A two-way ISPH-FVM coupling for two-phase flows

Mehdi Helal, Mostafa Safdari Shadloo, Vincent Moureau
CORIA, CNRS, Normandie Univ, UNIROUEN, INSA Rouen Normandie
76000 Rouen, France
Corresponding author: mehdi.helal@coria.fr

Mélody Cailler
Digital Sciences & Technologies Department
SAFRAN TECH
78114 Magny-Les-Hameaux, France

ABSTRACT

A new Smoothed Particle Hydrodynamics (SPH) - Finite Volume (FV) coupling is proposed to describe the global dynamics of multi-scale liquid-gas flows with a controlled level of fidelity and computational cost on unstructured meshes. This approach combines the advantages of a Lagrangian methodology to discretize and follow the strong liquid dynamics and of an Eulerian one to efficiently simulate the dynamics of the gas phase on a coarser grid.

I. INTRODUCTION

Liquid-gas flows with dynamic multi-scale interfaces are frequently encountered in various engineering applications such as gearboxes and fuel injection systems. Their accurate simulations involve taking into account major topological changes of the interface and its interaction with turbulence. Although traditional Eulerian methods are very efficient for simulating large regions with smooth properties, they require complex procedures to deal with moving interfaces. In contrast, meshless particle methods directly track the interface from the particle distribution but involve a higher computational cost.

In recent years, these two approaches have been combined in different ways to get the best out of both. Liu et al. [1] proposed a hybrid method in which one phase is described by the Moving Particle Semi-implicit method (MPS) and the other is defined on a stationary mesh. After solving the flow on the grid, the particles interpolate the velocity and correct their position. Ishii et al. [2] coupled the MPS method with the grid-based Cubic Interpolated Propagation (CIP) approach without employing a tight coupling between the two. Liu et al. [3], [4] proposed to combine particle-based methods with the finite volume method (FVM) by distributing particles only near the interface. In addition, the SPH-FVM coupling was also developed for free surface flows [5]–[7]. The computational domain is then partitioned into FVM and SPH regions, the SPH approximation being advantageous where the free-surface deformations occur.

The present work proposes a two-way coupling of the ISPH approach [8] with the FVM on unstructured meshes [9]. The Lagrangian representation of the liquid phase allows one to solve its dynamics and follow the deformations of the interface accurately with a resolution given by the computational particles while avoiding local refinement of the Eulerian grid. Complementarily, the Eulerian description of the gas phase is used to efficiently simulate the strong dynamics of the large turbulent scales on a coarser grid.

II. MATHEMATICAL MODELS

Each fluid is supposed to be incompressible and all its properties are considered to be constant. No phase change is taken into account.

A. Two-fluid formulation

Following the work of Drew [10] and Hirt and Nichols [11], the two-fluid system of equations can be derived. Assuming a sharp representation of the interface, for each phase k,

$$\begin{aligned} \frac{\partial}{\partial t}\left(\alpha_k\right) + \nabla\cdot\left(\alpha_k\mathbf{u}_k\right) =& 0 \\ \frac{\partial}{\partial t}\left(\alpha_k\rho_k\mathbf{u}_k\right) + \nabla\cdot\left(\alpha_k\rho_k\mathbf{u}_k\otimes\mathbf{u}_k\right) =& -\nabla\left(\alpha_k P_k\right) \\ &+\nabla\cdot\left(\alpha_k\tau_k\right) \\ &+\mathbf{F}^{\text{exchange}} \end{aligned} \tag{1}$$

where α is the volume fraction, ρ the density, P the pressure and τ the viscous stress tensor.

From this formulation, the Eulerian equations and the Lagrangian equations are derived.

