Physics Letters A 375 (2011) 839–844

Contents lists available at ScienceDirect

Physics Letters A

www.elsevier.com/locate/pla

The effects of flow structure and particle mass loading on particle dispersion in particle-laden swirling jets

Nan Gui [a,b], JianRen Fan [a,*], Song Chen [a]

[a] State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China
[b] College of Mechanical and Transportation Engineering, China University of Petroleum, Beijing 102249, PR China

ARTICLE INFO

Article history:
Received 14 October 2010
Received in revised form 21 December 2010
Accepted 22 December 2010
Available online 23 December 2010
Communicated by R. Wu

Keywords:
Swirling jets
Particle dispersion
Flow structure
Mass loading
Momentum coupling

ABSTRACT

A direct numerical simulation of particle dispersion in particle-laden swirling jets issued into a rectangular container through a round nozzle is carried out. The swirl number is $S = 1.42$ when the bubble vortex breakdown takes place. Two cases are simulated for comparison, i.e. five types of particles with Stokes numbers $St = 0.01$, 0.1, 1, 10 and 100 respectively under the same flow rate, and four types of particles with $St = 0.5$, 1, 5 and 10 respectively under the same mass loading. After simulation, it is found that the rectangular flow domain induces an important modification to the flow structure. It influences the dispersion characteristics in the peripheral cross area, forming a centrosymmetric dispersion of particles in the cross-sectional area. A quantitative analysis of the non-uniform particle dispersion is carried out. Moreover, the effect of mass loading on particle dispersion is explored and explained. It indicates the correlation between the inter-phase moment coupling and particle mass loading via the change of probability density function of the inter-phase velocity difference. Heavy mass loading causes an insufficient inter-phase momentum transport and the worse dispersion of large particles than that of small mass loading.

1. Introduction

Gas–solid dispersed flow is of great importance for both scientific researches and engineering applications, e.g. the dispersion characteristics of coal particle is important for transportation in ducts and combustion efficiency in combustion devices, etc. There exhibits a variety of interesting phenomena, e.g. preferential accumulations [1–3]. Many studies have shown that the preferential particle concentration is associated with large-scale structures which disperse particles effectively and dominate particle motions [4–6].

The particle dispersion in swirling flows is important for swirling combustion systems and gas–solid cyclone separators. With regard to particle concentration in swirling flows, Wicker and Eaton [7] showed the presence of large vortex structures which have similar effects on particle distribution. Apte et al. [8] and Gui et al. [9] carried out respectively a large-eddy simulation and a direct numerical simulation of swirling particle-laden flows in a coaxial-jet combustor corresponding to a previous experiment by Sommerfeld and Qiu [10,11], focusing on the particle dispersion characteristics and distribution patterns, etc. In these studies, particle dispersion is shown to be related closely to the large-scales vortex structure as well as the particle response characteristics. The particle response property is appropriately characterized by the Stokes number, which plays an important role in preferential concentration. Heavy particles tend to accumulate in regions where the strain rate dominates over vorticity, whereas light particles tend to accumulate in regions of intense vorticity [12].

Although the effects of larger vortex and the response property (characterized by the Stokes number) on particle dispersion are well explained, the effect of mass loading on particle dispersion has not been well explored. Moreover, the structure of large vortex may be influenced by the configuration of the flow domain. Does the modification of flow structure affect the particle dispersion, and how does it work? These issues have not been well explained yet. Thus, the present study will focus on the effects of flow structure and mass loading on modification of the characteristics of particle dispersions.

For numerical simulation of dispersed flows, the Lagrangian point-force/particle method has been used for a long history, which uses either a one-way coupling approach [13,14] or a two-way coupling approach [15]. In this approach, the dispersed phase is considered as discrete points and traced under the Lagrangian framework by solving the equations of motion, without taking into account the effects of particle volumes, such as wakes after particles. The carrier phase is simulated under the Eulerian approach by some types of CFD technique, such as RANS, LES or DNS, etc. The present study will use the point-particle tracking technique to

* Corresponding author. Fax: +86 0571 87991863.
E-mail address: fanjr@zju.edu.cn (J. Fan).

0375-9601/$ – see front matter
doi:10.1016/j.physleta.2010.12.049

simulate the disperse phase and the direct numerical simulation method to simulate the carrier phase. A particle-laden swirling flow with a large swirl number and moderate Reynolds numbers is carried out with reference to a previous experiment by Billant et al. [16]. The case with the particle number flow rate fixed and the case with the particle mass loading fixed are simulated to explore the effects of mass loading and large scales of flow structure on the behavior of particle dispersion.

2. Numerical method

2.1. Governing equation of fluids

The governing equations of the carrier phase are the three-dimensional, time dependent, incompressible Navier–Stokes equations:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{1}{Re}\frac{\partial}{\partial x_j}\frac{\partial}{\partial x_j}u_i - \boldsymbol{f}_p \tag{2}$$

where u_i, p are fluid velocity and pressure respectively, and $\boldsymbol{f}_p$ is the drag force of particles. When the particle density is far larger than that of the fluid, the drag force is of the leading order compared to other hydrodynamic forces [9,17]. Thus, the other forces, such as virtual mass force, Saffman force, Magnus force, pressure gradient force and buoyancy forces are omitted. As a result, only the $-\boldsymbol{f}_p$ is considered here as the backward force from particle to fluid.

The dimensionless flow domain is a rectangular container, with $30d \times 10d \times 10d$ in streamwise (x), lateral (y), spanwise (z) directions respectively, where $d = 0.4$ mm is the diameter of the jet at the inlet. In this study, "d" is used as the characteristic length for non-dimensionalization of the governing equations. The jet is issued from the centre of the upstream area at $x = 0$. The inlet axial and azimuthal velocity profiles are specified according to Ref. [16]. The downstream outlet in the x-direction is the non-reflecting boundary [18]. Otherwise, the side walls in the lateral and spanwise directions are set as nonslip wall boundaries.

