A VoF-LPT Solver for 3D Numerical Simulation of Aerated Slug Flow and Closure Law Development

Arianna Bonzanini,[1] Stefan Radl,[2] Pietro Poesio[1]

[1]Università degli Studi di Brescia, Italy. [2]Institute of Process and Particle Engineering, Graz University of Technology, Austria.

Contact information

pietro.poesio@unibs.it, radl@tugraz.at.

Introduction

Multiphase flow in long pipes is of high relevance to the chemical and petroleum industry. Often, gas-liquid flows are of central interest, in which different flow regimes manifest. Predicting flow in different regimes is connected with numerous challenges. Especially the so-called "aerated slug flow" regime (see Figure 1) in (almost) horizontal pipes requires specific attention to both (i) the physical models, as well as (ii) numerical schemata used [1].

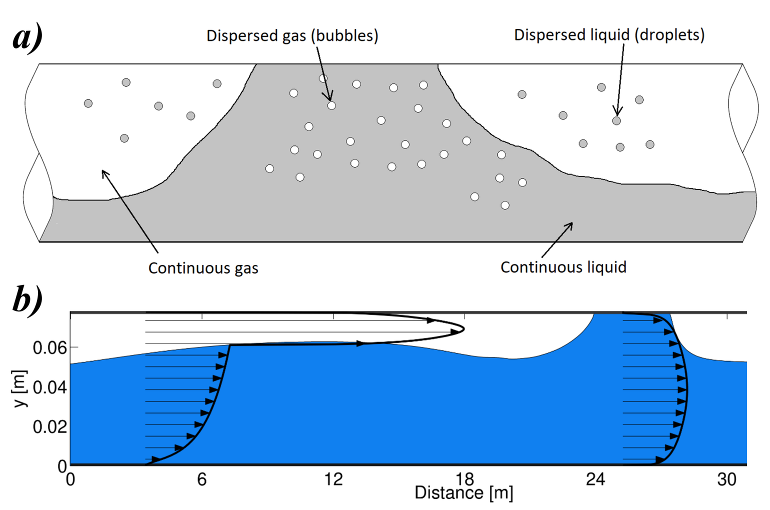

Fig. 1: Illustration of a four-fields two-phase flow in slug regime (panel a), as well as qualitative behaviour of velocity profiles along the pipe (panel b; adopted from [1]).

The present contribution is motivated by the lack of generally-valid closure laws to be used in one-dimensional models of gas-liquid flows in pipes. While significant improvements have been made in the recent past [2,3], the applicability of certain closures (e.g., for the bubble deposition rate on the free surface) is still limited. Especially, there is a lack of closures for one-dimensional slug flow models. Our overall aim is to systematically build such closure relations from detailed three-dimensional multiphase flow simulations of aerated slug flow. Here we present a new solver strategy, and first results.

Volume of Fluid & Lagrangian Bubble Tracking

Our multiphase flow solver is based on a coupled Volume of Fluid-Lagrangian Particle Tracking (VoF-LPT) strategy, and was developed in OpenFOAM®. Therefore, a new Lagrangian library was integrated in the available solver '*interFoam*'. The latter is the main flow solver, and describes the interface between the continuous gas and liquid phases using the VoF method. The motion of dispersed gas (i.e., bubbles in the continuous liquid phase) is handled by a Lagrangian particle tracking approach (see Figure 2). This separate handling of bubbles is necessary since the VoF method alone does not allow the description of many (resolved) gas bubbles at a reasonable computational cost. The approach used in our present study is – in spirit – similar to what has been proposed by Tomar et al. [4] in the field of primary atomization of liquid jets.

Currently, our new solver accounts also for the possibility of bubble deposition into the continuous gas phase. However, it cannot predict bubble generation at present due to the lack of a missing closure for the local bubble generation rate.

