I. Motivation

Multiphase buoyant intrusions are common in natural and engineered systems and span multiple time and length scales. Typically, the plumes contain solid particles (e.g. volcanic eruptions, sediment disposal in open waters), liquid droplets (e.g. piston engine fuel injection), bubbles (e.g. lake aeration), or their mixtures (e.g. oil spill in oceans).

The present study is aiming to track sediment fate in the operation of land reclamation to minimize sand loss through suspension in the water column, i.e. turbidity, during the dumping process. It is critical to conserve the sediment resources as well as ensure the accurate placement of the dumped material. The reduction of turbidity will also avoid excessive environmental impact to the surrounding ambient waters.

The present project is motivated by several issues associated with the dumping process, and in particular the initial plume formation phase. Through numerical simulations, we are exploring:

1) Buoyancy effects on rate of penetration and flow patterns with plumes.
2) Distribution of the dispersed phase within the plume
3) Development of improved models describing sediment plume behavior to guide engineering practices.
4) Providing a bridge between analytical models and experimental results.

II. Numerical Methods

Grids and boundary conditions

- Continuous phase: Large-Eddy Simulations with the sub-grid model of Dynamic Mixed Model, built-in filter functions, and inter-phase coupling terms.
- Dispersive phase:
  - Lagrangian Tracking: \[ m_0 \frac{d \theta}{dt} = F_{\text{drag}} + F_{\text{buoyancy}} + F_{\text{gravity}} + F_{\text{inertial}} + F_{\text{collision}} \]
  - \( F_{\text{buoyancy}} \) - gravity and buoyancy force; \( F_{\text{drag}} \) - drag; \( F_{\text{inertial}} \) - added mass
- Empirical coefficients
  - Drag coefficient: \[ C_D = \begin{cases} \frac{24}{Re} & \text{if } Re \leq 800, \\ 0.44 & \text{if } Re > 800 \end{cases} \]
  - Lift coefficient: \( C_L = 0.05 \)
  - Virtual mass Coefficient: \( C_p = 0.5 \)
- Inter-phase coupling: \[ F_{\text{drag}} = F_{\text{drag}} - F_{\text{buoyancy}} - F_{\text{inertial}} - F_{\text{collision}} \]

Computing environment

- The code is developed based on an open-source Computational Fluid Dynamics software OpenFOAM.
- Parallel calculations are made at the High Performance Computing cluster in Nanyang Technological University, Singapore

III. Steady State Plumes

<table>
<thead>
<tr>
<th>Nozzle diameter D (mm)</th>
<th>Particle diameter Dp (µm)</th>
<th>Inflow Velocity ( \text{U}_0 ) (m/s)</th>
<th>Volume Fraction ( q_v )</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.08</td>
<td>500</td>
<td>1.66</td>
<td>2.4%</td>
</tr>
</tbody>
</table>

Centerline Characteristics

Cross-sectional Characteristics

Comparison between simulations and experiments along the centerline (left: volume fraction; middle: particle velocity; right: fluid velocity)*

Comparison between simulations and experiments along cross-sections (left: volume fraction; middle: particle velocity; right: fluid velocity)*

IV. Particle Clouds

Comparison between simulations and experiments of particle cloud patterns (upper: numerical simulations; lower: experimental photos)**

Comparison between simulations and experiments of particle cloud evolution (left: height of the cloud center; middle: settling velocity of the cloud center; right: horizontal extent of the cloud)**

V. Conclusions and Future Plans

- The present numerical method can convincingly reproduce dilute particle-laden jets.
- The reproduced dense particle cloud is comparable to the experiment.
- CFD-DEM will be used to incorporate the particle-particle collision and other sophisticated models in order to achieve a more accurate result.

Acknowledgement

The research described in this project was funded in whole or in part by the Singapore National Research Foundation (NRF) through the Singapore-SUT Alliance for Research and Technology (SMART) Center for Environmental Sensing and Modeling (CENSAM).

References