Abstract Submitted
for the DFD20 Meeting of
The American Physical Society

Benchmarking of CFD solvers for the simulation of two-phase jets[1] YILIN ZHUANG, GABRIEL F. N GONALVES, CRISTIAN RICARDO CONSTANTE AMORES, THOMAS ABADIE, OMAR K. MATAR, Imperial College London — Multiphase flows have received significant interest due to their occurrence in a multitude of natural and industrial applications. In this study, we perform the benchmarking of solvers for the simulation of two-phase jets. The Navier-Stokes equation are solved under the assumption of the single-fluid formulation and therefore, the interfacial location must be also determined as part of the solution. We evaluate solvers based on both interface-capturing methods (such as Basilisk and OpenFOAM) and interface-tracking methods. We consider the scenario of a water drop formation from a cylindrical nozzle and the accuracy of the different solvers is compared with experimental results, in terms of jet-shape and drop-size. A thorough study in terms of the computational requirements (e.g., grid resolution) and the computational cost is also performed.

[1]This work is supported by the EPSRC MEMPHIS (EP/K003976/1) and PREMIERE (EP/T000414/1) Programme Grants, the Royal Academy of Engineering, and BP-ICAM.

Omar K. Matar Matar
Imperial College London

Date submitted: 31 Jul 2020 Electronic form version 1.4