International Journal of Multiphase Flow 56 (2013) 172–183

Contents lists available at SciVerse ScienceDirect

International Journal of Multiphase Flow

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

Particle and droplet deposition in turbulent swirled pipe flow

Francesco Zonta, Cristian Marchioli, Alfredo Soldati *

Centro Interdipartimentale di Fluidodinamica e Idraulica, Università degli Studi di Udine, Udine, Italy
Department of Fluid Mechanics, CISM, 33100 Udine, Italy

ARTICLE INFO

Article history:
Received 25 March 2013
Received in revised form 7 June 2013
Accepted 8 June 2013
Available online 17 June 2013

Keywords:
Turbulence
Swirling flows
Numerical simulation
Lagrangian tracking

ABSTRACT

In this work we study deposition of particles and droplets in non-rotating swirled turbulent pipe flow. We aim at verifying whether the capability of swirl to enhance particle separation from the core flow and the capability of turbulence to efficiently trap particles at the wall can co-exist to optimize collection efficiency in axial separators. We perform an Eulerian–Lagrangian study based on Direct Numerical Simulation (DNS) of turbulence, considering the effect of different swirl intensities on turbulence structures and on particle transfer at varying particle inertia. We show that, for suitably-chosen flow parameters, swirl may be superimposed to the base flow without disrupting near-wall turbulent structures and their regeneration mechanisms. We also quantify collection efficiency demonstrating for the first time that an optimal synergy between swirl and wall turbulence can be identified to promote separation of particles and droplets.

© 2013 Elsevier Ltd. All rights reserved.

1. Introduction

In-line separation of particles and droplets (simply referred to as particles hereinafter) from a gas stream is an important problem in many engineering applications, such as dedusting and demisting in process, oil and gas industries (Peng et al., 2004; Klujszo et al., 1999; Soldati et al., 1997). A widely used solution is to exploit the larger inertia of particles and propel them toward the wall via a suitable rotating motion of the mean flow. This motion can be generated by changes in flow geometry, as for instance by static vanes in axial separators for gas cleaning (Peng et al., 2004; Gomez et al., 2004), or by curved ducts in gas-cleaning cyclones and in hydro-cyclones for separation of liquid–liquid mixtures (Delfos et al., 2004).

In this work we address the problem of gas–solid/gas–liquid separation in axial tubes equipped with swirl vanes (swirl tubes, see Nieuwstadt and Dirkzwager, 1995). Design of such devices crucially depends on the interactions among swirling motions, near-wall turbulence and particles, which give rise to complex dynamics that involve turbulence forcing by swirl, and may have a strong effect on the near-wall turbulence regeneration mechanisms. Previous studies (Orlandi and Fatica, 1997; Eggels, 1994; Pettersson et al., 1998) have shown that swirling motions induced by a pipe rotating about its axis can influence turbulence to the point of flow re-laminarization. Such findings promoted a large effort dedicated to identify sustainable strategies for drag reduction by interrupting turbulence regeneration cycle, ultimately disrupting wall structures. However, the very same wall structures found detrimental for drag reduction are found beneficial for particle separation by enhancing deposition and wall trapping (Marchioli and Soldati, 2002; Soldati, 2005; Soldati and Marchioli, 2009). Therefore, for optimal separation design, characteristics of swirl and wall structures should be tuned to ensure fast transfer to the wall (due to swirl-induced centrifugation) and efficient trapping (due to turbulence-induced preferential concentration): once confined in the wall layer, particles may be removed using suction slots or similar filtration systems (Nieuwstadt and Dirkzwager, 1995).

Many detailed numerical (Orlandi and Fatica, 1997; Eggels, 1994; Pettersson et al., 1998) and experimental (Kitoh, 1991; Steenbergen and Voskamp, 1998; Parchen and Steenbergen, 1998; Pashtrapanska et al., 2006) studies on swirl-induced turbulence modification are available. However, most of them focus on drag reduction application and hence examine swirling motions generated by a rotating pipe wall (see Speziale et al., 2000 for a review) or by an imposed circumferential pressure gradient (Nygard and Andersson, 2010). The influence of swirl on turbulence is less clear when it is produced in the core of the pipe, e.g. by static tilted vanes. These two instances of swirled flow have a fundamental difference: swirl is centripetal in rotating pipes, where it leads to transport of vorticity and displacement of vortices away from the wall, but centrifugal in swirled tubes. Evidence of boundary layer thickening for centripetal swirl has been demonstrated (Orlandi and Fatica, 1997; Eggels, 1994). For centrifugal swirl we may hypothesize a qualitatively correspondent boundary layer thinning, which however was not investigated in detail previously.

