European Journal of Mechanics B/Fluids 53 (2015) 68–84

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

Large-eddy simulations of turbidity plumes in crossflow

Boudewijn Decrop [a,b,*], Tom De Mulder [a], Erik Toorman [c], Marc Sas [b]

[a] Hydraulics Laboratory, Dept. of Civil Eng., Ghent University, Sint-Pietersnieuwstraat 41, B-9000 Ghent, Belgium

[b] Coastal and Estuarine Section, IMDC, Coveliersstraat 15, B-2600 Antwerp, Belgium

[c] Hydraulics Laboratory, Catholic University of Leuven, Kasteelpark Arenberg, B-3001 Heverlee, Belgium

ARTICLE INFO

Article history:
Received 13 June 2014
Received in revised form
6 February 2015
Accepted 31 March 2015
Available online 11 April 2015

Keywords:
Large eddy simulation
Buoyant jets
Sediment transport
Plumes
Turbidity

ABSTRACT

Two-phase Large-Eddy Simulations (LES) of turbidity plumes in a crossflow are presented, representative for mixtures of water and fine sediment particles released through an overflow pipe of a dredging vessel. The model was tested based on experimental data of vertical plumes in a still environment as well as of plumes in crossflow. Simulations include the effect of the wake of a schematised hull shape on the dispersion and turbulent structure of the plume, as a schematisation of plumes released from dredging vessels. Criteria for a minimum fraction of resolved turbulent kinetic energy were used to evaluate the so-called completeness of the LES simulations. It is shown that while the grid resolution is a factor three to six lower compared to earlier Direct Numerical Simulation (DNS) simulations, comparable results on turbulent structures and turbulent kinetic energy can be obtained. Results of mean trajectory and plume dispersion show good agreement with experimental data. The different types of turbulent structures found in our experiments as well as in literature are reproduced by the model. It is shown that a release point shortly upstream of the schematised hull's stern causes the upper fringes of the plume to be torn off in cases with a relatively strong crossflow, which enhances mixing and hampers the containment of the material transported by a plume.

1. Introduction

Our specific field of interest is the highly-concentrated sediment-water plume released through the overflow shaft of a trailing suction hopper dredging vessel, e.g. Smith and Friedrichs [1]. The latter type of dredger is widely applied worldwide and allows dredging while sailing. By means of a suction pipe lowered to the sea- or riverbed, a sediment-water mixture is pumped into the hopper. Whereas the coarser sediment particles settle in the hopper, the finest sediment fractions flow overboard with the excess water through a vertical overflow dropshaft, the outlet of which is flush with the keel of the vessel's hull. As a consequence, a negatively buoyant plume of water and fine sediment particles is released vertically below the vessel. Because of the sailing speed of the vessel and/or the ambient currents, the generated plumes are subjected to a crossflow. To enable proper assessment of the environmental impact of the plumes it is important to be able to predict its fate and dispersion in the water column [2]. The subsequent interaction of the plume with the bed, is outside the scope of this paper.

The objective of this paper is to develop a numerical model capable of accurately predicting the mean trajectory and the turbulent dispersion of a negatively buoyant sediment plume in a crossflow. Both the flow field of a plume in crossflow as well as the flow field of a plume in crossflow influenced by a backward-facing step (BFS) are analysed thoroughly. To this end, a Large Eddy Simulation (LES) approach is selected. The model will be set up and validated based upon general plume data reported in literature, as well as scaled-down laboratory experiments with sediment plumes by the authors (Decrop et al.) [3]. In the latter experiments, the hull of the dredging vessel was highly schematised as a quasi prismatic box with a given draft. The numerical model will be validated both without and with the schematised hull. It was intended to set up a model with acceptable process times to enable future upscaling of the model to prototype scale, including realistic dredging vessel geometry and propeller action.

A buoyant jet or forced plume is a plume in which both an initial momentum flux and a buoyancy flux govern the flow up to a certain distance from the release point [4,5]. Pure plumes and pure jets, to the contrary, are only forced by a mass density difference and an initial momentum flux, respectively.

For round buoyant jets with top-hat velocity profile, initial volume, momentum and buoyancy fluxes can be written, respectively,

* Corresponding author at: Hydraulics Laboratory, Dept. of Civil Eng., Ghent University, Sint-Pietersnieuwstraat 41, B-9000 Ghent, Belgium.
E-mail address: boudewijn.decrop@ugent.be (B. Decrop).

http://dx.doi.org/10.1016/j.euromechflu.2015.03.013

as:

$$Q_0 = \frac{\pi}{4} D^2 W_0 \quad (1)$$

$$M_0 = \frac{\pi}{4} D^2 W_0^2 = W_0 Q_0 \quad (2)$$

$$B_0 = g_0' \frac{\pi}{4} D^2 W_0 = g_0' Q_0 \quad (3)$$

where D is the exit pipe diameter, W_0 is the (uniform) exit velocity and $g_0' = g(\Delta\rho/\rho)$ is the reduced gravity of the mixture being discharged in an ambient fluid.

In the case presented in this paper, the buoyancy force is generated by the presence of particles in a mixture. Assuming the mass density of the fluid phase in the mixture is equal to the mass density of the ambient fluid, g_0' can be written as

$$g_0' = g C_0 \frac{\rho_p - \rho_w}{\rho_w} \quad (4)$$

where C_0 is the initial particle volume concentration (m^3/m^3), ρ_p is the mass density of the particles and ρ_w is the mass density of the fluid phase in the mixture and of the receiving ambient fluid (both tap water in the experiments carried out by the authors).

The quantities Q_0, M_0 and B_0 are considered the primary variables in the study of turbulent jets, plumes and forced plumes [5]. In the remainder of this article, the forced plumes studied will be referred to as 'plumes'.

