# DIRECT NUMERICAL SIMULATIONS OF PARTICLE-DRIVEN GRAVITY CURRENTS IN A BASIN CONFIGURATION

L.F.R. Espath[1], L.C. Pinto[1], S. Laizet[2], J.H. Silvestrini[1]
[1]*Faculdade de Engenharia, Pontificia Universidade Católica do Rio Grande do Sul, Av. Ipiranga 6681, 90619-900 Porto Alegre - RS, Brasil*

[2]*Turbulence, Mixing and Flow Control Group, Department of Aeronautics, Imperial College London London, SW7 2AZ, United Kingdom*

*Abstract* Three-dimensional highly resolved Direct Numerical Simulations (DNS) of a particle-driven gravity current are presented for the lock-exchange problem in a basin configuration. Two Reynolds numbers are investigated in order to identify differences in the flow structures and dynamics. For this numerical study, we limit our investigations to gravity currents over a flat bed in which density differences are small enough for the Boussinesq approximation to be valid. The concentration of particles is described in an Eulerian fashion by using a transport equation combined with the incompressible Navier-Stokes equations, with possibility of particles deposition but no erosion and re-suspension. For this original flow configuration, it is found that the Reynolds number has a strong influence on the flow, in particular on the deposition pattern over the flat bed. Furthermore, we found out that the well-known lobe-anf-cleft patterns at the head of the current have a different shape than what is usually observed for the lock-exchange problem in a channel configuration. The curvated shape of the front has a significant twisting effect on the structures at the head of the current.

## INTRODUCTION

Gravity currents developing along a solid surface when a fluid is evolving in a another fluid with a lower density are exhibiting a highly complex dynamic with the occurrence of the well known lobe-and-cleft patterns at the head of the front. This leading zone is followed by a region of mixing with intense spanwise Kelvin-Helmholtz vortices at the interface between the current and the ambient fluid. One of the key features of gravity currents is the lobe-and-cleft structures located at the head of the current. They have been under scrutiny for a very long time with many experimental investigations and more recently with numerical investigations based on Direct Numerical Simulations [1]. However, most of those investigations were performed in a channel flow configuration for which the flow is constraint by the wall in the spanwise direction. For the first time to our knowledge, very acccurate DNS are performed in a basin configuration where the current can freely evolved in the spanwise direction as shown in Figure 1. In the present numerical investigations, the main aim is to better understand the spatio-temporal evolution of a gravity current for various Reynolds numbers for the lock-exchange basin configuration. The focus will be on the lobe-and-cleft structures at the front of the current and on the deposition pattern at different stages of the evolution of the gravity current.

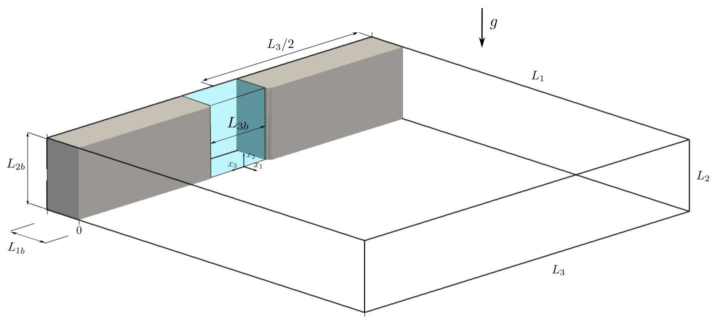


**Figure 1.** Schematic view of the initial configuration of the lock-exchange flow problem for the basin configuration.

## NUMERICAL METHODS AND FLOW CONFIGURATION

The particle are enclosed in a small portion of the domain $L_{1b} \times L_{2b} \times L_{3b}$ (see Figure 1) separated by a gate from the clear fluid. Then, the gate is removed and the concentration of particles flows due to gravity, without any constraint in the spanwise direction. We choose half of the box height as the characteristic length scale $h = L_b/2$. The velocity scale is defined by the buoyancy velocity as $u_b = \sqrt{g'h}$. The reduced gravitational acceleration $g'$ is defined as $g' = g(\rho_p - \rho_0)c_i/\rho_0$ where $\rho_p$ and $\rho_0$ are the particle and clear fluid density respectively, $g$ is the gravitational acceleration and $c_i$ is the initial volume fraction of the particles in the lock. With these two scales and the kinematic viscosity of the fluid $\nu$, we can define the Reynolds number as $Re = u_b h/\nu$. All variables are made dimensionless using $c_i$, $h$ or/and

$u_b$. The computational domain $L_1 \times L_2 \times L_3 = 12 \times 2 \times 12$ is discretized with $1201 \times 193 \times 1201$ mesh points. For the $Re = 5000$ case, 289 mesh points are used in the vertical direction and two simulations are performed, up to $t = 4$ with a reduced computational domain ($L_1 = L_3 = 6$) and then up to $t = 20$ with $L_1 = L_3 = 12$ in order to capture the smallest features of the flow during the early stage of the release. Boundary conditions for the velocity and the concentration can be found in [1]. The deposition of particles is ensured via a 1D convection equation at the bottom of the computational domain. We solve the incompressible Navier-Stokes equations using the Boussinesq approximation along with a scalar transport equation on a Cartesian mesh with our high-order flow solver **Incompact3d** which is based on sixth-order compact schemes for spatial discretization and a third order Adams-Bashforth scheme for time advancement. Full details about the code can be found in [2]. The size of the present simulations are such that we have no alternative but to use the parallel version of this code [3]. In order to design the basin configuration (see Figure 1), we used a customized Immersed Boundary Method in order to impose a zero velocity boundary condition at the wall of the lock and a no-flux boundary condition for the concentration of particles.