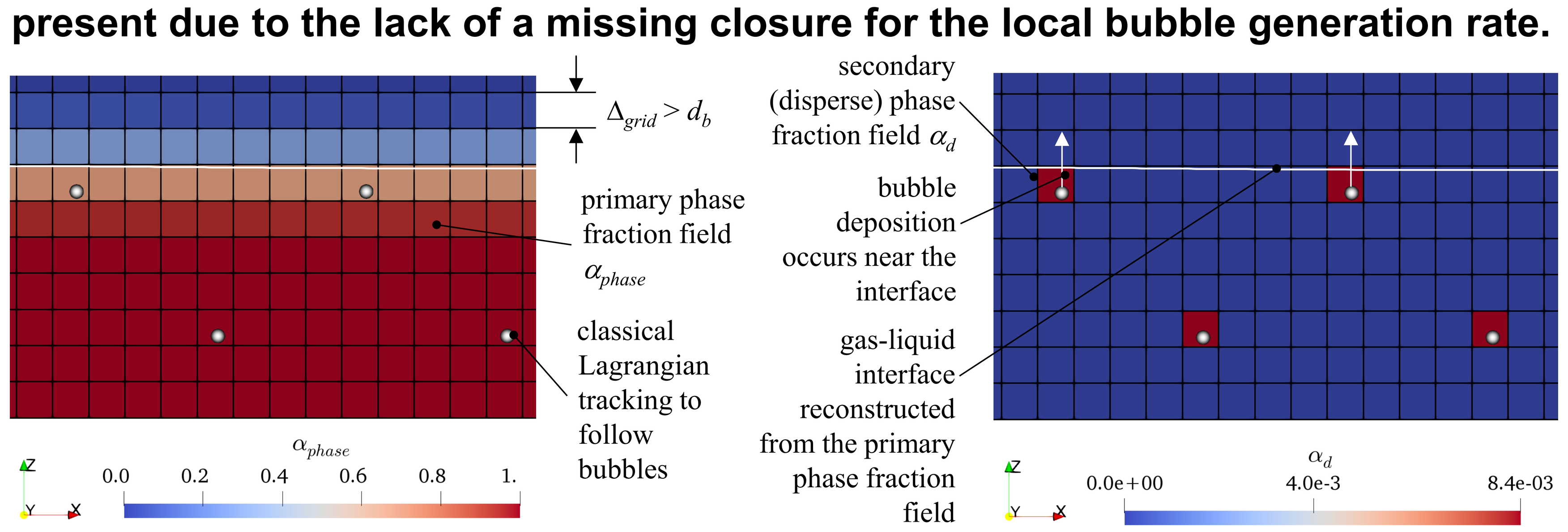

Fig. 2: Idea of the combined Volume of Fluid (VoF) and Lagrangian bubble tracking approach to predict deposition of bubbles in the continuous gas phase.

Implementation and Benchmarking

A new cloud class has been implemented in OpenFOAM® that allows for particle removal near the gas-liquid interface. Specifically, we define a threshold of the liquid volume fraction upon which the bubble is deposited: the bubble is first marked as deactivated, and subsequently removed. At the removal time step, which happens before performing the cloud motion, the deactivated particle is deleted from the domain, and the correct amount of gas is denoted to the continuous gas phase. Also, the removed bubble's gas volume is no longer accounted for when constructing the secondary (disperse) phase fraction field α_d (see Figure 2). This ensures overall mass conservation. The separation of the deactivation and the effective deletion event in two time steps is required by the need of counting the deleted particle volume fraction. Figure 3 illustrates the procedure that is used to delete bubbles, and correctly shift the bubble volume from the secondary phase field α_d to the primary phase field α_{phase}.