B. Eulerian Governing equations

As in [12], a unified mean velocity field is introduced

$$\mathbf{u} = \alpha_l^{\text{SPH}}\mathbf{u}_l^{\text{SPH}} + \alpha_g\mathbf{u}_g \tag{2}$$

with $\cdot_l^{\text{SPH}}$ the projected liquid particle properties. It gives rise to a unique system of equations describing both the gas and liquid phases on the grid:

$$\begin{aligned} \nabla\cdot\left(\mathbf{u}\right) =& \frac{\partial}{\partial t}\left(\alpha_l^{\text{SPH}}\right) + \nabla\cdot\left(\alpha_l^{\text{SPH}}\mathbf{u}_l^{\text{SPH}}\right) \\ \frac{\partial}{\partial t}\left(\mathbf{u}\right) + \nabla\cdot\left(\mathbf{u}\otimes\mathbf{u}\right) =& -\frac{1}{\rho_g}\nabla\left(\alpha_g P_g\right) + \frac{1}{\rho_g}\nabla\cdot\left(\alpha_g\tau_g\right) \\ &+\frac{1}{\rho_g}\mathbf{F}^{\text{exchange}} \\ &+\frac{\partial}{\partial t}\left(\alpha_l^{\text{SPH}}\mathbf{u}_l^{\text{SPH}}\right) \\ &+\nabla\cdot\left(\alpha_l^{\text{SPH}}\mathbf{u}_l^{\text{SPH}}\otimes\mathbf{u}_l^{\text{SPH}}\right). \end{aligned} \tag{3}$$

The resulting set of equations is similar to the pure gas equations except for the additional transport terms.

C. Lagrangian Governing equations

On the other hand, the liquid SPH equations are obtained directly from Eq. 1, considering that the liquid volume fraction at a SPH particle is equal to 1:

$$\begin{aligned} \nabla \cdot (\mathbf{u}_l) =& 0 \\ \frac{\partial}{\partial t}(\rho_l \mathbf{u}_l) + \nabla \cdot (\rho_l \mathbf{u}_l \otimes \mathbf{u}_l) =& -\nabla (P_l) + \nabla \cdot (\tau_l) \\ & - \mathbf{F}^{\text{exchange}}. \end{aligned} \tag{4}$$

The main challenge then lies in modeling the exchange terms that represent the pressure and viscosity contributions at the interface.

D. Coupling methodology

The model is based on two conservative exchange terms between the phases.

It first assumes the velocity equilibrium at the interface and introduces a velocity relaxation process:

$$\mathbf{F}^{\text{relax}}_{l\to g} = -\mathbf{F}^{\text{relax}}_{g\to l} = \rho_g \frac{\chi_l}{\eta}\left(\mathbf{u}_l^{\text{SPH}} - \mathbf{u}\right) \tag{5}$$

where η is a relaxation time parameter and χ a masking term equal to 1 near the interface and 0 elsewhere.

In addition, the immiscibility of the two phases $\alpha_l^{\text{SPH}} + \alpha_g = 1$ is ensured by a conservative non-penetration force:

$$\begin{aligned} \mathbf{F}^{\text{np}}_{l\to g} =& \rho_l \alpha_g \frac{\delta \mathbf{u}}{\Delta t} \\ \mathbf{F}^{\text{np}}_{g\to l} =& \rho_l \alpha_l \frac{\delta \mathbf{u}}{\Delta t} \end{aligned} \tag{6}$$

where $\delta\mathbf{u}$ is the velocity correction solution to the following equation

$$\frac{1 - \left(\alpha_l^{\text{SPH}} + \alpha_g\right)}{\Delta t} + \nabla \cdot 1\delta\mathbf{u} = 0. \tag{7}$$

Assuming $\delta\mathbf{u}$ is derived from a potential, $\delta\mathbf{u} = \nabla\phi$ with ϕ a scalar, it comes to solving the following Poisson equation on the grid:

