DEVELOPMENT AND DEPLOYMENT OF DIFFUSE INTERFACE PHASE-FIELD METHODS IN OPENFOAM

H. MARSCHALL[1], M. BAGHERI[1], N. SAMKHANIANI[2],
A. STROH[2], B. FROHNAPFEL[2], M. WÖRNER[3]
[1] *Computational Multiphase Flow, Technical University Darmstadt,*
Alarich-Weiss Str. 10, 64287 Darmstadt, Germany
[2] *Karlsruhe Institute of Technology (KIT), Institute of Fluid Mechanics,*
Kaiserstr. 10, 76131 Karlsruhe, Germany
[3] *Karlsruhe Institute of Technology (KIT), Institute of Catalysis Research and Technology,*
Engesserstr. 20, 76131 Karlsruhe, Germany

Keywords: *Multiphase CFD, phase-field method, diffuse interface model, drop impact*

We have developed a unified solver framework for two-phase flow based on diffuse-interface phase-field methods [1], which is to be released within the FOAM-extend project. In contrast to standard sharp interface model approaches, phase-field methods rely on diffuse interface models. As their name suggests, these methods allow for diffusion of the phase constituents in a thin interfacial region of well-defined thickness, thus, promoting a smooth but rapid transition of phase properties such as density and viscosity. Particularly, capillary-dominated two-phase flow can be dealt with at high accuracy, i.e. parasitic currents are found to be low and consistently converging under mesh refinement [2].
The present work focus on droplet impact and impingement scenarios at high dynamics. Recent simulations for both droplet impact on thin liquid films of the same fluid, and droplet impingement and bouncing on a heated hydrophobic surface show very good agreement with experiments (see Fig. 1). The talk will detail on necessary method enhancements to achieve this.

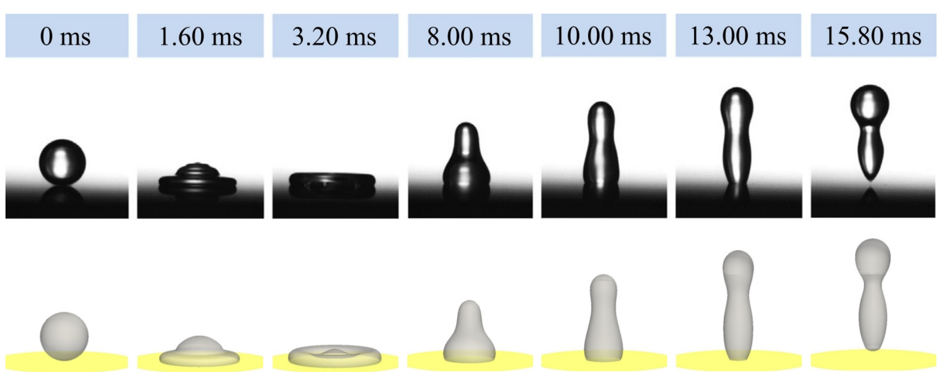

Figure 1: Image sequence of bouncing droplet ($d_0 = 2.3$ mm, We $= 20$, $T_{d,0} = 20$°C) on the smooth hydrophobic surface ($\theta_e = 120$, $T_s = 60$°C). Top: experiment [**?**], bottom: simulation.

Acknowledgments

Funded by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) – Project-ID 237267381 - SFB TRR 150.

References

[1] X. Cai, H. Marschall, M. Wörner, and O. Deutschmann, “Numerical Simulation of Wetting Phenomena with a Phase field Method using OpenFOAM,” Submitted, 2015.

[2] F. Jamshidi, H. Heimel, M. Hasert, X. Cai, H. Marschall, and M. Wörner, “On suitability of phase-field and algebraic Volume-Of-Fluid OpenFOAM solvers for gasliquid microfluidic applications,” *Comput. Phys. Commun.*, vol. 236, pp. 72–85, 2019.