Euler-Lagrangian Simulations of Multiphase Plumes for Environmental Engineering

Ruo-Qian Wang ‡† and E Eric Adams

1 Department of Civil and Environmental Engineering, Massachusetts Institute of Technology, Cambridge, MA 02139

March 5, 2011

Bubble plumes, sediment release, environmental dredging, oil spill

Multiphase plumes, formed when a continuous phase fluid is mixed with immiscible buoyant particles, droplets or bubbles, are common in natural and engineered systems and span multiple time and length scales, e.g. volcanic eruptions, piston engine fuel injection, and oil spills in ocean. The present study aims to track the fate of sediment released to the ocean during land reclamation to minimize sand loss and associated turbidity and environmental impact in the water column.

To numerically investigate this phenomenon, an Euler-Lagrangian solver is developed under the framework of open-source CFD software – OpenFOAM. The continuous phase of the flow is simulated by a sub-solver based on pisoFoam, which is a transient finite volume solver for incompressible turbulent flow; whereas the dispersed phase is treated by a Lagrangian tracking model sub-solver, which involves accurate modeling of drag forces, buoyancy forces, fluid stress, added mass, and lift forces. The two sub-solvers are two-way coupled by 1) interpolation schemes to sample local flow fields to drive the particle motion, and 2) a Particle-Source-in-Cell (PSI) technique to transfer particle induced flow back to flow field. In this arrangement, the particles are simulated in a curvilinear grid mesh, and the grid resolution can be reduced to a level lower than the particle size. In addition, the Lagrangian Particle Tracking is discretized semi-implicitly to achieve high stability in the simulations of the dispersed phase.

To validate the present algorithms, two extreme cases are simulated: a particle-laden momentum jet, and a buoyant plume formed from air bubbles. The computational domain is discretized by a structured, curvilinear cylinder mesh, which is stretched to concentrate more grid points in the sharp shear layer (shown in Fig 1). Particles/bubbles of uniform size are injected randomly within a circle, either with or without initial velocity, in order to reproduce the orifice. The Dynamic Mixed subgrid model is used for turbulence closure.

Initial results compare favorably with existing experimental data. More quantitative results will be presented in the poster.

‡ Corresponding Author: Ruo-Qian Wang (rqwang@mit.edu)
REFERENCES


