Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(2):272-282
DOI: 10.1016/S1001-6058(16)60737-0
www.sciencedirect.com/
science/journal/l0016058

Effects of finite-size heavy particles on the turbulent flows in a square duct*

Zhao-wu Lin (林昭武)[1], Xue-ming Shao (邵雪明)[1], Zhao-sheng Yu (余钊圣)[1], Lian-ping Wang (王连平)[2]
1. *State Key Laboratory of Fluid Power and Mechatronic Systems, Department of Mechanics, Zhejiang University, Hangzhou 310027, China, E-mail: lin3030612023@163.com*
2. *Department of Mechanical Engineering, University of Delaware, Newark, Delaware 19716, USA*

(Received August 12, 2015, Revised January 27, 2016)

Abstract: A parallel direct-forcing fictitious domain method is applied in fully-resolved numerical simulations of particle-laden turbulent flows in a square duct. The effects of finite-size heavy particles on the mean secondary flow, the mean streamwise velocity, the root-mean-square velocity fluctuation, and the particle concentration distribution are investigated at the friction Reynolds number of 150, the particle volume fraction of 2.36%, the particle diameter of 0.1 duct width, and the Shields number ranging from 1.0 to 0.2. Our results show that the particle sedimentation breaks the up-down symmetry of the mean secondary vortices, and results in a stronger secondary-flow circulation which transports the fluids downward in the bulk center region and upward along the side walls at a low Shields number. This circulation has a significant impact on the distribution of the mean streamwise velocity, whose maximum value occurs in the lower half duct, unlike in the plane channel case. The flow resistance is increased and the turbulence intensity is reduced, as the Shields number is decreased. The particles accumulate preferentially at the face center of the bottom wall, due to the effect of the mean secondary flow. It is observed that the collision model has an important effect on the results, but does not change the results qualitatively.

Key words: Turbulent duct flow, particle-laden flow, mean secondary flow, fictitious domain method

Introduction

The turbulent flow in a square duct is characterized by the presence of mean cross-stream fluid motions. This kind of secondary flows, induced by turbulence fluctuations, takes the form of eight symmetrical vortices, with two counter-rotating vortices in pairs in each quadrant of the duct (see Fig.2). The mean secondary flows transport the fluid momentum from the bulk region to the corner areas along each corner bisector, and then back to the bulk regions along the wall bisectors. The early experimental measurements of the turbulent flows in a square duct were focused on the Reynolds stresses as the source for the generation of mean secondary flows. Direct numerical simulations of the single-phase turbulent flow in a square duct were performed by Gavrilakis[1], Uhlmann et al.[2] and Pinelli et al.[3].

There are limited studies of the particle-laden turbulent flows in a square duct in the literature. Winkler et al.[4] investigated the preferential concentration of particles in a fully developed turbulent square duct flow, and observed that particles tended to accumulate in regions of high strain-rate and low swirling strength. Sharma and Phares[5] reported that the mean secondary flow enhanced the lateral mixing for passive tracers and low-inertia particles, and higher inertia particles accumulated close to the wall. Winkler et al.[6], Yao and Fairweather[7] and Yao et al.[8] investigated the particle deposition in turbulent square duct flows. The results of Winkler et al.[6] show that the deposition occurs with greater probability near the center of the duct walls than at the corners. On the other hand, Yao and Fairweather[7] and Yao et al.[8] concluded that high-inertia particles tend to deposit close to the corners of the duct floor, while low-inertia particles deposit near the floor center. Yao and Fairweather[9] investi-

---

* Project supported by the National Natural Science Foundation of China (Grant Nos. 11372275, 51376162), the Research Fund for the Doctoral Program of Higher Education of China (Grant No. 20130101110035).

Biography: Zhao-wu Lin (1984-), Male, Ph. D. Candidate
Corresponding author: Zhao-sheng Yu,
E-mail: yuzhaosheng@zju.edu.cn

gated the resuspension of inertial particles in a turbulent square duct flow and demonstrated the important role of the mean secondary flow in the resuspension process.

