Modelling Flow in a Fluidized Bed

Aryan Gupta
Sardar Vallabhbhai National Institute of Technology, Surat

Under the Guidance
of
Dr. Raghvendra Gupta
Professor at Indian Institute of Technology, Guwahati

Nikhil Chitnavis
PHD at Indian Institute of Technology, Madars

ABSTRACT

This work presents a numerical simulation of gas–solid flow dynamics in a bubbling fluidized bed using the open-source CFD package OpenFOAM. The Eulerian–Eulerian two-fluid approach, implemented via the "twoPhaseEulerFoam" solver which models both phases as interpenetrating continua.

A 2D rectangular fluidized bed geometry discretized using a uniform mesh generated by *blockMesh* is simulated for inlet velocities of 0.38 m/s and 0.46 m/s. The study systematically investigates the effects of grid size, time step, and boundary conditions on solution stability and accuracy. Solid volume fraction profiles of particles, extracted using ParaView, are validated against experimental data from Taghipour et al. (2005) and Liu et al. (2014).

Key findings reveal that drag models and boundary conditions critically influence bed hydrodynamics. Mesh sensitivity analysis demonstrates that finer grids (0.003 m) and less time steps improve accuracy and produce symmetrical profiles, but at a higher computational cost. This study underscores the importance of selecting appropriate drag models, boundary conditions, and numerical parameters for accurate and efficient fluidized bed simulations. The results presented here outperform those of prior studies in terms of agreement with experimental data.

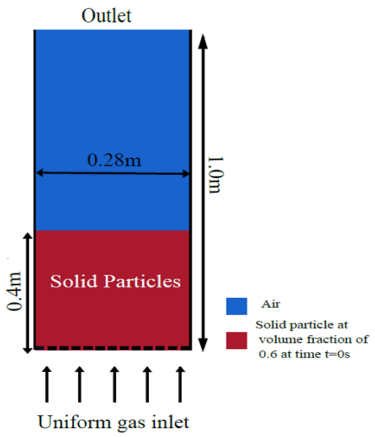

Figure 1: Schematics of a Fluidized bed