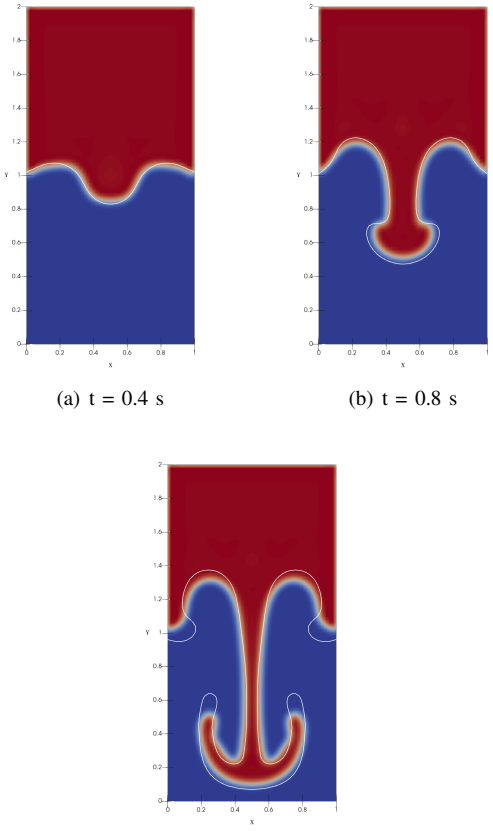
$$\Delta\phi = \frac{\left(\alpha_l^{\text{SPH}} + \alpha_g\right) - 1}{\Delta t}. \tag{8}$$

E. Numerical framework

The particle positions are first updated using a Runge-Kutta scheme, while incompressibility is ensured by solving a Poisson equation through a conjugate gradient method. The liquid SPH velocity field is projected and the liquid volume fraction is reconstructed on the grid using an integral conserving method [14]. The gas flow is then solved on the grid. The relaxation force is implicitly imposed during the prediction and the correction step. A central 4th-order finite volume scheme is used for spatial discretization and a 4th-order TFV4A scheme, a Runge–Kutta like scheme, for the time integration. The Poisson equation is solved with a deflated pre-conditioned conjugate gradient. At the end of the iteration, the relaxation force accumulated as the gas advanced is interpolated by the particles. Finally, the non-penetration force is computed and imposed during the Lagrangian and Eulerian prediction step of the next iteration.

III. Results

The methodology, implemented in the low-Mach number platform YALES2 [9], is applied to a 2D Rayleigh-Taylor instability as in [1]. In the initial state, the heavier fluid $\rho_1 = 3\ \text{kg/m}^3$ is placed upon a lighter one $\rho_2 = 1\ \text{kg/m}^3$ in a channel of size $L_x = 1\ \text{m}$ and $L_y = 2\ \text{m}$. The first one is represented by the SPH particles with the resolution $\Delta_x^{\text{SPH}} = 0.01\ \text{m}$ while the second one is computed on the Cartesian grid with the resolution $\Delta_x^{\text{GRID}} = \Delta_y^{\text{GRID}} = 0.02\ \text{m}$. Both fluids have the same kinematic viscosity $\nu_1 = \nu_2 = 0.01\ \text{m}^2/\text{s}$ and the surface tension is not considered. Gravity is set at $10\ \text{m/s}^2$. A sinusoidal perturbation, $y = 1 + A_0 \cos(2\pi x)$ with the amplitude $A_0 = 0.06\ \text{m}$ is applied at the interface. The boundaries are treated as solid walls. The time step is set to $\Delta t = 5.0 \times 10^{-4}\ \text{s}$. As shown in Fig. 1, under the action of gravity, the disturbance grows when the spike of the heavier fluid is moving downward into the lighter one. To validate the model's ability to represent the interface dynamics, Fig. 2 compares the position of the lowest point of the interface with the results obtained in [1] and an Eulerian approach based on Accurate Conservative Level-set (ACLS) method [13] taking $\Delta_x^{\text{ACLS}} = \Delta_y^{\text{ACLS}} = 0.02\ \text{m}$. The results obtained with the proposed strategy are in good agreement with the two reference solutions considered here.



(a) t = 0.4 s (b) t = 0.8 s

(c) t = 1.2 s

Fig. 1. Evolution of the interface of the Rayleigh-Taylor instability using the SPH-FVM coupling in comparison with the ACLS method results (white lines).