Plumes released from a circular pipe or orifice exhibit self-similarity in the region past the so-called zone of flow establishment, typically at about seven pipe diameters [4,5]. The profiles of flow velocity and tracer concentration collapse when normalised by the appropriate parameters and they follow a Gaussian distribution. This allows researchers to compare experimental and numerical results of plumes of different Q_0, M_0 and B_0. In this work, the self-similarity property will be used to validate the LES model by simulating a vertical plume and comparing the results with measurements by Decrop et al. [3], which are shown to be consistent with single-phase plume experiments by Papanicolaou and List [6], Shabbir et al. [7] and Dai et al. [8].

The flow in plumes released vertically in a crossflow is characterised by large-scale turbulent structures. These are related to the blending of initial vertical momentum with crossflow horizontal momentum (counter-rotating vortex pair, CRVP) as well as Kelvin–Helmholtz instabilities resulting in leading-edge vortices and associated convection cells ejecting in the direction of the buoyancy force vector (e.g. Tian and Roberts [9], Cambonie et al. [10] and Diez et al. [11]). Additionally, a pattern of wake vortices similar to the vortices in the wake of a cylinder has been observed by Fric and Roshko [12] and Muldoon and Acharya [13] in the wake of a jet in cross flow. These vortices are, however, clearly distinct from a von Karman vortex street since they originate from the crossflow boundary layer. The two-liquid case of these flows has been studied intensively, whereas turbulent plumes in which negative buoyancy is created by fine particles are not very well studied. Much of the large scale flow structure is expected to be similar to the intermiscible liquids case, however, turbulent diffusion of mass is expected to behave differently in dispersed two-phase flows. Different authors have shown that the turbulent Schmidt number, Sc_t has a different average value for particulate suspensions compared to the standard value of 0.7 for intermiscible liquids diffusion [14–17]. In principle, a two-phase LES model using the mixture model for the dispersed phase of the spreading of a sediment laden jet should resolve enough of the turbulent motions to provide a spreading rate associated with the diffusivity in a sediment plume. The sub-grid scale turbulent Schmidt number is derived from test-filtered turbulent fluxes [18], as described in Section 2.1.2.

Dimensional analysis leads to two main dimensionless numbers characterising plumes released in a cross-flow, the densimetric Froude number F_Δ and the velocity ratio λ.

$$F_\Delta = \frac{W_0}{\sqrt{gD\Delta\rho/\rho_w}}; \qquad \lambda = \frac{W_0}{U_0}. \quad (5)$$

Trajectories followed by plumes in crossflow are determined by several possible flow regimes: jet regime, bent jet regime, plume regime and the bent plume regime. In some literature, e.g. Wright [19], these are referred to as momentum-dominated near field (MDNF), momentum-dominated far field (MDFF), buoyancy-dominated near field (BDNF) and buoyancy-dominated far field (BDFF) respectively, e.g. Wright [19].

The plumes studied here range within (F_Δ, λ) limits occurring in sediment plumes released from dredging vessels. Even though the initial relative density difference is usually in the order of 1–10%, the buoyancy is relatively weak compared to the crossflow in these cases, with the velocity ratio usually in the range $0.2 < \lambda < 2$. Therefore, the momentum length scale z_M (Eq. (6)) is larger than the buoyancy length scale z_B (Eq. (7)) in most cases, leading to a plume trajectory sequence MDNF–MDFF–BDFF. However, in strong cross-flow cases both z_M/D and z_B/D are around or less than unity, due to which the plume transforms very rapidly to the BDFF regime. The plume trajectory in the two main regimes occurring in the studied flows (MDFF, BDFF) scale with z_M and z_B, respectively.

$$z_M = \frac{M_0^{1/2}}{U_0} \quad (6)$$

$$z_B = \frac{B_0}{U_0^3}. \quad (7)$$

As mentioned above, many types of turbulent structures occur in plumes in crossflow. Some of the large-scale structures have a steady nature and can also be solved with a Reynolds-Averaged Navier–Stokes (RANS) model. Therefore, one could wonder why a more time-consuming LES model is used in that case. Many of the larger turbulent structures are only resolved in a time-domain model, for example buoyancy bursts referred to as cauliflowers and tracer entrained in a wake vortices (observed by Smith and Mungal [20] in a jet). Moreover, it is known that the turbulence in jets and plumes in cross-flow is anisotropic [21]. This makes the use of RANS model with an isotropic turbulence model less suited. The influence of the geometry of the surrounding walls on these structures can therefore ideally be studied using time-domain solutions. Also, future study of pulsed plumes using the presented model would require time-domain resolution in any case [22,23]. Frequency analysis of the turbulent fluctuations of velocity and tracer concentration is possible using LES results. Experiments in the past show both $-5/3$ and -3 exponential spectral energy cascade laws [8].

LES simulations of jets and plumes in a still environment have been reported repeatedly in the past [24–26]. LES simulations of a dispersed two-phase plume in still environment were described by Dimitrova et al. [27]. A limited number of studies, though, reports LES simulations of jets or single phase plumes in a cross-flow, e.g. by Yuan et al. [28], Recker et al. [29] and Coussement et al. [22]. Muppidi and Mahesh [30,31] performed Direct Numerical Simulations (DNS) simulations of transverse jets and passive scalar transport therein. To the best of our knowledge, LES of two-phase, small Stokes number particle plumes in crossflow did not receive much attention yet. In order to set up an LES model of good quality, two requirements are defined. The first one is to ensure that the percentage of resolved turbulent kinetic energy (TKE) is sufficient. Pope [32] defined a criterion for the minimum amount of turbulence to be resolved in an LES model. The idea of complete