## RESULTS

The sudden release of the particle-fluid mixture along the left wall of the computational domain leads to the streamwise and spanwise evolution of a gravity current into the ambient fluid over the entire computational domain. Figure 2 (left) shows the turbulent structures illustrated by the $Q-$criterion for the simulation with $Re = 5000$. It can be seen that the current can be divided into two main regions, a very large one with very intense twisted lobe-and-cleft structures at the head of the current and a smaller secondary one with relatively larger elongated vortices. The curved shape of the current is responsible for the twisted lobe-and-cleft structures. One can also noticed that the density and size of the lobe-and-cleft structures vary radially with a smaller shape for the structures aligned with the spanwise direction and a longer shape for the structures aligned with the streamwise direction. Figure 2 (middle and right) shows the deposit map for the two simulation at $t = 20$. For both simulations the deposition is maximum very close to the lock in agreement with what is obtained in a channel configuration [1]. It is clear that the Reynolds number is strongly influencing the deposition patterns. For the lowest Reynolds number, the deposition pattern is still symmetric and it is possible to identify very large lobe-and-cleft structures at the head of the current. For the highest Reynolds number, cracks can be seen, preferencially in the streamwise direction and the lobe-and-cleft structures are much thiner than in the low Reynolds number simulation.

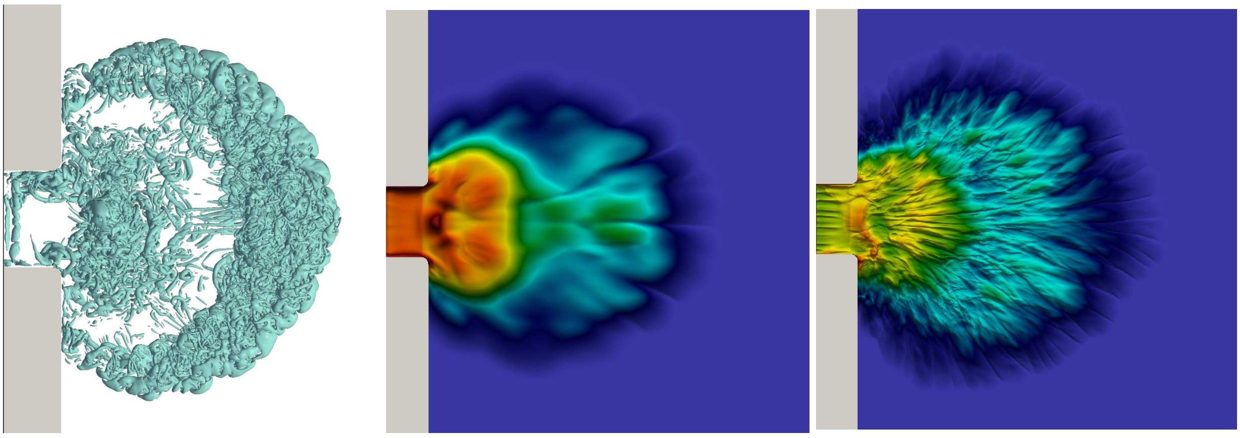

**Figure 2.** Turbulent structure of the gravity currents illustrated by the $Q-$criterion for the isovalue $Q = 1$ at $t = 20$ for the simulation with $Re = 5000$ (left). Corresponding Deposit maps at $t = 20$ for the simulation with $Re = 1000$ (middle) and for the simulation with $Re = 5000$ (right) where the red color corresponds to the maximum deposition.

In the final paper, we will further investigate this flow configuration and statistical results will be presented such as the temporal evolution of the front location, the sedimentation rate, the friction velocity as well as the complete energy budget. Comparisons will be made with different spanwise dimension $L_{3b}$ for the basin and with a channel flow configuration.

## References


[1] L.F.R. Espath, L.C. Pinto, S. Laizet, and J.H. Silvestrini. Two- and three-dimensional direct numerical simulation of particle-laden gravity currents. *Computers & Geosciences*, **63**:9–16, 2014.

[2] S. Laizet and E. Lamballais. High-order compact schemes for incompressible flows: A simple and efficient method with the quasi-spectral accuracy. *J. Comp. Phys.*, **228**:5989–6015, 2009.

[3] S. Laizet and N. Li. Incompact3d: A powerfull tool to tackle turbulence problems with up to O($10^5$) computational cores. *Int. J. Heat and Fluid Flow*, **67**:1735–1757, 2011.