In these studies of two-phase flows, the point-particle approximation was employed to deal with the particle motion, which is valid in principle only when the particle size is smaller than the Kolmogorov length scale and the particle volume fraction is low. In recent years, the interface-resolved direct numerical simulation methods were used to study the mechanisms of the interactions between the turbulence and the finite-size particles, to determine the interface between a particle and the fluid and all turbulent structures with the direct numerical simulation method. Such methods were applied to simulations of particle-laden wall-bounded turbulent flows such as the pipe flow[10] and the horizontal channel flows[11-15]. We investigated the effects of the finite-size neutrally buoyant particles on the turbulent flows in a square duct with the interface-resolved DNS method, and observed that the particle addition increased the intensity of the mean secondary flow. In the present paper, we report our results on the particle effects on the turbulent duct flow when the particle sedimentation effect is present. It is noted that in the previous studies based on the point-particle approximation, the modulation of the turbulent duct flow by the particles has not been made clear.

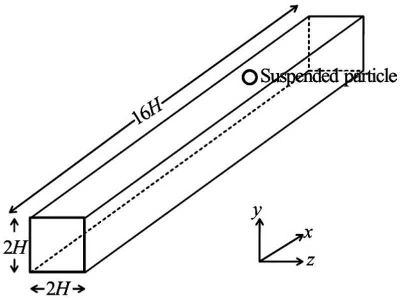

Fig.1 Geometrical model for the duct flow

1. Numerical model

1.1 *Flow model*

A schematic diagram of the geometrical model for the duct flow is shown in Fig.1. The x-axis is aligned in the streamwise direction. The z-axis direction is taken as the spanwise direction and y-axis direction as the transverse direction, as in the plane channel flow case. The corresponding velocity components in the (x, y, z) directions are $\boldsymbol{u} = (u, v, w)$, respectively. The no-slip velocity boundary condition is imposed at the duct walls and the periodic boundary condition is imposed in the streamwise direction. We denote the half width of the duct as H. In the present study, the computational domain is $[0, 16H] \times [-H, H] \times [-H, H]$.

We take H as the characteristic length and the friction velocity u_τ as the characteristic velocity in the non-dimensionalized scheme. The friction velocity is defined as $u_\tau = \sqrt{\tau_w / \rho_f}$, with τ_w being the mean shear stress on all walls, and ρ_f the fluid density. Thus, the Reynolds number is defined as $Re_\tau = u_\tau H / \nu$, with ν being the fluid kinematic viscosity. The pressure gradient is kept constant in our simulations, as $(-\mathrm{d}p / \mathrm{d}x)^e = 2\tau_w / H$, and its dimensionless value normalized by $\rho_f u_\tau^2 / H$ is 2.

1.2 *Direct-forcing fictitious domain method*

A parallel direct-forcing fictitious domain method (DF/FD) is employed for the simulation of particle-laden turbulent duct flows. The fictitious domain (FD) method for the particulate flows was originally proposed by Glowinski et al.[16]. The key idea of this method is that the interior of the particles is filled with the fluids and the inner fictitious fluids are enforced to satisfy the rigid body motion constraint through a pseudo body force, introduced as a distributed Lagrange multiplier in the FD formulation[16]. In what follows, we describe the DF/FD method briefly, and the details can be found in Yu and Shao[17].

For simplicity of description, we will consider only one spherical particle in the following exposition. The particle density, volume and moment of inertia, the translational velocity, the angular velocity and the position are denoted by ρ_s, V_p, J, $\boldsymbol{U}$, $\boldsymbol{\omega}_p$ and $\boldsymbol{X}_p$, respectively. Let $P(t)$ represent the solid domain and Ω the entire domain including the interior and the exterior of the solid body. By introducing the following scales for the non-dimensionalization: H for length, u_τ for velocity, H / u_τ for time, $\rho_f u_\tau^2$ for the pressure, and $\rho_f u_\tau^2 / H$ for the pseudo body force, the dimensionless FD formulation for the incompressible fluids and the spherical particles can be written as follows:

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = \frac{\nabla^2 \boldsymbol{u}}{Re_\tau} - \nabla p + 2 + \boldsymbol{\lambda} \quad \text{in} \quad \Omega \tag{1}$$

$$\boldsymbol{u} = \boldsymbol{U} + \boldsymbol{\omega}_s \times \boldsymbol{r} \quad \text{in} \quad P(t) \tag{2}$$

$$\nabla \cdot \boldsymbol{u} = 0 \quad \text{in} \quad \Omega \tag{3}$$

$$(\rho_r - 1) V_p^* \left(\frac{\mathrm{d}\boldsymbol{U}}{\mathrm{d}t} - Fr \frac{\boldsymbol{g}}{g} \right) = -\int_P \boldsymbol{\lambda} \mathrm{d}\boldsymbol{x} \tag{4}$$