Euler-Lagrangian Simulation of Multiphase Plumes in Stratified Flows

***Ruo-Qian Wang[1], Rui Sun[2], and †Heng Xiao[2]**

[1]Department of Civil and Environmental Engineering, University of California, Berkeley, CA, USA.
[2]Department of Aerospace and Ocean Engineering, Virginia Tech, Blacksburg, VA, USA

*Presenting author: rqwang@berkeley.edu
†Corresponding author: hengxiao@vt.edu

Abstract

Multiphase plumes in stratified environment are often present in natural processes and environmental engineering. They were used to prevent ice in harbors, to aerate and destratify lakes and reservoirs, and to inject CO_2 particles into deep sea for carbon sequestration. They can also be found in dredging operations, deep-sea blowouts of oil and gas, and sediment-carrying discharge at river mouths and wastewater outfalls. It is necessary to understand its physics to improve our capability to predict and use them.

Euler-Lagrangian coupling scheme is a high accurate method to simulate multiphase flow. It resolves the fluid phase using Computational Fluid Dynamics (CFD) and resolves the solid phase by Langrangian tracking of individual particles (also known as Discrete Element Method or DEM). Since particles are treated individually, this method can easily realize particle collision and polydispersion (non-uniform particle size). An open-source Euler-Lagarngian multiphase flow solver has been developed for non-buoyant flows, i.e. SediFoam. The purpose of the present study is to extend the solver to buoyant flows and reproduce the particle-laden jet in a stratified environment.

The solver is developed under the framework of coupling a CFD software, OpenFOAM and a DEM software LAMMPS. The continuous phase of the flow is simulated by a sub-solver based on the buoyantBoussinesqPimpleFoam, which is a transient finite volume solver for incompressible turbulent buoyant Boussinesq flow; whereas the disperse phase is treated by Lagrangian Tracking Model sub-solver, which involves modeling the highly accurate drag force, buoyancy force, fluid stress, added mass, and lift forces. The two sub-solvers are coupled by 1) interpolation schemes to sample local flow field to drive the particle motion, and 2) a Particle-Souce-in-Cell (PSI) technique to transfer particle induced flow back to flow field. In this arrangement, the particles are allowed to be simulated in a curvilinear grid; and the grid resolution can be reduced to a level lower than the particle size. In addition, the Lagrangian Particle Tracking is descretised semi-implicitly to achieve a high stability in the simulations of the dispersed phase.

A particle-laden jet is reproduced to validate the present algorithm. The results will be compared with the existing experimental results from the literature. The centerline and the spreading features of the jet will be presented as well as the turbulence characteristics.

Keywords: Multiphase flow, openfoam, lammps, CFD-DEM, jet and plumes, sediment transport