Computers and Fluids 123 (2015) 23–31

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Direct numerical simulation of cylindrical particle-laden gravity currents

N. Zgheib [a,b], T. Bonometti [b,c], S. Balachandar [a,*]

[a] Department of Mechanical and Aerospace Engineering, University of Florida, Gainesville, FL 32611,USA
[b] Université de Toulouse, INPT, UPS, IMFT (Institut de Mécanique des Fluides de Toulouse), UMR 5502, Allée Camille Soula, F-31400 Toulouse, France
[c] CNRS; IMFT; F-31400 Toulouse, France

ARTICLE INFO

Article history:
Received 27 October 2014
Revised 5 August 2015
Accepted 5 September 2015
Available online 12 September 2015

Keywords:
Direct numerical simulations
Particle-laden flow
Turbidity currents

ABSTRACT

We present results from direct numerical simulations (DNS) of cylindrical particle-laden gravity currents. We consider the case of a full depth release with monodisperse particles at a dilute concentration where particle–particle interactions may be neglected. The disperse phase is treated as a continuum and a two-fluid formulation is adopted. We present results from two simulations at Reynolds numbers of 3450 and 10,000. Our results are in good agreement with previously reported experiments and theoretical models. At early times in the simulations, we observe a set of rolled up vortices that advance at varying speeds. These Kelvin–Helmholtz (K–H) vortex tubes are generated at the surface and exhibit a counter-clockwise rotation. In addition to the K–H vortices, another set of clockwise rotating vortex tubes initiate at the bottom surface and play a major role in the near wall dynamics. These vortex structures have a strong influence on wall shear-stress and deposition pattern. Their relations are explored as well.

1. Introduction

Particle driven currents are a special form of gravity currents in which the density difference is caused by the suspension of particles within an interstitial fluid forming the current. If the mixture density of such a suspension is larger than that of the ambient fluid, it will advance primarily horizontally as a turbidity current [12,17]. Turbidity currents are inherently more complex than homogeneous conservative currents because the density of the current (and consequently the density difference between the current and the ambient) may vary temporally and spatially as a result of the settling and entrainment of particles. The effective settling speed of particles, for example, may depend on particle Reynolds number, particle flocculation, and interaction with surrounding turbulence. On the other hand, if the current is traveling fast enough over an erodible bed, it may entrain particles causing it to move even faster and consequently entrain more particles in a self-reinforcing cycle.

Particulate gravity currents are observed in many industrial, environmental, and geological situations. Owing to their destructive nature, turbidity currents constitute a major factor in the design of underwater structures such as pipelines and cables [22]. In an industrial context, they are essential for transporting sediments that may contain pollutants. Furthermore, they are responsible for the formation of submarine canyons as well as for sedimentation transport into the deep oceans.

Particulate constant volume releases [1–3,6,12,14,18] have been studied. However, these finite releases are invariably dominated by fronts. Often in turbidity currents, it is very important to look at the body of the current after the head had long moved away. Experimental and computational investigation of the body is somehow harder and is usually investigated through constant flux currents [11,16,20,23]. In the present context, we explore a finite-volume cylindrical release of particle-laden fluid in clear ambient surrounding. We wish to identify the dynamics of the current, specifically the three-dimensional propagation and vortical structures of the current. Here we are only concerned with deposition and neglect the effects of resuspension. In reality, resuspension of particles may play a role, but the mechanisms of re-suspension are not fully understood and models of resuspension rate remain empirical with large uncertainties (see [26]). In order to make the problem simple and manageable, we look only at the problem of deposition.

Predicting the deposition pattern or the sediment erosion resulting from a turbidity current necessitates a good understanding of the mechanism of sediment transport and particle deposition, which are highly dependent on the dynamics of the current, the level of turbulence, and the fluid-particle interaction. As a result, a great level of simplification is generally taken, usually through depth averaging, when studying particulate-driven currents. Some of the models include the Box Model [2,9], which is a simple and fast way to model the extent, speed, and sedimentation rate of turbidity currents. The Box Model is not directly derived from the Navier–Stokes equations; however, it considers the current to evolve with negligible entrainment through a series of height diminishing concentric cylinders for an axisymmetric lock release configuration. In addition to depth averaging, no radial variation is allowed.

* Corresponding author. Tel.: +1 352 392 8909; fax: +1 352 392 1071.
E-mail address: bala1s@ufl.edu (S. Balachandar).

http://dx.doi.org/10.1016/j.compfluid.2015.09.001

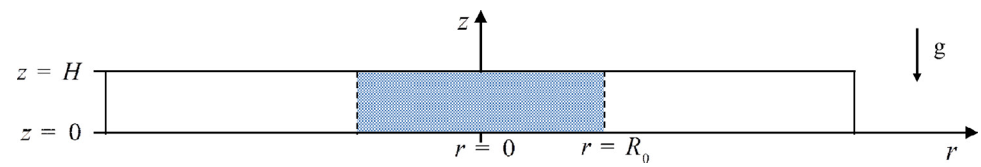

Fig. 1. Side view of the initial setup of the cylindrical lock-exchange flow inside a rectangular box of size $L_x \times L_y \times H = 30 \times 30 \times 1$. Initially, a cylindrical gate of radius $R_0 = 2$ placed at the center of the domain separates particle-laden fluid from the ambient clear fluid. Once the gate is lifted, the simulation starts and the particle-laden fluid begins to intrude horizontally into the ambient.