The Reynolds number is $Re = U_0 d/\nu = 606$ and 3000 for cases 1 and 2 respectively (Table 1), where U_0 is the inflow mean streamwise velocity and also the characteristic velocity. The swirl number is defined as the ratio of maximum azimuthal velocity to the mean streamwise velocity at the inlet $S = 2V_{\max}/U_0$, which is fixed as $S = 1.42$ here.

To solve the above equations, a total number of $384 \times 128 \times 128$ grids are used, which can resolve the scales of turbulence as fine as about $0.075d$. The Kolmogorov length scale η $(\sim (\nu^3/\varepsilon)^{1/4})$ is estimated as $\eta = 0.065d$ when $Re = 606$. Thus, the spatial resolution requirement for direct numerical simulation is met. The simulation time step is $\Delta t = 0.005$, and a total period of dimensionless time of $T = 100$ is computed.

To perform numerical solution, the finite volume method and the fractional-step projection technique [19] are applied. An explicit low-storage, third-order Runge–Kutta scheme [20] is used for time integration. A direct fast elliptic solver is used to solve the Poisson equation. The validation of the numerical method has already been done in a previous study of Ref. [21].

2.2. Motion equation of particles

In this study, focusing on the dispersion characteristics of particles, the discrete phase is assumed to be: a) a dilute flow regime, where the particle–particle collisions are omitted; b) spherical particles with uniform diameters and densities; c) as demonstrated by Gui et al. [9], the Saffman force and Magnus force are of the secondary importance when compared to the drag force, they are omitted here as well as other types of hydrodynamic forces. Hence, only the drag force is computed.

Table 1
Simulation conditions for the two cases.

Case 1: keep $\dot{n}_p$ $(=10/\text{step})$	$St = 0.01$	$m_l = 1.41 \times 10^{-5}$ (kg/kg)
	$St = 0.1$	$m_l = 4.45 \times 10^{-4}$ (kg/kg)
	$St = 1$	$m_l = 1.41 \times 10^{-2}$ (kg/kg)
	$St = 10$	$m_l = 4.45 \times 10^{-1}$ (kg/kg)
	$St = 100$	$m_l = 1.42$ (kg/kg)
Case 2: keep m_l $(= 0.134$ kg/kg$)$	$St = 0.5$	$\dot{n}_p = 268/(1$ time step$)$
	$St = 1$	$\dot{n}_p = 95/(1$ time step$)$
	$St = 5$	$\dot{n}_p = 17/(2$ time step$)$
	$St = 10$	$\dot{n}_p = 3/(1$ time step$)$

$\dot{n}_p$ the particle number flow rate (/step); m_l the particle mass loading (kg/kg).

Based on these assumptions, the motion equations of any discrete particle are solved in a deterministic way. For any particle, the particle motion equation is:

$$m_p \frac{d\tilde{\mathbf{v}}_p}{dt} = \frac{\pi d_p^2}{8} C_D \rho_f |\tilde{\mathbf{u}} - \tilde{\mathbf{v}}_p| (\tilde{\mathbf{u}} - \tilde{\mathbf{v}}_p) \tag{3}$$

where ρ_f and $\mathbf{u}$ are fluid density and velocity vector respectively. $\mathbf{v}_p$, m_p, d_p are particle velocity, mass, diameter respectively. '$\tilde{\ }$' denotes the dimensional variable. $C_D = 24f/Re_p$ is the drag coefficient. $f = 1 + 0.15Re_p^{0.687}$ [22] is the drag factor and $Re_p = |\mathbf{u} - \mathbf{v}_p| d_p/\nu$ is the particle Reynolds number. By simple deductions, Eq. (3) is reduced to

$$\frac{d\tilde{\mathbf{v}}_p}{dt} = \frac{f}{\tau_p}(\tilde{\mathbf{u}} - \tilde{\mathbf{v}}_p) \tag{4}$$

where $\tau_p = \rho_p d_p^2/(18\mu)$ is the particle aerodynamic response time. Finally, Eq. (4) is non-dimensionalized into

$$\frac{d\mathbf{v}_p}{dt} = \frac{f}{St}(\mathbf{u} - \mathbf{v}_p) \tag{5}$$

$$\frac{d\mathbf{X}_p}{dt} = \mathbf{v}_p \tag{6}$$

where $St = \tau_p/\tau_f = \tau_p/(d/U_0)$ is the Stokes number.

Initially, the particles are generated in the cross-sectional area at the jet inlet with a uniform random distribution. An initial velocity difference between the particle and fluid phase is set, i.e. the inlet velocity of particles $V_{p,0} = 0.59U_0$ here.

As aforementioned, two cases are simulated (Table 1): firstly, we kept the number flow rate of particles with different Stokes numbers at a low Reynolds number. Secondly, we kept the mass loading of particles under different number flow rates at a moderate Reynolds number. It is necessary to mention that the conditions of two cases do not match each other exactly. We tried to simulate the cases with a large range of variation of the parameters, but it is greatly restricted by the computer performance. For example, under the same mass loading, the number of particles of $St = 0.01$ and 100 may vary over a wide range of $\sim O(10^6)$. It goes beyond the computer capacity. Thus, the simulation conditions are restrained.

3. Results and discussion

3.1. The effect of flow structure on particle dispersion

At first, Fig. 1 shows the flow structure of the large vortices with regard to the vortex breakdown (case 1, at $t = 5$, $x = 10d$). It is shown that there exists an evident recirculation zone enclosed by the large-scale vortices (Fig. 1a). It is the so-called bubble vortex breakdown. Moreover, it is found from Fig. 1b that