method," Journal of Computational Physics, Vol. 227, pp. 264-278, 2007.
[9]N. Grenier, M. Antuono, A. Colagrossi, D. Le Touzé, and B. Alessandrini, "An Hamiltonian interface SPH formulation for multi-fluid and free surface flows," Journal of Computational Physics, Vol. 228, pp. 8380-8393, 2009.
[10]J. Monaghan and A. Rafiee, "A simple SPH algorithm for multi-fluid flow with high density ratios," International Journal for Numerical Methods in Fluids, Vol. 71, pp. 537-561, 2013.

[11]K. Szewc, J. Pozorski, and J.-P. Minier, "Simulations of single bubbles rising through viscous liquids using smoothed particle hydrodynamics," International Journal of Multiphase Flow, Vol. 50, pp. 98-105, 2013.
[12]A. Zainali, N. Tofighi, M. Shadloo, and M. Yildiz, "Numerical investigation of Newtonian and non-Newtonian multiphase flows using ISPH method," Computer Methods in Applied Mechanics and Engineering, Vol. 254, pp. 99-113, 2013.
[13]Z. Chen, Z. Zong, M. B. Liu, L. Zou, H. T. Li, and C. Shu, "An SPH model for multiphase flows with complex interfaces and large density differences," Journal of Computational Physics, Vol. 283, pp. 169-188, 2/15/ 2015.
[14]A. Ghasemi V, B. Firoozabadi, and M. Mahdinia, "2D numerical simulation of density currents using the SPH projection method," European Journal of Mechanics-B/Fluids, Vol. 38, pp. 38-46, 2013.
[15]J. J. Monaghan, "Smoothed particle hydrodynamics," Annual Review of Astronomy and Astrophysics, Vol. 30, pp. 543-574, 1992.
[16]J. P. Morris, P. J. Fox, and Y. Zhu, "Modeling low Reynolds number incompressible flows using SPH," Journal of Computational Physics, Vol. 136, pp. 214-226, 1997.
[17]J. J. Monaghan, "Simulating free surface flows with SPH," Journal of Computational Physics, Vol. 110, pp. 399-406, 1994.
[18]S. J. Cummins and M. Rudman, "An SPH projection method," Journal of Computational Physics, Vol. 152, pp. 584-607, 1999.
[19]A. J. Chorin, "Numerical solution of the Navier-Stokes equations," Mathematics of Computation, Vol. 22, pp. 745-762, 1968.
[20]W. Gao, Y.-l. Duan, and R.-x. Liu, "The finite volume projection method with hybrid unstructured triangular collocated grids for incompressible flows," Journal of Hydrodynamics, Ser. B, Vol. 21, pp. 201-211, 2009.
[21]P. W. Cleary, "Modelling confined multi-material heat and mass flows using SPH," Applied Mathematical Modelling, Vol. 22, pp. 981-993, 1998.
[22]R. Xu, P. Stansby, and D. Laurence, "Accuracy and stability in incompressible SPH (ISPH) based on the projection method and a new approach," Journal of Computational Physics, Vol. 228, pp. 6703-6725, 2009.
[23]S. Nugent and H. Posch, "Liquid drops and surface tension with smoothed particle applied mechanics," Physical Review E, Vol. 62, p. 4968, 2000.
[24]S. Adami, X. Hu, and N. Adams, "A new surface-tension formulation for multi-phase SPH using a reproducing divergence approximation," Journal of Computational Physics, Vol. 229, pp. 5011-5021, 2010.
[25]T. Breinlinger, P. Polfer, A. Hashibon, and T. Kraft, "Surface tension and wetting effects with smoothed particle hydrodynamics," Journal of Computational Physics, Vol. 243, pp. 14-27, 2013.
[26]J. Brackbill, D. B. Kothe, and C. Zemach, "A continuum method for modeling surface tension," Journal of Computational Physics, Vol. 100, pp. 335-354, 1992.
[27]J. P. Morris, "Simulating surface tension with smoothed particle hydrodynamics," International Journal for Numerical Methods in Fluids, Vol. 33, pp. 333-353, 2000.
[28]J. Monaghan, "Smoothed particle hydrodynamic simulations of shear flow," Monthly Notices of the Royal Astronomical Society, Vol. 365, pp. 199-213, 2006.
[29]M. Shadloo, A. Zainali, and M. Yildiz, "Simulation of single mode Rayleigh–Taylor instability by SPH method," Computational Mechanics, Vol. 51, pp. 699-715, 2013.
[30]D. Layzer, "On the Instability of Superposed Fluids in a Gravitational Field," The Astrophysical Journal, Vol. 122, p. 1, 1955.
[31]S. Hysing, S. Turek, D. Kuzmin, N. Parolini, E. Burman, S. Ganesan, et al., "Quantitative benchmark computations of two-dimensional bubble dynamics," International Journal for Numerical Methods in Fluids, Vol. 60, pp. 1259-1288, 2009.