Hessisches Kompetenzzentrum
für Hochleistungsrechnen

Multiphysics Simulation of Reactive Bubbly Flows

Project Manager
Constantin Habes

Principal Investigator
Dr.-Ing. Holger Marschall

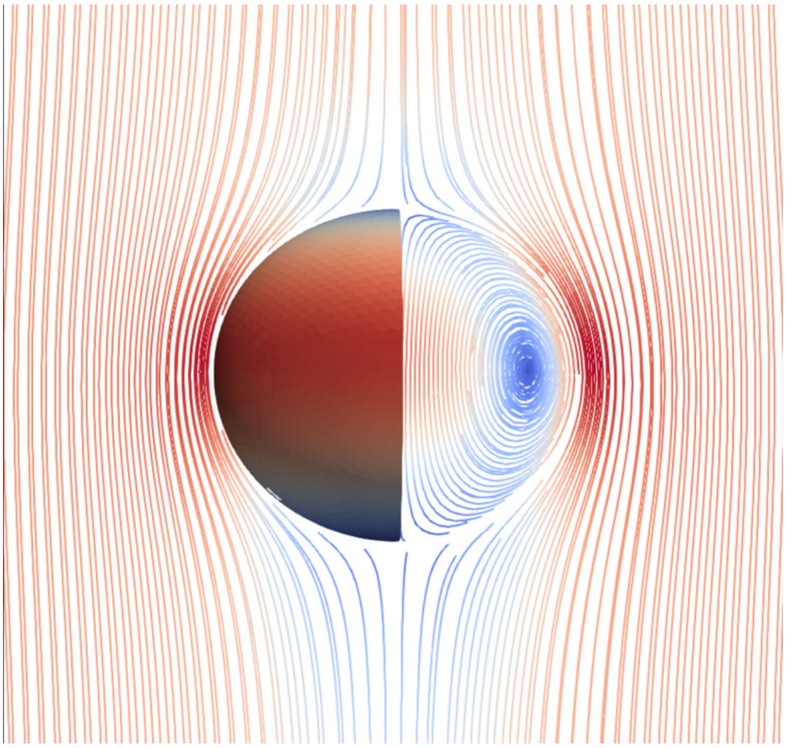
Project Term
2022 - 2022

Clusters
Lichtenberg Cluster Darmstadt

Software
OpenFOAM

Institute
Mathematical Modeling and Analysis (MMA)

University
Technische Universität Darmstadt



Introduction

Historically, CFD codes implementing arbitrary Lagrangian Eulerian interface tracking (ALEIT) for the simulation of bubbly flows were highly customised for this specific application. However, taking a broader look at multiphase flows in general, it is noticeable that they resemble other multiphysics systems in their basic structure. Analogous to Fluid structure interaction (FSI) or conjugate heat transfer problems (CHT), multiphase systems also involve several regions with special physical properties that interact with each other across interfaces. Exploiting these structural similarities, a novel unified multiphysics framework for OpenFOAM named multiRegionFoam is introduced, which incorporates the ALE-IT and is tested for its application towards reactive bubbly flows. In this context, the parallelisability of the new framework is addressed in particular, since interface coupled multiphysics simulations in general and simulations of reactive bubbly flows in particular are very computationally intensive.

Methods

Development of a a unified, highly parallelisable multiphysics framework that implements a Dirichlet-Neumann algorithm convergence control based on interface residuals and is tested for its application towards reactive bubbly flows using the ALE-IT. For the validation of the bubbly flow test simulations, the results were compared to experimental data and simulation data of previous studies.

Results

A unified, highly parallelisable multiphysics framework called multiRegionFoam was successfully developed/ further developed, which proved to outperform previously used ALE-IT codes regarding parallel speedup. The test cases of inert single rising bubbles showed that multiRegionFoam is very well able simulate their behaviour during the rise. However, for larger bubbles, that undergo stronger deformations during the rise, a small underestimation of the terminal rise velocities could be observed. Due to time limitations of the project, robust simulations of reactive rising bubbles could not be performed within the scope of the work.

Discussion

With respect to the new unified multiphysics framework, it can be concluded that all foundations have been laid for it to be used for the simulation of reactive bubbly flows. However, further research must be carried out to reassess the interface curvature calculation and the implementation of the finite area method operators in foam-extend 4.1, as these could be the cause for the underestimated rise velocities. Also, the coupling of reactive species transport across interfaces needs to be further investigated in order to make multiRegionFoam applicable to reactive bubbly flows in the future. Moreover, further work should focus on integrating surfactant transport and the use of subgrid-scale models in multiRegionFoam. In addition, the effect of fully-implicit pressure-velocity coupling, as well as the use of monolithic interface coupling on the overall convergence of reactive bubbly flows should be investigated.

Figures

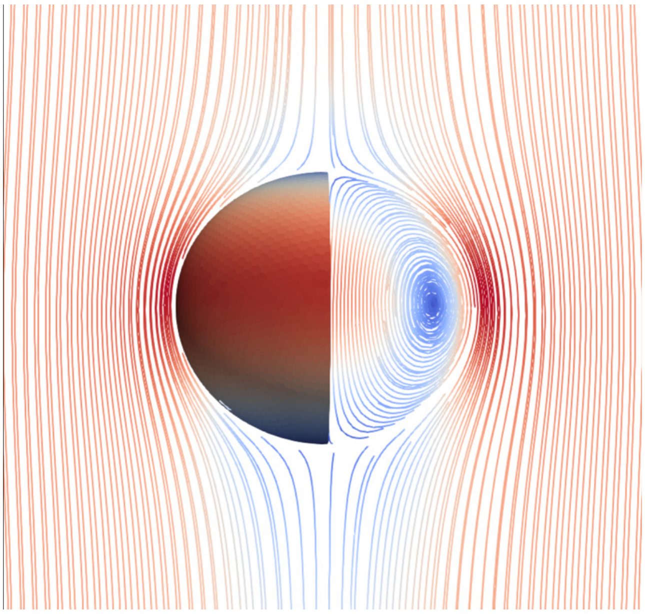


Figure 1: Streamlines coloured according to the velocity magnitude (blue is slow)

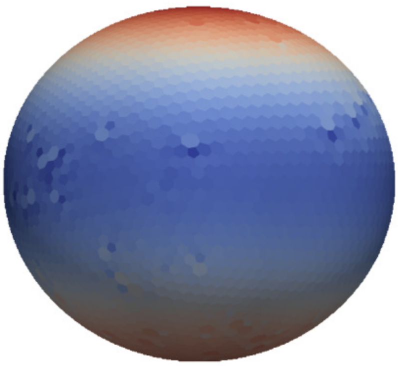


Figure 2: Interface pressure field on the liquid side (blue is low)

Last Update: 2022-12-22 15:58