International Journal of Heat and Mass Transfer 64 (2013) 155–161

Contents lists available at SciVerse ScienceDirect

International Journal of Heat and Mass Transfer

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

Direct numerical simulation of a two-phase three-dimensional planar jet

Kun Luo [a], Nan Gui [b], Jianren Fan [a,*], Kefa Cen [a]

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

ARTICLE INFO

Article history:
Available online 11 May 2013

Keywords:
Direct numerical simulation
Particle dispersion
Coherent structures
Particle statistics
Three-dimensiional planar Jet

ABSTRACT

To investigate dispersion and statistics of heavy particles in three-dimensional well-developed turbulent shear flows, direct numerical simulation is used to study a particle-laden, spatially-evolving planar jet with a moderate Reynolds number of 3000. The governing equations of the gas phase are solved by the fractional-step projection schemes with finite volume method. The particles are traced in the Lagrangian framework based on one-way coupling. The instantaneous distribution of heavy particles at intermediate Stokes numbers has non-uniform clustering spatial structure, which can be better characterized by the correlation dimension. However, from the statistical point of view, the particle dispersion and particle dynamics are found to be non-linearly monotonously dependent on the particle Stokes number when the flow is well-developed. The larger Stokes number leads to the higher ensemble-averaged slip velocity, the higher ensemble-averaged particle kinetic energy as well as the higher ensemble-averaged particle Reynolds number with approximate power scaling laws. In addition, it is demonstrated that the Stokes number has a profound effect on inter-particle relative velocity.

1. Introduction

As one of the basic free shear flows, turbulent jets can be used to evaluate turbulence models and explore turbulence physics. Multiphase turbulent jets even exist widely in engineering applications, such as pneumatic transport, coal combustion and aerosol reaction. To predict and control the coherent structures and particle dispersion in the flows is of great importance for optimized design and efficient applications.

Chung and Troutt [1] simulated particle dispersion in an axisymmetric jet using the discrete vortex element approach. They reported that the particle dispersion extent depends strongly on the ratio of particle aerodynamic response time to the characteristic time of the jet. Aggarwal and Uthuppan [2] numerically investigated the effect of controlled excitation on particle dispersion in an axisymmetric jet and reported that the effect on particle dispersion for all forcing cases is the most significant when the Stokes numbers of particles are near unity. Mashayek and Jaberi [3] studied dispersion of solid particles in forced isotropic low-Mach-number turbulent flows by direct numerical simulation (DNS), and reported a constant ratio of the RMS Mach number to the mean Mach number for different particles. Glaze and Frankel [4] investigated the effect of dispersion characteristics on particle temperature in an idealized non-premixed reacting jet. It was found that the particle temperature behavior is a strong function of the spatial dispersion behavior. Fan et al. [5] studied particle dispersion in a two-dimensional turbulent plane jet. The local-focusing phenomenon of particle dispersion is discussed. The previous studies have demonstrated that the particle concentration, temperature and other fields are strongly affected by large-scale structures. The effect of large-scale structures on particles can be characterized in terms of the particle Stokes number, defined as the ratio of the particle response time to the characteristic time of large-scale structures. For Stokes numbers near 1, particles concentrate largely in high strain-rate regions and the cores of vortex structures are essentially devoid of particles. This organized particle distribution has been known as preferential concentration [6].

However, there is little work on the effects of three-dimensional small-scale vortex structures on particle dispersion and statistics in turbulent jets, which is important to industrial applications. In the present study, we investigate the influences of the particle Stokes number on heavy particle dispersion in a three-dimensional well-developed planar jet. The manuscript is organized as follows. In Section 2 we describe the flow configuration, the governing equations and the numerical methods. The numerical results and discussions are given in Section 3. The last section is devoted to summary and conclusions.

* Corresponding author. Tel./fax: +86 571 8795 1764.
E-mail address: fanjr@zju.edu.cn (J. Fan).

0017-9310/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijheatmasstransfer.2013.04.023

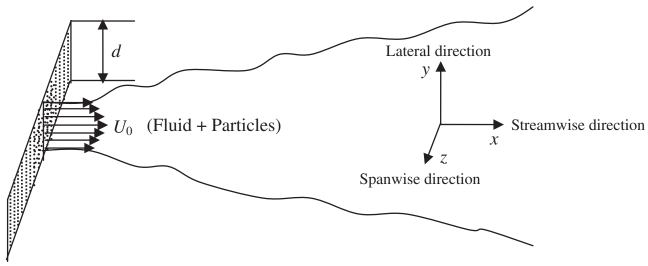

Fig. 1. Sketch of the three-dimensional gas–solid two-phase turbulent plane jet.

2. Mathematical descriptions

2.1. Flow configuration and boundary conditions

Fig. 1 shows the sketch of the three-dimensional gas–solid two-phase turbulent planar jet investigated in the present study, where U_0 denotes the bulk velocity of the jet inflow. The ratio of the nozzle width d to the initial momentum thickness θ_0 is set to be 20. The Reynolds number of the jet based on the nozzle width and the inflow velocity is 3000. The span of the computational domain in the streamwise (x), lateral (y) and spanwise (z) direction is $20 \times 20 \times 6.4d$. The fluid and the particles are injected into the computational domain through the whole nozzle.