A more complex model is based on the Shallow Water equations [3,24,8], which are derived by vertically averaging the Navier–Stokes equations under the assumption of high length-to-thickness aspect ratio. However, because of the variable volume fraction of the current, an equation of particle conservation is further required. Such models do not usually account for sediment entrainment on the basis that the velocities are insufficient to lift up particles, however the flow is considered to be sufficiently energetic so that turbulent mixing maintains vertically uniform properties.

Most research on axisymmetric particle-laden gravity currents has mainly revolved around the early experiments of Bonnecaze et al. [3] and theoretical models mostly based on the Box Model and Shallow Water equations [24,13]. Our objective in this study is to consider a scenario that is similar to what has been investigated experimentally and examine it through direct numerical simulations. Highly resolved simulations for cylindrical density-driven finite-release currents have been investigated in the past with results comparing favorably with experiments [4]. Here we consider direct numerical simulations of particle-laden currents resulting from the release of an initial cylindrical fluid-particle mixture.

The DNS will allow us to explore the three-dimensional structures of the current from iso-surfaces of density and iso-surfaces of the swirling strength that show the intensity and structure of the coherent eddies. In many applications, these large scale vortical structures will play an important role in the erosion and resuspension of particles by locally modifying the shear stress at the bottom wall. They also play an important role in the deposition of particles by transporting low particle concentration fluid (particle-laden current mixed with ambient) from the current's top layers towards the bottom wall and consequently decreasing the local settling rate. This study will be limited to finite-releases of full-depth cylindrical gravity currents with dilute concentrations of monodisperse particles. The paper is arranged as follows. The mathematical formulation is outlined Section 2. In Section 3, we present our simulation results and compare, where possible, to previous experimental and theoretical data. Finally, main conclusions are presented in Section 4.

2. Mathematical formulation

A side view of the problem setup is shown in Fig. 1. Initially, a cylindrical gate separates a relatively heavier (compared with the ambient) particle-laden fluid of initial density $\rho_{c0} = (\rho_p - \rho_a)\phi_0 + \rho_a$ in its interior from the surrounding clear ambient fluid of density ρ_a. Both fluids are initially at the same level and occupy the entire height of the domain (see Fig. 1). Here, ρ_p represents the density of suspended particles, and ϕ_0 is the initial volume fraction occupied by those particles.

Our focus is to simulate buoyancy driven flows with dilute suspensions, where particle-particle interactions may be neglected. We consider monodisperse particles whose size is much smaller than characteristic length scale H of the problem. The particle-laden solution will be treated as a continuum and a two-fluid formulation is adopted. We follow Cantero et al. [6] by implementing an Eulerian–Eulerian model of the two-phase flow equations. The model involves (i) mass (ii) and momentum conservation equations for the continuum fluid phase, (iii) an algebraic equation for the particle phase where the particle velocity is taken to be equal to the local fluid velocity and an imposed settling velocity derived from the Stokes drag force on the particles, (iv) and a transport equation for the volume fraction (particle phase). The non-dimensional system of equations read

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\frac{D\mathbf{u}}{Dt} = \phi \mathbf{e}^g - \nabla p + \frac{1}{Re}\nabla^2 \mathbf{u} \tag{2}$$

$$\mathbf{u}_p = \mathbf{u} + \mathbf{u}_s \tag{3}$$

$$\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \mathbf{u}_p) = \frac{1}{Sc\,Re}\nabla^2 \phi \tag{4}$$

Here $\mathbf{e}^g$ is a unit vector pointing in the direction of gravity. Unless otherwise stated, all parameters are non-dimensionalized. The height H of the domain is taken as the length scale, $U = \sqrt{g'_0 H}$ as the velocity scale, $T = H/U$ as the time scale, ρ_a as the density scale, and $\rho_a U^2$ as the pressure scale. The reduced gravitational acceleration is defined as $g'_0 = (\rho_p - \rho_a)\phi_0 g/\rho_a$. We denote by $\mathbf{u}_p$ and ϕ the velocity and the volume fraction of the particle phase, respectively. $\mathbf{u}$ and p correspond to the velocity and total pressure of the continuum fluid phase, respectively. The settling velocity $\mathbf{u}_s$ is determined from the Stokes drag force on spherical particles with small particle Reynolds numbers. Here, the density of particles is assumed to be appreciably larger than that of the ambient fluid such that the dominant force on the particle is the Stokes drag. The Reynolds and Schmidt numbers in Eqs. (2) and (4) are defined as

$$Re = UH/\nu, \quad Sc = \nu/\kappa \tag{5}$$

In the above, κ and ν represent the molecular diffusivity and kinematic viscosity of the ambient (interstitial) fluid, respectively.

The simulation is carried out inside a rectangular box of dimensions $L_x \times L_y \times H = 30 \times 30 \times 1$ using a spectral code that has been extensively validated [4,5]. Periodic boundary conditions are imposed along the sidewalls for the continuous and particle phases. No-slip and free-slip conditions are imposed for the continuous phase along the bottom and top walls, respectively. Mixed and Neumann boundary conditions are imposed for the particle phase at the top and bottom walls, which translate into zero particle net flux and zero particle resuspension, respectively.

$$\boxed{\text{at } z=1 \quad \frac{1}{Sc\,Re}\frac{\partial \phi}{\partial z} - \mathbf{u}_s \phi = 0,} \quad \boxed{\text{at } z=0 \quad \frac{\partial \phi}{\partial z} = 0.} \tag{6}$$

To be more explicit, the condition (6) at the bottom wall allows for a non-zero advective flux which is due to the sedimentation of particles. Since we assume zero resuspension, the sediment flux across the bottom boundary is the deposit. For more details and a discussion about the choice of the present boundary conditions, the reader is referred to Cantero et al. [6]. We present results from two simulations that differ solely by the Reynolds number. The details