* Corresponding author at: Centro Interdipartimentale di Fluidodinamica e Idraulica, Università degli Studi di Udine, Udine, Italy. Tel.: +39 0432 558020; fax: +39 0432 558027.

E-mail address: soldati@uniud.it (A. Soldati).

0301-9322/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2013.06.002

The first objective of this work is therefore to analyze the flow field in an axial swirl tube separator. We know from literature that (i) swirl is beneficial in quickly propelling particles to the wall (Kitoh, 1991; van Esch and Kuerten, 2008); (ii) wall turbulence is efficient in trapping particles at the wall (Marchioli and Soldati, 2002; Soldati, 2005); and (iii) high swirl intensity can destroy near-wall turbulence structures (Orlandi and Fatica, 1997; Eggels, 1994): here, we want to assess the possibility of tuning swirl characteristics to prevent disruption of near-wall structures due to excessive centrifugal forcing. This is relevant for real-life separation devices, e.g. vertical separators, where near-wall turbulence and not just centrifugation is crucial for separation. Second objective is to examine the behavior of particles and droplets in such flow field, focusing on separation and collection efficiency. Few studies on similar problems are available, which were recently performed by Kuerten and co-workers (Kuerten et al., 2005; van Esch and Kuerten, 2008), albeit for the case of rotational phase separators, where swirl is generated using a rotating cylindrical filter element.

For both objectives, we performed an Eulerian–Lagrangian parametric study of particle dispersion in swirled pipe flow via Direct Numerical Simulation (DNS) of turbulence. DNS ensures the most accurate numerical prediction of particle trajectories. This is a crucial aspect when evaluating separation processes. As demonstrated by Marchioli et al. (2008a), Large Eddy Simulation (LES) is not yet a full alternative to DNS for two-phase flow simulations: Because of filtering, an intrinsic feature of LES, proper modeling of subgrid turbulence is required in the equation of particle motion to avoid time-accumulating filtering errors on trajectories and consequent underestimation of separation efficiency. Such models, however, are currently unavailable (Bianco et al., 2012).

To our knowledge, this is the first fundamental study based on DNS of particle-laden swirled flow in a non-rotating pipe. Results discussed in this paper provide a proof of concept, based on sound physical arguments, that the synergy between swirl-induced centrifugal mechanisms and turbulence-induced trapping mechanisms can be exploited to enhance particle separation in bounded turbulent flows. Though fundamental, this study has strong applicative implications since the flow configuration mimics the behavior of a swirl tube.

2. Problem formulation and numerical methodology

The physical problem investigated in this work considers turbulent flow of air (with density $\rho = 0.965$ kg/m^3 and kinematic viscosity $\nu = 1.57 \times 10^{-5}$ m^2/s) in a pipe with radius $R = 0.025$ m. The flow Reynolds number is $Re = U_{cP}R/\nu = 5000$ with $U_{cP} = 3.14$ m/s the centerline velocity of the laminar Poiseuille flow. The corresponding shear Reynolds number is $Re_\tau = u_\tau R/\nu = 171$ with $u_\tau = \sqrt{\tau_w/\rho} \simeq 0.108$ m/s the shear velocity (τ_w being the mean shear stress at the wall). These values match those considered in previous numerical studies dealing with the same flow configuration (Nygard and Andersson, 2010; Kuerten et al., 2005; van Esch and Kuerten, 2008; Orlandi and Fatica, 1997; Eggels, 1994). The flow configuration is sketched in Fig. 1 together with the cylindrical reference frame. The computational domain consists of an upstream pipe that has axial length $L = 10R$, followed by a downstream pipe of equal radius and length $L = 20R$, in which the swirling motion takes place. For both pipes turbulence is simulated using $N_r \times N_\theta \times N_z = 88 \times 129 \times 129$ (resp. 257) nodes in the radial, azimuthal and axial directions, respectively. Nodes are equally spaced in z and θ, with non-uniform radial refinement (hyperbolic tangent method) close to the wall. The grid resolution is $\Delta z^+ = 13.3$ in the axial direction, while it ranges from $(r \cdot \Delta\theta)^+_{min} = 0.028$ to $(r \cdot \Delta\theta)^+_{max} = 8.25$ in the circumferential direction. The first grid node away from the wall is located at $r^+ = 0.42$, ensuring a resolution sufficient to describe all flow length scales, the Kolmogorov length scale being equal to approximately 1.85 wall units (Marchioli et al., 2003). The time step size is $dt^+ = 0.0117$, sufficient to cope with stability requirements (Courant number restriction). No-slip and no-cross boundary conditions are enforced at the pipe wall for the fluid velocity components.

