Incompressible SPH - FVM coupling for two-phase flows in complex geometries

Mehdi Helal[1,2,*], Mélody Cailler[1], Mostafa Safdari Shadloo[2], Vincent Moureau[2]
[1] *CORIA, CNRS, Normandie Univ, UNIROUEN, INSA Rouen, 76000 Rouen, France*
[2] *SAFRAN TECH, Digital Sciences & Technologies Department, 78114 Magny-Les-Hameaux, France*
[*] *Corresponding author: mehdi.helal@coria.fr*

Abstract:
The main objective of this work is to develop an hybrid incompressible SPH - FVM approach to describe the global dynamics of multi-scale liquid-gas flows with a controlled level of fidelity and computational cost. To this aim, we propose to use the advantages of a Lagrangian methodology to follow the strong liquid dynamics and of an Eulerian one to describe efficiently the gas dynamics on a coarser grid.

Keywords: Multi-scale, Two-phase flow, Euler, Lagrange, Two-way coupling, Unstructured grids.

Introduction:
Liquid-gas flows with dynamic multi-scale interfaces are encountered in various industrial applications such as gearboxes and fuel injection systems. Mastering turbulent two-phase flows in such components is crucial to ensure their performance. To this end, predictive and low CPU-demanding numerical simulation is of high interest. The present work presents a two-way coupling of the incompressible Smoothed Particle Hydrodynamics method (SPH) [1] with the Finite-Volume Method (FVM) on unstructured meshes [2]. The Lagrangian representation of the liquid phase allows to solve its dynamics and follow the deformations of the interface accurately with a resolution given by the computational particles. On the other hand, the Eulerian description of the gas phase is used to efficiently simulate the strong dynamics of the large turbulent scales on a coarser grid.

Numerical Method:
While the numerical approach for the incompressible SPH describing the liquid phase is standard, the FVM advancement of the gas phase taking into account the presence of the liquid is more complex. The combined advancement of SPH and FVM requires two coupling ingredients: i) penalization of the Eulerian velocity in the liquid region with the SPH projected velocity, ii) computation of the additional source term in the liquid SPH momentum equation to conserve momentum. This strategy is inherited from the Volume of Solid approach [3, 4], in which an Eulerian composite velocity is introduced: $\mathbf{u} = \alpha_l^{SPH}\mathbf{u}_l^{SPH} + \alpha_g \mathbf{u}_g$ with $\cdot_l^{SPH}$ the projected liquid particle properties and α the volume fraction. The Eulerian composite velocity advancement reads:

$$\begin{aligned}
\nabla\cdot(\mathbf{u}) =& \frac{\partial}{\partial t}\left(\alpha_l^{SPH}\right) + \nabla\cdot\left(\alpha_l^{SPH}\mathbf{u}_l^{SPH}\right)\\
\frac{\partial}{\partial t}(\mathbf{u}) + \nabla\cdot(\mathbf{u}\otimes\mathbf{u}) =& \frac{\partial \alpha_l^{SPH}\mathbf{u}_l^{SPH}}{\partial t} + \nabla\cdot\left(\alpha_l^{SPH}\mathbf{u}_l^{SPH}\otimes\mathbf{u}_l^{SPH}\right) - \frac{1}{\rho_g}\nabla\left(\alpha_g P_g\right) + \frac{1}{\rho_g}\nabla\cdot\left(\alpha_g \tau_g\right)\\
&+ \frac{1}{\rho_g}\underbrace{\int_{\Gamma}\left(-P_g I + \tau_g\right)\cdot dA}_{\text{Exchange term}}
\end{aligned} \tag{1}$$

with Γ the liquid-gas interface. The main challenge then lies in properly modelling the exchange term in (1). A first attempt would be to use an IBM-like penalization [4] of the Eulerian velocity $\mathbf{u}$ in the liquid region with the SPH projected velocity: $\boldsymbol{F}^{\text{pen}} = \frac{\chi_l}{\eta}\left(\mathbf{u}_l^{SPH} - \mathbf{u}\right)$ with χ_l the mask equal to 1 in the liquid region and zero elsewhere, and $\eta = \beta\Delta t$ the penalization time. On the other hand, a source term appears in the incompressible SPH momentum equation by interpolating the penalization force on the particles:

$$\begin{aligned}
\frac{\mathrm{D}\rho_l}{\mathrm{D}t} =& -\rho_l\nabla\cdot\mathbf{u}_l\\
\frac{\mathrm{D}\mathbf{u}_l}{\mathrm{D}t} =& -\frac{1}{\rho_l}\nabla p_l + g + \nu_l\Delta\mathbf{u}_l + \frac{1}{\rho_l}F^{\sigma} - \frac{1}{\rho_l}\underbrace{F^{\text{pen}}}_{interpolated}
\end{aligned} \tag{2}$$