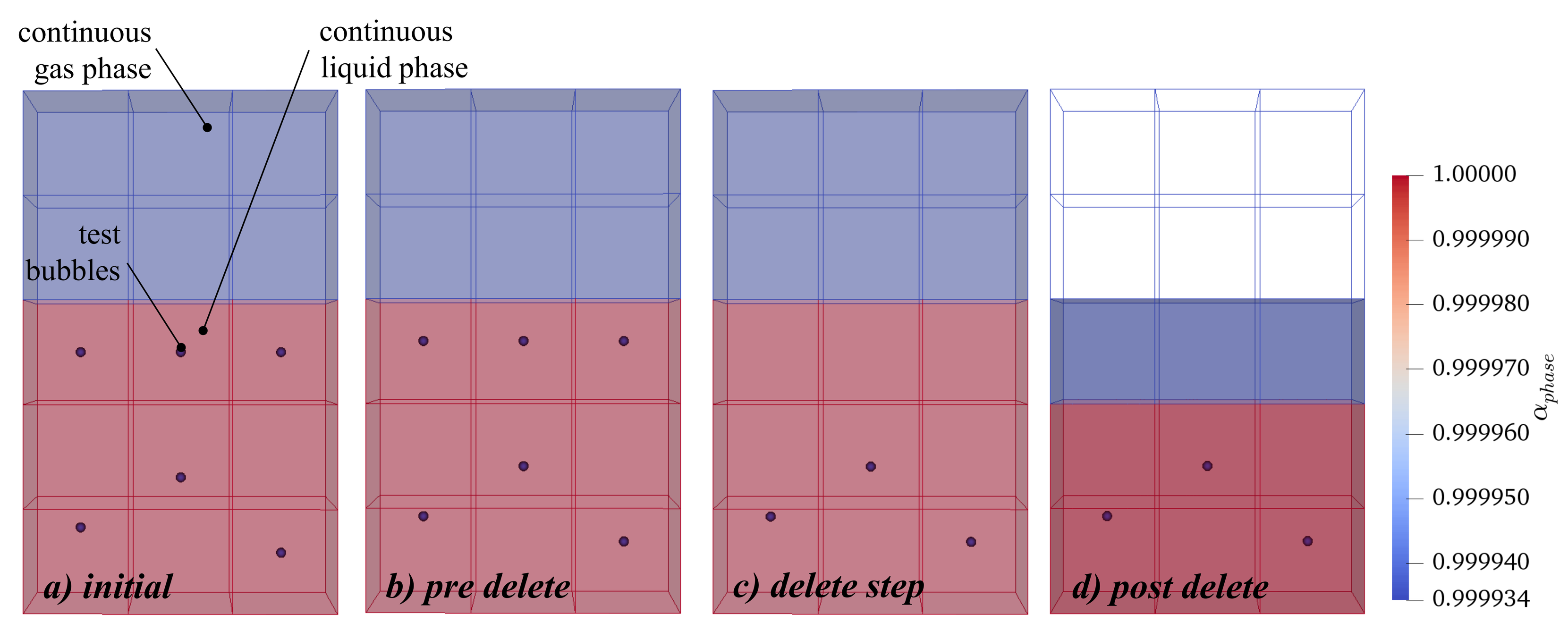

Fig. 3: Computational domain with boundary conditions for the temperature field. A flow field is driven in the *x* direction by means of an imposed pressure gradient. An artificial heat sink prevents the fluid from saturating.

Results for Horizontal Pipe Flow

The simulation setup illustrated in Figure 4 aims at representing a realistic flow situation, and is meant to demonstrate the feasibility of the developed VoF-LPT solver. Specifically, we consider a pseudo two-dimensional channel with the dimensions L_x / L_y / L_z = 2 m / 0.01 m / 0.08 m, discretized with an appropriately fine grid (N_x / N_y / N_z = 1000 / 1 / 40). In this channel we consider a slug body in the middle of the channel, which is set as the initial condition. For the initial and inlet velocity of the gas and the liquid we assume a speed of 2.0 m/s. Bubbles are injected at the beginning of the simulation at random positions with a diameter of 0.001 m, and an initial velocity of zero. Bubbles are transported by the continuous liquid, float up, and ultimately merge with the continuous gas phase, i.e., they deposit in the gas slug. At the moment of the deposition event, the bubbles' volume is transferred to the computational cell in which they are.

Figure 4 shows snapshots at various times as the slug moves and the bubbles deposit. Our data indicates that deposition is fast, and occurs extremely rapid in regions with a gas slug, i.e., in which α_{phase} < 0.5. Bubbles dispersed in regions that are fully occupied with liquid, first move against gravity, and subsequently deposit in the propagating gas slug.

