International Journal of Multiphase Flow 99 (2018) 454–466

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Fully resolved direct numerical simulation of multiphase turbulent thermal boundary layer with finite size particles

Junjie Xia[a,b], Luo Kun[a], Jianren Fan[a,*]

[a] State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, P.R. China
[b] Wuhan Second Ship Design and Research Institute, Wuhan 430200, P.R. China

ARTICLE INFO

Article history:
Received 1 May 2017
Revised 12 November 2017
Accepted 16 November 2017
Available online 20 November 2017

Keywords:
Turbulent boundary layer
Finite size particles
Heat transfer
Buoyancy

ABSTRACT

In order to investigate the effects of the buoyancy and finite size particles on the heat transfer process in the turbulent thermal boundary layer, we have carried out three simulations in the present paper, a single-phase neutral boundary layer, an unstable boundary layer with buoyancy effect, and a multi-phase boundary layer with thousands of finite size particles. This is the first time that particle-resolved direct numerical simulation (PR-DNS) is used in the study of thermal boundary layer. The DNS results show the turbulent statistics as well as the thermal structures. It turns out that both the buoyancy and the finite size particles will dramatically affect the turbulent statistics of the boundary layer. Detailed comparisons between the three simulations reveal that the buoyancy effect reshapes the coherent structures in the boundary layer, while finite size particles mainly induce additional disturbance all over the computational domain. Specifically, the Reynolds shear stress and the wall normal turbulent heat flux are remarkably enhanced in the log region by the effect of buoyancy. On the other hand, the finite size particles cause remarkable increment of velocity fluctuations all over the boundary layer, while have the effect of stabilizing temperature fluctuation near the wall.

1. Introduction

Turbulent boundary layer is a fundamental issue in fluid dynamics which has attracted scientists' attention for more than one hundred years. When there is temperature difference between the fluid and the wall, velocity boundary layer and thermal boundary layer develop at the same time. Multiphase turbulent boundary layer is common in many industrial and environmental areas, such as fuel combustion, pollution control, food industry and life science. But the mechanism of turbulent transport and the role dispersed phase plays in it are far from a consensus.

There are many researches on single-phase turbulent thermal boundary layer. Most of them treat temperature as passive scalar (Nagano and Tagawa, 1995; Kong et al., 2000; Li et al., 2009; Wu and Moin, 2010; Araya and Castillo, 2012). Only a few of them have taken temperature gradient buoyancy into consideration (Hattori et al., 2007; Gajusingh and Siddiqui, 2008). There are also many studies on gas-solid two-phase non-isothermal turbulent channel flow (Zonta et al., 2009; Jaszczur, 2011; Kuerten et al., 2011; Arcen et al., 2012). All of them handle fluid-particle momentum and thermal exchanges by making use of the Lagrangian point-particle model.

With the development of computer technology and numerical method, direct numerical simulations of multiphase flow with a great number of finite-size particles have become possible in the recent decades (Ten Cate et al., 2004; Lu and Tryggvason, 2006; Uhlmann, 2008; Lucci et al., 2010; Bellani et al., 2012; Ji et al., 2014; Vowinckel et al., 2014; Motta et al., 2016). However, only a few of them have taken into account of the wall effect, and none of them studied inter-phase heat transfer process.

Particle-resolved direct numerical simulation (PR-DNS) is superior to Lagrangian point-particle model in the following two ways. Firstly, in point-particle model, momentum and heat exchanges between fluid and particles are calculated by empirical formulas, which are considered as only the function of the particle Reynolds number. While the PR-DNS is designed to resolve the velocity and temperature field over the whole computational domain, hence the momentum and heat exchanges are directly governed by the Navier-Stokes and the energy conservation equations. Secondly, the PR-DNS can provide detailed information of particle wake and its interaction with the inherent coherent structures in turbulent boundary layer, which is beyond the reach of point-particle simulations.

* Corresponding author.
E-mail address: fanjr@zju.edu.cn (J. Fan).

https://doi.org/10.1016/j.ijmultiphaseflow.2017.11.012
0301-9322/

However, direct numerical simulation of turbulent boundary layer itself is a arduous task with huge consumption of computer resources. In wall units, it restricts the upper-limit of the smallest grid spacing to be $\Delta y^+ \leq 1$; while typical thickness of boundary layer is of $\delta^+ \sim O(1000)$, according to Jiménez (2012). Meanwhile, available particle resolving technologies, such as overset grid method, arbitrary Lagrangian-Eulerian, Immersed Boundary Method (IBM), Distributed Lagrange Multiplier/Fictitious Domain and Lattice Boltzmann method, require resolutions with at least $O(10)$ grids points per diameter for smooth implementation. It makes the PR-DNS of multiphase turbulent boundary layer even more resource consuming.

Benefitting from the well validated high efficiency ghost-cell based high-order IBM method developed by Xia et al. (2014, 2015), the present paper accomplishes the first attempt to simulate thousands of finite-size particles in a spatio-temporal developing turbulent thermal boundary layer. Buoyancy effect is also considered. Under the arrangement of the present study, particles-fluid momentum and thermal interactions are directly resolved, instead of using point-particle model. Since the point-particle approach is only valid for particles sufficiently small compared to the Kolmogorov length scale. In the present study, the grid size is of the Kolmogorov length scale. Moreover, the point particle model only gives a rough estimation of the average inter-phase hydrodynamic force and heat transfer rate (Xia et al., 2017). Furthermore, the finite size effect of particles, such as the wakes and their interaction with coherent structures in the turbulent boundary layer, can also be revealed in the present study.

