International Journal of Multiphase Flow 51 (2013) 55–64

Contents lists available at SciVerse ScienceDirect

International Journal of Multiphase Flow

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

The effect of large-scale turbulent structures on particle dispersion in wall-bounded flows

Matteo Bernardini *, Sergio Pirozzoli, Paolo Orlandi

Department of Mechanical and Aerospace Engineering, University of Rome 'La Sapienza', Via Eudossiana 18, 00184 Rome, Italy

ARTICLE INFO

Article history:
Received 31 July 2012
Received in revised form 22 October 2012
Accepted 27 November 2012
Available online 14 December 2012

Keywords:
Lagrangian particle tracking
Large-scale structures
Direct numerical simulation
Particle-laden wall-bounded flows
Couette flow

ABSTRACT

The effect of large-scale turbulent structures on the motion of heavy particles in wall-bounded turbulent flows is investigated by mining a direct numerical simulation database of particle-laden turbulent Couette flow, and comparing the results with a turbulent Poiseuille at similar friction Reynolds number. It is shown that the large-scale structures found in the core of the Couette flow have an influence on the turbophoretic mechanism, leading to different distributions of the particle concentration. The main differences in the two flows are observed in the spatial distribution of the suspended phase, which is found to be strongly dependent on the structure of the underlying streamwise velocity field. In addition to the standard particle streaks in the inner layer, spaced at 100 wall units, typical of the Poiseuille flow, in the Couette case particles with non-negligible inertia respond to the large-scale structures of the core, by organizing themselves into large-scale rows whose typical spanwise separation is of the order of 4–5 channel half-height.

1. Introduction

The dispersion and deposition of solid particles in turbulent wall-bounded flows is a topic of fundamental interest in a wide range of applications, which embrace a large variety of sciences. Typical examples include dispersion of pollutants in the atmosphere, transport and sedimentation processes in environmental systems, and deposition of particles and droplets in steam turbines, to cite but a few (Guha, 2008). As a consequence, fundamental studies investigating the motion of particles in canonical wall-bounded turbulent flows like pipes and channels are abundant in literature, both from the experimental and the numerical perspective.

One of the most interesting and investigated phenomena of particle dispersion in wall-bounded flows is the turbophoresis (Caporaloni et al., 1975), which consists in the preferential accumulation of heavy particles in the near-wall region, caused by gradients of turbulence intensities (Reeks, 1983). This behavior is a consequence of the inertial characteristics of particles, and the degree of the near-wall accumulation depends on the particle response time, which is a measure of the particle inertia, and denotes the time scale with which any slip velocity between the particles and the fluid is equilibrated. It has been found (Kaftori et al., 1995a; García et al., 1995) that particle motion in the boundary layer, as well as the entrainment and deposition processes, are mainly driven by the action of coherent structures, which cause the formation of particle streaks near the wall, creating suitable conditions for particle entrainment or deposition. Since inertial particles are often concentrated below the low-speed streaks, where they remain trapped for a long time, the average particle velocity is lower than the fluid's (Kaftori et al., 1995b).

Significant advances in the analysis of the particle motion in wall turbulence have been gained in the last two decades thanks to direct numerical simulation (DNS) coupled with Lagrangian particle tracking, used for the first time by McLaughlin (1989) to simulate aerosol particle deposition in a turbulent channel flow at low Reynolds number. Later numerical simulations (Pedinotti et al., 1992; Pan and Banerjee, 1996; van Haarlem et al., 1998; Uijttewaal and Oliemans, 1996) have essentially confirmed the scenario observed in experiments, and have helped to explain in detail the particle transfer mechanisms. A comprehensive analysis of the turbophoretic processes and of the connections between particle motions and near-wall turbulent structures is provided by Marchioli and Soldati (2002), who point out that particle transfer in wall-bounded flows is mainly driven by strong spatially-coherent ejection and sweep events of the inner-layer, which act bringing particles toward and away from the wall, favouring segregation in the viscous sublayer, where particles tend to cluster below the low-speed streaks. While this has become a popular way to explain turbophoresis mechanisms, it should be noted that there is also evidence of the opposite and works can be found in literature where turbophoresis is observed in inhomogeneous flows that do not contain coherent structures (Iliopoulos et al., 2003; Skartlien, 2007; Arcen and Tanière, 2009).

* Corresponding author.
E-mail address: matteo.bernardini@uniroma1.it (M. Bernardini).

0301-9322/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2012.11.007

Looking at the available literature, it appears that numerical studies on particle dispersion and deposition are mainly limited to low Reynolds number channel and pipe flows. These studies have mainly focused on the role played by the turbulent structures populating the near-wall region, whereas the possible influence of large-scale structures from the outer layer has not been established yet. However, evidence of large-scale accumulation patterns have been recently reported by Sardina et al. (2011) by performing DNS of particle-laden turbulent channel with two computational boxes with different size. Discrepancies in the wall accumulation level up to 10% were reported, which those authors attributed to the different behavior of the large-scale motions of the carrier phase caused by numerical constraining effects. However, that study was also limited to low Reynolds number (the friction Reynolds number being $Re_\tau = 180$), which can mask more significant large-scale effects. This issue might be relevant in light of the relatively recent discovery (Kim and Adrian, 1999) of very large scale motions (VLSMs) in the core region of channel and pipe flows, which exert an influence on the near-wall coherent structures through the superposition of low-wavenumber energy, leaking from a secondary energy site in the logarithmic layer, whose imprint increases with Reynolds number, becoming appreciable only for $Re_\tau \geqslant 1000$ (Re_τ being the friction Reynolds number). Recent studies (Mathis et al., 2009; Bernardini and Pirozzoli, 2011) have also shown that large-scale motions are not simply superimposed on the near-wall region, and the nature of the inner/outer layer interaction is inherently nonlinear, characterized by mechanisms of amplitude modulation.

