Hypopycnal and Hyperpycnal Particle-Laden Gravity Currents

Jorge Silvestrini[1], *Bruno Farenzena*[1], *Felipe Schuch*[1], *Ezequiel Francisco*[1], *Ricardo Frantz*[1], *Karina Ruschel*[1] & *Sylvain Laizet*[2]

[1]Escola Politécnica, Pontifícia Universidade Católica do Rio Grande do Sul, Porto Alegre, Brasil
[2]Turbulence, Mixing and Flow Control Group, Department of Aeronautics, Imperial College, London, UK

Gravity currents are common in nature, either in the atmosphere due to cold and sea-breeze fronts, in avalanches of airborne snow or debris flows, or in the ocean due to turbidity currents and river plumes. They are created by the release of a fixed-volume or a sustained-flux of a ligther (hypopycnal) or heavier (hyperpycnal) suspension into an ambient fluid. In this talk, we will discuss about some aspects of particle-laden flows beginning with a linear stability analysis (LST) of hypopycnal shear flow. Then, we will show some recent results of plunging plumes and, finally, we will focus on the dynamics of underflows in channel and basin configurations.

Regular flooding events can produce hypopycnal river plumes that remain at the surface of the ocean and can transport fine material for up to tens of kilometers from the river mouths. This sediment transport from buoyant plumes to ocean waters occurs due to a vertical sediment flux generated by turbulent mixing. The turbulent mixing between the fresh river water and the ocean environment can produce non-uniform sedimentation, and this can be linked directly to gravitational instabilities usually described as *convective sedimentation*. An extensive LST investigation is initially performed, where the dominant unstable modes are identified.

Particle-laden flows entering a lighter ambient fluid (hyperpycnal flows) can plunge at a sufficient depth and their deposits might preserve important records across a variety of climatic and tectonic settings. Several studies were carried out in order to understand this complex phenomenon using classic box models, natural deposit records and laboratory experiments. Spatio-temporal results generated by turbulence-resolving 3D simulations of poly-disperse hyperpycnal plumes, over a bed slope with up to three granulometric fractions, will be presented. The temporal evolution of plunge position, necessary depth for plunging and entrainment coefficient will be discussed and compared with existing experimental data.

Finally, after the plunge, a hyperpycnal flow is formed. In the last two decades, high-resolution numerical simulations of gravity currents for moderate Reynolds numbers have been performed by means of Direct Numerical Simulations (DNS), with significant contributions to the understanding of the spatio-temporal evolution. Interestingly, only few studies of gravity currents are based on Large Eddy Simulation (LES). This study presents an original way to perform high-fidelity simulations of particle-laden gravity-driven for high Reynolds numbers. Following our previous works exclusively based on DNS (Re up to $O(10^4)$), we now focus on Implicit Large Eddy Simulations (ILES) in order to reach Reynolds numbers of $O(10^6)$. In this approach, the effect of the non-resolved small-scale structures is dealt with targeted numerical dissipation introduced in the discretization of the viscous term. Two flow configurations are studied: the well-known channelized finite-volume case and a less conventional basin set-up, in which the flow can freely evolved in the streamwise and spanwise directions. This study focuses on instantaneous visualizations of the lobe-and-cleft structures at the head of the current but also on the sedimentation rate, suspended mass, energy budget and the deposition patterns. Comparison with results from classical sub-grid models for LES will be also shown to get an in-depth analysis of their behaviour for these particular transient particle-laden flows. For all the numerical simulations, the incompressible Navier-Stokes equations and the scalar transport equation, under the Boussinesq approximation are solved on a Cartesian mesh with the high-order flow solver *Incompact3d*.