Initially, a shear layer with the velocity profile as follows is given in the region of the nozzle width d in the flow-field:

$$u = \frac{U_0}{2} + \frac{U_0}{2}\tanh\left(\frac{y}{2\theta_0}\right) \quad v = w = 0, \tag{1}$$

where θ_0 is the initial momentum thickness. u, v and w represent the streamwise, lateral and spanwise velocities, respectively. To generate realistic turbulent inflow boundary conditions, a digital filter based method proposed by Klein et al. [7,8] is applied. This method is able to reproduces first and second order one point statistics, and has been demonstrated to be simple, flexible and accurate. In the present study, velocity fluctuations with intensity of 2% are generated and imposed on the inflow top-hat profiles. Outside the region of the nozzle width d, all the fluid velocities are set as zero. At the outflow boundary, Neumann boundary conditions for the velocity and the pressure are used. At the top and the bottom boundaries, the pressure is set to be zero and the tangential velocities are interpolated to allow mass entrainment. In the spanwise direction, periodic boundary conditions are applied.

For grid resolution, it has been shown in previous study [9] that when the grid scale used in DNS is smaller than or in the same order of the Kolmogorov micro scale η, one can get the solutions with enough precision. In the present study, the Kolmogorov micro scale η is estimated to be $0.05d$. So uniform staggered grid $\Delta x = \frac{1}{15}d = 1.33\eta$ and $\Delta z = \frac{1}{20}d = \eta$ are arranged along the streamwise and the spanwise directions. In the region of $-4.5d < y < 4.5d$, uniform grid $\Delta y = \frac{1}{30}d = 0.67\eta$ is arranged to capture small-scale structures in the core shear region of the jet. But outside of this region, stretch grid is used along the lateral direction. The total grid points $300 \times 400 \times 128 = 15.36 \times 10^6$ are used along x, y and z directions.

2.2. Governing equations for fluid

The gas phase is regarded as an incompressible Newtonian fluid. The inter-phase momentum coupling is neglected. When the body force is not included, the non-dimensional governing equations for the fluid motion can be expressed as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{2}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{1}{\mathrm{Re}}\frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

where the characteristic velocity and length scales are U_0 and d, respectively.

To solve the above equations, the finite volume method and the fractional-step projection technique [10] based on staggered Cartesian mesh are applied. Central differences are used for spatial discretization and an explicit low-storage, third-order Runge–Kutta scheme [11] is used for time integration. A direct fast elliptic solver is used to solve the Poisson equation. Considering the CFL rule, the computational time step is set to be 0.02. Total about 15 flow-through times are performed for statistics.

2.3. Governing equations for particles

The heavy particles are traced in the Lagrangian framework. Assume the particles are spherical with same diameter d_p and density ρ_p of 2000 for each case. Since the maximum value of the ratio of the particle diameter to the Kolmogorov length scale is around 0.2, the traditional point force model is used to describe particle motion. The potential forces acting on a particle, such as the pressure gradient, virtual mass, lift, and Basset forces can be neglected based on the larger particle–fluid density ratio of 2000 [12]. Then the main forces become the Stokes drag force and the gravity force. Compared to the Stokes drag force, the gravity force is negligible. As a result, the governing equation for particle motion can be written as:

$$m_p\frac{d\boldsymbol{V}}{dt} = \frac{\pi d_p^2}{8}C_D\rho_g|\boldsymbol{U} - V|(\boldsymbol{U} - V) \tag{4}$$

where C_D is the coefficient of drag force, $C_D = \frac{24}{\mathrm{Re}_p}f$. f is the modification factor, $f = 1 + 0.15\mathrm{Re}_p^{0.687}$ if $\mathrm{Re}_p \leqslant 1000$ [13], where Re_p is the Reynolds number of particles, defined as $\mathrm{Re}_p = |\boldsymbol{U} - \boldsymbol{V}|\, d_p/\nu$.

Substituting the aerodynamics response time $\tau_p = \rho_p d_p^2/18\mu$ into Eq. (3), yields the non-dimensional governing equation for particle motion:

$$\frac{d\boldsymbol{V}}{dt} = \frac{f}{\mathrm{St}}(\boldsymbol{U} - \boldsymbol{V}) \tag{5}$$

where $\boldsymbol{V}$ the velocity vector of the particle, $\boldsymbol{U}$ the velocity vector of fluid at the position of the particle. St is the particle Stokes number, defined as $\mathrm{St} = \frac{\rho_p d_p^2/(18\mu)}{d/U_0}$. The velocity and displacement of the particle can be obtained by integrating Eq. (5). Third-order Lagrangian interpolating polynomial is used to get the fluid velocity at the position of particle.

To investigate the dependence of particle dispersion and particle dynamics on the particle Stokes number, the selected Stokes number ranges from 0 to 67.75. The particles at the Stokes number of 0 represent the fluid tracer. The corresponding governing equation is

$$\frac{d\boldsymbol{X}}{dt} = \boldsymbol{U} \tag{6}$$

where $\boldsymbol{X}$ is the particle position and $\boldsymbol{U}$ is the fluid velocity at the particle position.

Before the particles are injected into the computational domain, they are distributed uniformly in the nozzle and their velocities equal the velocities of the local fluid. 231 particles are released every 10 time steps. The maximum volume fraction of the particles is about 10^{-4}, and the maximum mass loading is about 0.1. Thus, the present gas–solid two-phase jet is assumed as a dilute flow,