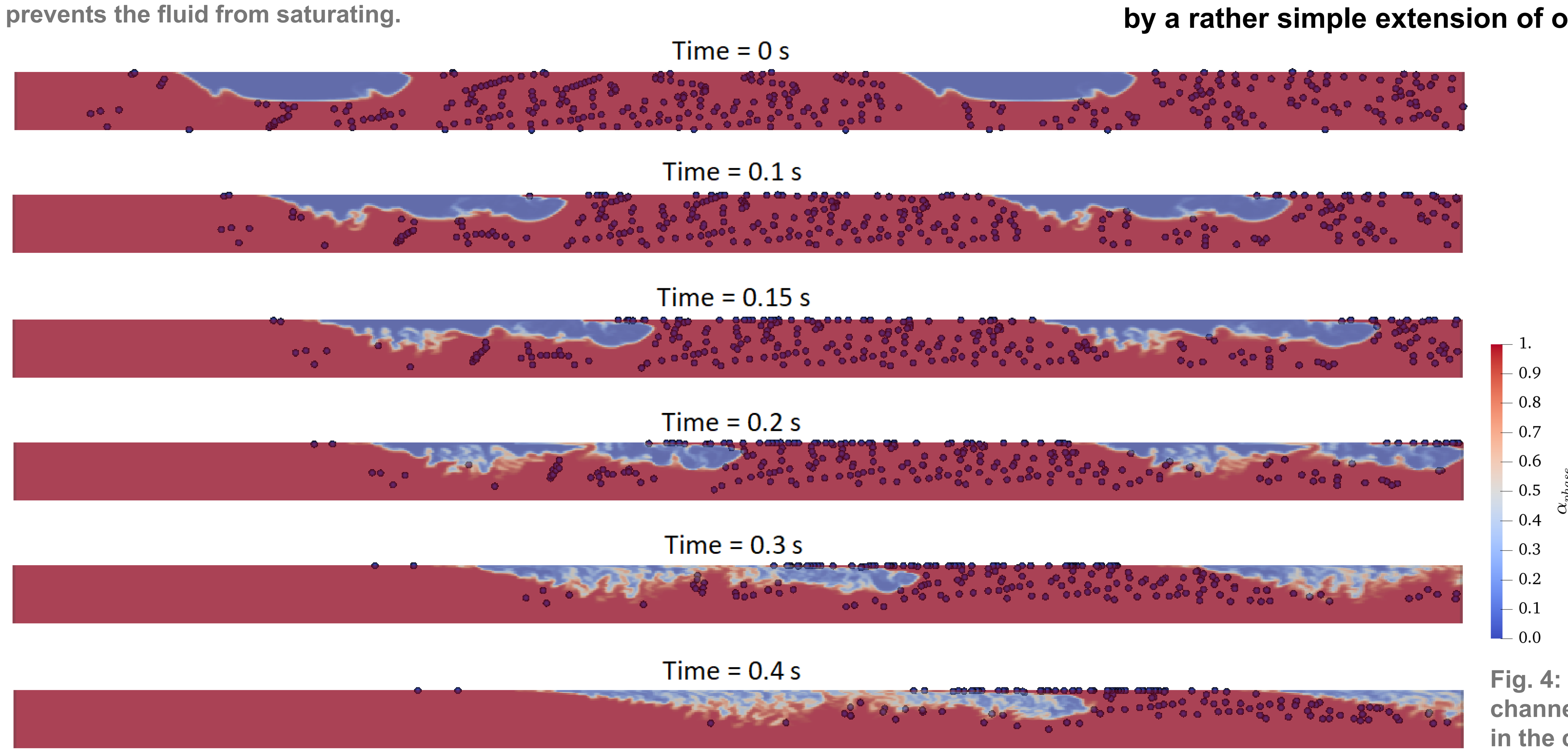

Fig. 4: Slug evolution in a 2D channel, with bubble deposition in the continuous gas phase.

Conclusion & Outlook

We present the a numerical strategy, namely a VoF-LPT solver, that aims on simulating details of multiphase flow in the so-called aerated slug flow regime. Also, first results relevant for this flow regime have been generated, illustrating the feasibility of the approach. Most important, due to the adoption of a splitting approach (i.e., the dispersed gas phase is handled differently as the free gas-liquid interface), the resolution requirements are much lower compared to a situation in which one aims at fully resolving both bubbles and the free interface. Also, modeling bubble-bubble coalescence (or bouncing) is possible by a rather simple extension of our VoF-LPT framework.

Future work will focus on the development of a rigorous closure to predict bubble deposition rates in aerated slug flow. Specifically, we plan to couple our solver with the toolbox CPPPO [5] in order to compute filtered values of all relevant fields (velocities, volume fractions, etc.). These filtered quantities will then be correlated with the deposition rate, and will be ultimately used to calibrate parameters of a suitable functional form. Similar strategies were already successfully applied in the past to gas-particle flows for the development of, e.g., force closures [6].

References

[1] A. Bonzanini, Development and improvement of numerical methods for one-dimensional multiphase flow in pipes (2018), PhD thesis, Università degli Studi di Brescia, Italy.

[2] A. Bonzanini et al., Velocity profiles description and shape factors inclusion in a hyperbolic, one-dimensional, transient two-fluid model for stratified and slug flow simulations in pipes (2018). *Petroleum,* in press.

[3] A. Bonzanini et al., Simplified 1D incompressible two-fluid model with artificial diffusion for slug flow capturing in horizontal and nearly horizontal pipes (2017), *Energies* 10(9),1372.

[4] G. Tomar et al., Multiscale simulations of primary atomization (2010). *Computers & Fluids* 39 1864–1874.

[5] F. Municchi et al., Highly efficient spatial data filtering in parallel using the opensource library CPPPO (2016), *Computer Physics Communications* 207, 400-414.

[6] S. Radl and F. Municchi, Spatial Filtering for Scale Bridging and Its Application to Transport in Dense Particle Beds (2018), *Advances in Chemical Engineering* 53, 153-237.