Abstract Submitted
for the DFD20 Meeting of
The American Physical Society

An opensource tool for filtered two-fluid simulations of fluidized gas-particle flows OLIVER DAISEY, FEDERICO MUNICCHI, University of Nottingham, JAN HENDRIK CLOETE, SINTEF Industry — In the past decades, computational fluid dynamics (CFD) tools based on the two-fluid (Euler-Euler) approach have been developed on the basis of the kinetic theory of granular flows (hence, assuming a local homogeneously cooling state). More recently, filtered two-fluid models have emerged, incorporating the effects of mesoscale structures (clusters or bubbles) within their constitutive laws and enabling the use of coarser grids. These approaches bear numerous similarities with the Large Eddy Simulation (LES) methods employed in single-phase CFD. However, they are generally more complicated due to the multiphase nature of the problem. In this work, we present a numerical solver based on the opensource finite volume library OpenFOAM that solves the filtered two-fluid equations for fluidized gas-particle flows using a wide range of diverse models developed in recent literature. We illustrate the phase coupling algorithm and discuss the implementation of anisotropic drag and pressure forces, as well as detailing the practical implementation of different classes of models (dynamic, with or without the solution of additional transport equations). Finally, we compare the predictions from different constitutive models against experimental results.

Federico Municchi
University of Nottingham

Date submitted: 07 Aug 2020 Electronic form version 1.4