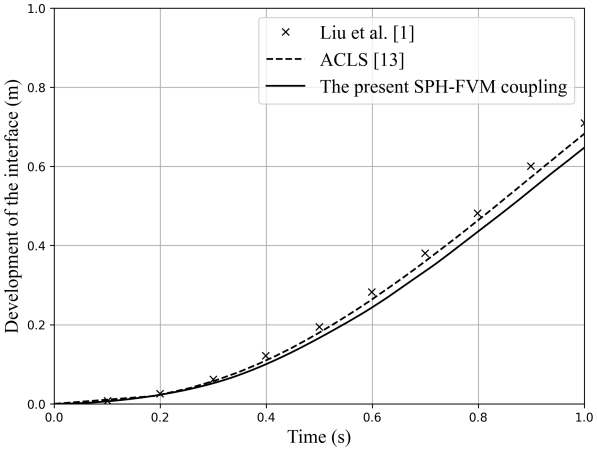


Fig. 2. Comparison of the evolution of the lowest point on the interface

IV. Conclusion

A first step towards a two-way SPH-FVM coupling for liquid/gas flows has been proposed. Although it is difficult to compete in accuracy with the SPH method using two populations of particles or with the Eulerian ones already present in the literature, this work could be of interest for industrial applications such as lubrication or fuel injection. For these applications, on the one hand these classical approaches are too costly and in the other hand one-population SPH would be preferred but currently lacks some gas phase retroaction onto the liquid. In these cases, the proposed method is an interesting alternative. To evaluate the potential of this method, this work aims to consider surface tension and to move towards more complex configurations to validate the methodology.

References

[1] J. Liu, S. Koshizuka and Y. Oka, "A hybrid particle-mesh method for viscous, incompressible, multiphase flows," in *J. Comput. Phys.*, vol. 202, pp. 65–93, 2005.

[2] E. Ishii, T. Ishikawa and Y. Tanabe, "Hybrid particle/grid method for predicting motion of micro- and macrofree surfaces," in *J. Fluids. Eng.*, vol. 128, pp. 921–930, 2006.

[3] X. Liu, L. Guo, K. Morita and S. Zhang, "Development of a hybrid particle-mesh method for two-phase flow simulations," in *Int. J. numer. Methods Fluids*, vol. 82, pp. 334–347, 2016.

[4] X. Liu, K. Morita and S. Zhang, "A conservative finite volume-particle hybrid method for simulation of incompressible interfacial flow," in *Comput. Methods Appl. Mech. Engrg.*, vol. 355, pp. 840–859, 2019.

[5] S. Marrone, A. Di Mascio and D. Le Touzé, "Coupling of Smoothed Particle Hydrodynamics with Finite Volume method for free-surface flows," in *J. Comput. Phys.*, vol. 310, pp. 161–180, 2016.

[6] E. Napoli, M. De Marchis, C. Gianguzzi, B. Milici and A. Monteleone, "A coupled Finite Volume–Smoothed Particle Hydrodynamics method for incompressible flows," in *Comput. Methods Appl. Mech. Engrg.*, vol. 310, pp. 674–693, 2016.

[7] L. Chiron, S. Marrone and A. Di Mascio, "Coupled SPH–FV method with net vorticity and mass transfer," in *J. Comput. Phys.*, vol. 364, pp. 111–136, 2018.

[8] S. J. Cummins and M. Rudman, "An SPH Projection Method," in *J. Comput. Phys.*, vol. 152, pp. 584–607, 1999.

[9] V. Moureau, P. Domingo and L. Vervisch, "Design of a massively parallel CFD code for complex geometries," in *Comptes Rendus Mécanique,*, vol. 339(2), pp. 141-148, 2011.

[10] D. A. Drew, "Mathematical modeling of two-phase flow," in *Annual review of fluid mechanics,* vol. 15(1), pp. 261-291, 1983.

[11] C. W. Hirt and B. D. Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries," in *J. Comput. Phys.,* vol. 39(1), pp. 201-225, 1981.

[12] I. Tsetoglou et al., "A volume-of-solid implicit volume penalty method for moving-body flows," in *Int. J. numer. Methods Fluids,* vol. 97(2), pp. 117-150, 2025.

[13] R. Janodet et al., "A massively parallel accurate conservative level set algorithm for simulating turbulent atomization on adaptive unstructured grids," in *J. Comput. Phys.,* vol. 458, 2022

[14] B. Rottgers and A. Arth, "SPH to Grid: a new integral conserving method," in *arXiv: Instrumentation and Methods for Astrophysics,* 2018.