To our knowledge, a clear description of the effects of large-scale structures on the particle organization in wall-bounded flows is still lacking and it is the main goal of the present work. Unfortunately, direct numerical simulations of particle-laden high-Reynolds-number wall-bounded flows are still impracticable, also because of the very long time taken by particles to attain a steady state condition (Portela et al., 2002). As a matter of fact, while simulations of wall-bounded turbulent flows have reached moderate values of Re_τ (up to 2000) (Hoyas and Jiménez, 2006; Schlatter et al., 2009; Pirozzoli and Bernardini, 2011), the highest Reynolds number achieved in a particle-laden turbulent channel flow which the authors are aware of is $Re_\tau = 587$ (Vinkovic et al., 2011), still too low to observe significant large-scale effects (Hutchins and Marusic, 2007). Moreover, to properly take into account the large-scale structures in the computation, long and wide computational domains are needed (del Álamo et al., 2004), which makes such simulations yet more challenging. To investigate large-scale effects on particle motion we then propose to use a different flow configuration, which is inherently characterized by the presence of large-scale structures, namely the turbulent Couette flow. In this flow, large-scale motions spanning the entire channel height are known to exist, having a typical length up to 30 h and spanwise size of 4 h (Bech et al., 1995; Komminaho et al., 1996) (where h is the channel-half height). An investigation of the properties of these motions has been recently carried out by the present authors (Pirozzoli et al., 2011), who have shown that the turbulent Couette flow is characterized by the presence of inner/outer layer interaction mechanisms similar to those observed in high-Reynolds-number turbulent channels, and can be considered a useful alternative to probe large-scale effects in wall-bounded flows. In this work, we then perform a direct numerical simulation of a particle-laden turbulent Couette flow, and compare the results with those obtained for a canonical Poiseuille at the same friction Reynolds number.

The paper is organized as follows. Details on the numerical strategy for the carrier and suspended phase are provided in Section 2. Results of the simulations are discussed in Section 3, both in terms of instantaneous visualizations and particle statistics. Conclusions are finally provided in Section 4.

2. Numerical methodology

2.1. Simulation of the carrier phase

The numerical solver integratePs the incompressible Navier–Stokes equations for a divergence-free velocity field

$$\begin{aligned} &\frac{\partial u_j}{\partial x_j} = 0 \\ &\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} - H\,\delta_{i1}, \end{aligned} \tag{1}$$

where H is the (negative) pressure gradient required to maintain a constant flow rate, u_i ($i = 1,2,3$) is the component of the velocity vector in the ith direction, ρ is the (constant) density, p is the pressure, and ν is the kinematic viscosity. The equations are discretized in an orthogonal coordinate system (the streamwise, wall-normal and spanwise direction are indicated indifferently with x_1, x_2, x_3 or x, y, z) and solved by a second-order finite-difference method relying on a staggered arrangement of the flow variables, which guarantees that kinetic energy is globally conserved in the limit of inviscid flow and exact time integration. Time advancement is achieved by means of a third-order low-storage Runge–Kutta algorithm coupled with the second-order Crank–Nicolson scheme, combined in the fractional step procedure as in Le and Moin (1991), whereby the convective and diffusive terms are treated explicitly and implicitly, respectively. Further details concerning the numerical method and the implementation strategy can be found in Orlandi (2000).

The computational domain is a box with size $L_x = 12\pi h$ in the streamwise direction and $L_z = 4\pi h$ in the spanwise direction, h being the channel-half height. No-slip boundary conditions are applied at the two walls (located at $y = \pm h$) and periodicity is exploited in the homogeneous directions (x and z). Two direct numerical simulations have been performed, whose parameters are reported in Table 1, together with information on the computational resolution. The first one is a canonical turbulent Poiseuille flow, hereinafter referred to as P flow, performed at friction Reynolds number $Re_\tau = h\ u_\tau/\nu = 183$. The second computation is a plane turbulent Couette flow ($Re_\tau = 167$), labeled as C flow in the following, performed by setting $H = 0$ in Eq. (1) and moving the two walls in opposite direction. Note from Table 1, that a large domain has been selected for both the simulations, this choice being dictated by the need to accommodate the large structures of the Couette flow.

It is also worth noting that the Poiseuille flow simulation has been performed in the convective reference frame for which the bulk velocity is zero at each streamwise station, i.e. the frame in which the net streamwise mass flux is zero. This property is automatically satisfied in the Couette flow setup, with the two walls moving in the opposite direction at the same velocity. In addition to allowing a larger computational time-step, this choice reduces the dispersion errors induced by the finite-difference discretization, leading to results close to those obtained with pseudo-spectral methods (Bernardini et al., 2013).

The mean velocity profiles and turbulent fluctuation intensities are displayed in Fig. 1, reported in wall units (the superscript + is used throughout the paper to denote normalization by u_τ and ν) and compared with the Poiseuille flow data by del Álamo et al.

Table 1
Computational parameters for particle-laden turbulent Poiseuille and Couette flow.

Case	Re_τ	L_x/h	L_z/h	Δx^+	Δz^+	N_x	N_y	N_z	T^+
P flow	183	12π	4π	8.9	4.5	768	128	512	31230
C flow	167	12π	4π	8.2	4.1	768	128	512	37000