The numerical method proposed by Verzicco and Orlandi (1996) was adopted for the simulation of both pipes, with the obvious specification of different inlet/outlet boundary conditions. This method solves for the continuity and Navier–Stokes equations in cylindrical coordinates (not shown here for brevity) using a second-order finite-difference discretization for the spatial derivatives, while time derivatives are computed using a third-order low-storage Runge–Kutta scheme for the non-linear terms and an implicit Crank–Nicholson scheme for the viscous terms. The pressure–velocity coupling is handled using a fractional step method which ensures incompressibility at each substep of the Runge–Kutta scheme. Code validation can be found in Marchioli et al. (2003). Flow statistics up to second order show excellent agreement with those of Eggels et al. (1994) and Fukagata and Kasagi (2002). Variables are expressed in outer units, obtained using R and U_{cP} as reference length and velocity for normalization. The corresponding space and time scales are $\mathcal{L} = R$ and $\mathcal{T} = R/U_{cP}$, respectively. In the following, we will also refer to variables expressed in wall units, which will be identified by superscript "+". Wall units are obtained using $\mathcal{L} = \nu/u_\tau$ and $\mathcal{T} = \nu/u_\tau^2$ for normalization.

To reproduce the axially-decaying spin imparted to the fluid through the inclined vanes in swirl tubes, DNS of spatially-developing turbulent flow in the downstream pipe is performed. Inflow conditions for the fluid velocity are obtained from an auxiliary DNS of swirl-free particle-free periodic flow in the upstream pipe: Fully-developed fluid velocity profiles at the outlet section of the periodic flow domain (indicated as u_θ^{PPF} hereinafter) are superimposed to a prescribed Batchelor Vortex (BV) profile (Muntean et al., 2005) which mimics the swirling motion imparted by the tube vanes. The BV produces a centrifugal (rather than centripetal) forcing on the flow and is characterized by the following azimuthal velocity:

$$u_\theta^{BV}(r) = \frac{\Omega R_c^2}{r}\left[1 - \exp\left(-\frac{r^2}{R_c^2}\right)\right], \qquad (1)$$

which depends on two parameters: the characteristic angular velocity of the vortex, Ω, and the characteristic vortex radius, R_c, a measure of the vortex core radial extent. This type of swirl, which is not boundary-driven and generates a rotating fluid core of axially-decaying intensity, was investigated in several experimental studies (Kitoh, 1991; Steenbergen and Voskamp, 1998; Parchen and Steenbergen, 1998) at Reynolds numbers much higher than those considered here. The only numerical work we are aware of was performed by Muntean et al. (2005) who, however, used a Reynolds stress model to simulate turbulence and a $k - \epsilon$ model to specify inlet turbulent quantities. The resulting velocity distribution at the inlet of the spatially-developing flow domain is:

$$u_\theta(r, \theta, z = 0, t) = u_\theta^{PPF}(r, \theta, t) + u_\theta^{BV}(r). \qquad (2)$$

The present approach proved to be highly accurate, with little or no adjustment of the solution near the inlet boundary and no transient convected downstream (Lund et al., 1998; Sbrizzai et al., 2009). In this study, we fixed $R_c = 0.003$ m and considered three different values for Ω yielding different values of the swirl number S (defined as ratio of axial flux of angular momentum to axial flux of axial momentum): $\Omega_0 = 0$ s^{-1}, corresponding to the swirl-free motion (referred to as $S_0 = S(\Omega_0) = 0$ case); $\Omega_L = 375$ s^{-1}, corresponding to

Download English Version:

https://daneshyari.com/en/article/667325

Download Persian Version:

https://daneshyari.com/article/667325

Daneshyari.com