Preliminary results:

The methodology, implemented in the low-Mach number platform YALES2 [2], is applied to a 2D test case where a droplet of diameter D_d undergoes a constant volume force F. The configuration consists of a periodic domain of size $L_x = 16D_d$ and $L_y = 8D_d$ occupied by air and a water droplet initially located at $(2D_d\ 4D_d)$. The grid resolution is given by $\Delta_x/D_d = 8$ while the SPH resolution is given by $\Delta_x^{SPH}/D_d = 32$. The calculation is compared with a level-set based simulation [5] with Adaptive Mesh Refinement (AMR) considering a one-fluid formalism for the velocity. In this full Euler simulation, the grid resolution at the interface is $\Delta_{x,\Gamma}/D_d = 32$ *i.e.* similar to the SPH resolution. The observations from these initial results indicate that, although the wake is similar, the drag force exerted on the droplet is underestimated.

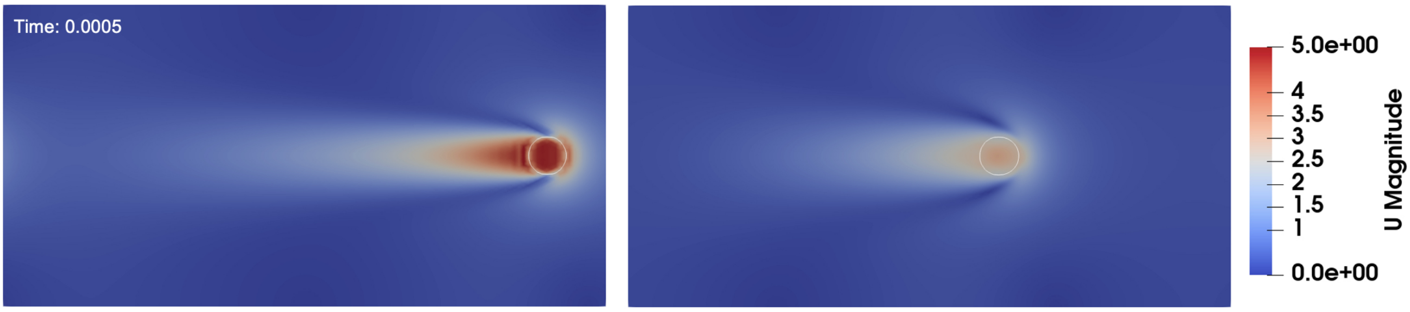

Figure 1: Velocity field at time $T = 0.0005\,\mathrm{s}$ obtained with SPH - FVM coupling [left] and full Euler [right]

Conclusion:

This work represents a first step of a two-way SPH - FVM coupling for two-phase flows. Further work on the penalization formulation has to be achieved but the preliminary results are promising. In addition, this work intends to go towards more complex configurations to validate the methodology.

References

[1] S.J. Cummins, M. Rudman, *An SPH Projection Method*, J. Comp. Phys., **152**(2), 584–607, 1999.

[2] V. Moureau, P. Domingo, L. Vervisch, *Design of a massively parallel CFD code for complex geometries*, Comptes Rendus Mécanique, **339**(2), 141–148, 2011.

[3] C. W. Hirt, B. D. Nichols, *Volume of fluid (VOF) method for the dynamics of free boundaries*, J. Comp. Phys., **39**, 201–225, 1981.

[4] I. Tsetoglou, M. Cailler, P. Bénard, G. Lartigue, V. Moureau, J. Réveillon, *A volume-of-solid implicit volume penalty method for moving-body flows*, Int. J. Numer. Methods Fluids, **in press**, 2024.

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