Simulating an Arbitrary Number of Immiscible Fluids Using a Phase-Field Method: Development and Validation

M. Bagheri[1], M. Wörner[2], H. Marschall[1]
[1]Technische Universität Darmstadt, Darmstadt
[2]Karlsruhe Institute of Technology (KIT), Karlsruhe

In this study we present the most recent work in the context of method development, implementation, and validation of a phase-field interface capturing method supporting solving for the evolution of an arbitrary number of immiscible, incompressible fluids. Herewith, we further enhanced our two-phase (N=2) diffuse interface library implemented in FOAM-extend 4.1 in a consistent and conservative manner to incorporate multiple (N>2) phases.

When underlying a diffuse interface phase-field methodology [1], to devise a consistent and robust generic multiphase (N>2) flow framework is non-trivial: it is vital that the model implementation is fully reduction consistent [2, 3], i.e. the N-phase model reduces to the traditional two-phase model upon existence of only two phases in the system, and to obey thermodynamic constraints, i.e. the N-phase model is mass and momentum conservative, as well as follows the second law of thermodynamics and Galilean invariance [2, 3].

For validation we consider a comprehensive set of validation cases. For instance, in a 3-phase scenario, we considered the floating liquid lens problem where an oil drop that is initially floating on the air-water interface is subjected to different magnitudes of gravitational acceleration [see Fig. 1]. The final thickness of the drop will then be compared to an equilibrium and an asymptotic analysis available in literature.

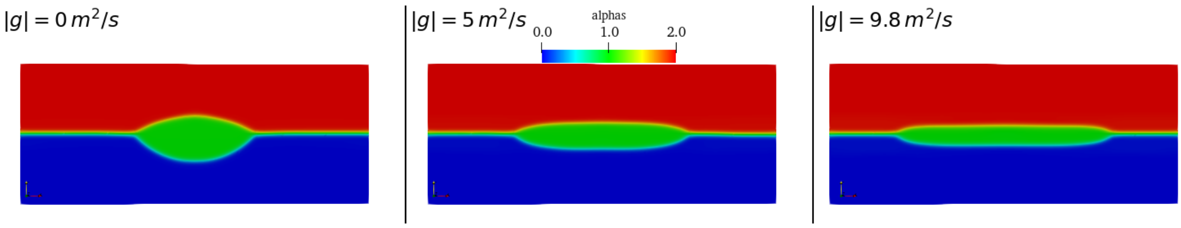

Fig. 1: Final configuration of the floating liquid lens problem for different values of gravitational acceleration. Blue: Water, Red: Air, Green: Oil

In a 4-phase scenario, we considered the four-phase fluid mixture problem. In this problem, dynamics of four immiscible, incompressible, and isothermal fluids with different transport properties is assessed [see Fig. 2].

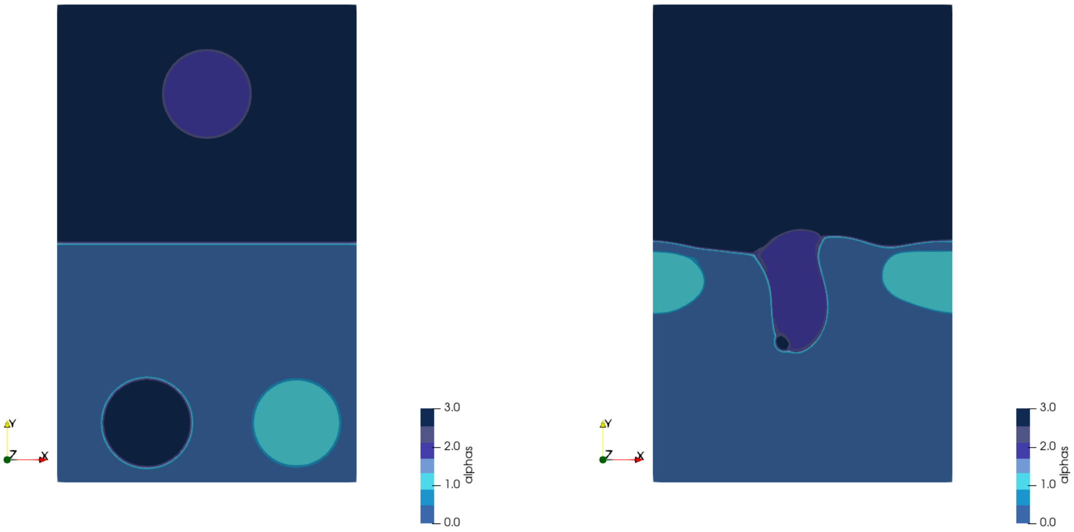

Fig. 2: Four-phase fluid mixture problem, initial state (left), the state after 0.25s (right)

The multiphase diffuse-interface model is implemented in OpenFOAM (FOAM-extend 4.1) as model library and top-level solver referred to as phaseFieldFoam. Following a rigorous object-oriented and physics-guided development paradigm has resulted into a versatile code design which is extendible towards more (multi-)physics, which we will detail on in the talk. Moreover, the framework fully supports the deployment of modern High Performance Computing techniques, such as dynamic load balancing (DLB) along with Adaptive Mesh Refinement (AMR).

Acknowledgments

The authors acknowledge the financial support by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) project number 237267381 SFB/Transregio 150 sub-project B08. Part of the calculations have been conducted on the Lichtenberg High Performance Computer of the TU Darmstadt.

References

[1] Cahn, J. W., & Hilliard, J. E. (1958). Free energy of a nonuniform system. I. Interfacial free energy. The Journal of chemical physics, 28(2), 258-267.

[2] Dong, S. (2018). Multiphase flows of N immiscible incompressible fluids: a reduction-consistent and thermodynamically-consistent formulation and associated algorithm. Journal of Computational Physics, 361, 1-49.

[3] Huang, Z., Lin, G., & Ardekani, A. M. (2021). A consistent and conservative volume distribution algorithm and its applications to multiphase flows using Phase-Field models. International Journal of Multiphase Flow, 103727.