The motivation of the present study lies in two aspects. One is to inspect the capacity of the ghost-cell based IBM method on simulating fully resolved multiphase turbulence in bounded area, and prepare for the future work of fully resolved coal combustion simulation. The other is to investigate the influences of buoyancy and finite size particle on the statistics, structures and development of the thermal turbulent boundary layer.

The structure of the paper is arranged as follows. Section 2 will give a brief introduction of numerical strategies for solving governing equations of the fluid and solid phase. In Section 3, the problem description and the case definitions are provided. Then, Section 4 consists of main results and discussion. Finally, concluding remarks are given in Section 5.

2. Governing equations

In the present study, we deal with constant properties viscous incompressible Newtonian fluid. The dimensionless conservation equations for mass, momentum and thermal energy are solved:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + \frac{1}{Re_{\delta_{2,in}}} \nabla^2 \mathbf{u} - \frac{Gr_{\delta_{2,in}}}{Re^2_{\delta_{2,in}}} \theta \frac{\mathbf{g}}{g} \tag{2}$$

$$\frac{\partial \theta}{\partial t} + \mathbf{u} \cdot \nabla \theta = \frac{1}{Re_{\delta_{2,in}} \cdot Pr} \nabla^2 \theta \tag{3}$$

where $\mathbf{u} = (u, v, w)$ is the dimensionless velocity vector and P is the dimensionless pressure, by taking the free-stream velocity U_∞ as characteristic velocity, the momentum thickness at the inflow plane of the boundary layer $\delta_{2,\,in}$ as characteristic length scale. The dimensionless temperature is defined as $\theta = (T - T_w)/(T_\infty - T_w)$, where T_∞ is the free-stream temperature, T_w is the wall temperature and T is the dimensional fluid temperature.

The three dimensionless characteristic numbers in the governing equations are Reynolds number $Re_{\delta_{2,in}} = (\rho_0 \cdot U_\infty \cdot \delta_{2,in})/\mu$, Prandtl number $Pr = (c_P \cdot \mu)/k$, and Grashof number $Gr_{\delta_{2,in}} = g\beta(T_\infty - T_w)\rho_0^2 \delta_{2,in}^3/\mu^2$. From them, two additional characteristic numbers can be derived: Peclet number $Pe = Re \cdot Pr$ and Richardson number $Ri_{\delta_{2,in}} = Gr_{\delta_{2,in}}/Re^2_{\delta_{2,in}}$. And ρ_0, μ, β, c_P, k are fluid density, dynamic viscosity, thermal expansion coefficient, specific heat and coefficient of thermal conductivity respectively. Then the fluid kinematic viscosity is $\nu = \mu/\rho_0$. $\mathbf{g}$ is the vector of gravitational acceleration, and its magnitude $g = |\mathbf{g}|$.

The pressure-Poisson equation derived by applying the divergence operator to the momentum equations replaces the continuity Eq. (1). Then Eq. (1) is satisfied indirectly through the solution of the pressure equation. Eqs. (2) and (3) are integrated in time using a four-stage fourth-order Runge-Kutta method with the third-order Adams-Bashforth method for convection terms and Crank-Nicolson method for diffusion terms.

For suspended solid particles, their translational velocity $\mathbf{u}_p$ and rotational velocity $\boldsymbol{\omega}_p$ are governed by the Newtonian equations of motion, respectively, given by:

$$m_p \frac{d\mathbf{u}_p}{dt} = m_p \mathbf{g} + \mathbf{F}_{f\to s} \tag{4}$$

$$I_p \frac{d\boldsymbol{\omega}_p}{dt} = \mathbf{T}_{f\to s} \tag{5}$$

where m_p and I_p are the mass and the moment of inertia of the particle, respectively.

And the $f \to s$ terms represent the drag and torque exert upon the particle by the fluid. They are calculated from integrating viscous stress and pressure contribution components around the sphere surface:

$$\mathbf{F}_{f\to s} = \oint_S \mathbf{f}_{f\to s} dS = \oint_S (\mu \nabla \mathbf{u} \cdot \mathbf{n} - p\mathbf{n}) dS \text{ and}$$
$$\mathbf{T}_{f\to s} = \oint_S (\mathbf{r} - \mathbf{r}_p) \times \mathbf{f}_{f\to s} dS \tag{6}$$

where $\mathbf{n}$ is the outward unit normal vector, $\mathbf{r}$ are position vectors to points at particle surface, $\mathbf{r}_p$ is the position vector to the center of the sphere, and the integrating area S is the surface of the sphere.

In the present study, the particles' temperature hold constant, and equal to the wall temperature $T_p = T_w$. The particles are imaged as reaction particles. The wall temperature is higher than the free-stream temperature $T_w > T_\infty$ during the simulations.

We employ a ghost-cell based high-order immersed boundary method (IBM) to represent the existence of solid particles in the computational domain. Its core idea is approximating the Taylor series expansion on a body-intercept point by an Nth-order polynomial ($N = 3$ in the present study), which consists of the nearest ghost point whose flow information is to be determined, and a set of adjacent fluid points on which flow information are already known. In practice, the value on the ghost point is specified by minimizing the error between the approximate polynomial and the Taylor series expansion. This is done by constructing and solving a least square problem. More details about the mathematical derivation and the numerical implement are available in our previous papers Xia et al. (2014, 2015).

Moreover, the particle-particle and particle-wall contact forces are calculated by the soft-sphere collide model during the simulation, instead of the commonly used lubrication correction. The development of this model can trace back to Cundall and Strack (1979). Numerical implement of the model can be found in van der Hoef et al. (2006), and its combination with IBM method has been given by Breugem (2010).

3. Numerical implementation

In order to isolate the influences of buoyancy and finite-size particles, we simulate three cases, a single-phase neutral boundary