Powder Technology 269 (2015) 320–336

Contents lists available at ScienceDirect

Powder Technology

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

Effect of collisions on the particle behavior in a turbulent square duct flow

Hao Zhang [a,b], F. Xavier Trias [a], Andrey Gorobets [a,c], Assensi Oliva [a,*], Dongmin Yang [d,*], Yuanqiang Tan [b], Yong Sheng [d]

[a] Heat and Mass Transfer Technological Center, Technical University of Catalonia, Terrassa, Barcelona 08222, Spain
[b] School of Mechanical Engineering, Xiangtan University, Hunan 411105, China
[c] Keldysh Institute of Applied Mathematics of RAS, 4A Miusskaya Sq., Moscow 125047, Russia
[d] School of Civil Engineering, University of Leeds, Leeds LS2 9JT, UK

ARTICLE INFO

Article history:
Received 25 April 2014
Received in revised form 7 July 2014
Accepted 30 August 2014
Available online 16 September 2014

Keywords:
Collisions
Particle behavior
Turbulent duct flow
DNS
DEM

ABSTRACT

The effect of collisions on the particle behavior in a fully developed turbulent flow in a straight square duct at $Re_\tau = 300$ is numerically investigated. The hydrodynamic modeling of the fluid phase is based on direct numerical simulation. The kinematics and trajectory of the particles as well as the collisions are described by the discrete element method. Three sizes of particles are considered with diameters equal to 50 μm, 100 μm and 500 μm. Firstly, the particle transportation by turbulent flow is studied in the absence of the gravitational effect. It is found that the collisions play an important role in the particle distribution especially in the near-wall regions. The inter-particle collisions enhance the particle diffusion in the direction perpendicular to streamwise flow and make the particles distribute more uniformly near the wall. Then, the particle deposition is studied under the effect of the wall-normal gravity force in which the influence of collisions on the particle resuspension rate and the final stage of particle distribution on the duct floor are discussed, respectively. The collisions are found to have influence on the particle resuspension rate near the duct floor whereas hardly affect the particle behavior near the duct center. Under the gravitational effect the 50 μm particles deposit more efficiently near the side walls but the 100 μm and 500 μm particles preferentially deposit near the center of the duct floor. Moreover, all the sizes of particles tend to concentrate near the center of the duct floor at the final stage of the particle deposition when the inter-particle collisions are considered.

1. Introduction

Particle-laden turbulent square duct flows are commonly encountered in both engineering and environmental applications such as the management of dust in clean rooms, chemical reactions involving a particulate catalyst, the flow of liquid and gas mixtures through process equipment and in the combustion of liquid sprays [1]. This configuration is of high interest because the behavior of the solid particles can be affected by the secondary vortexes that are known as the secondary flow of Prandtl's second kind.

Under the action of the gravitational force, the particles may deposit on the duct floor and form solid beds. The depositions are highly related to the blockages and erosion on the duct wall which may cause a significant reduction of its life-span. Moreover, these processes are always accompanied by the particle resuspension and complex inter-particle collisions that make the problem even more difficult to be solved. A well understanding of how these flows behave is of clear benefit to assist the reliable scale-up and re-design of processes of different types.

1.1. Single phase turbulent flows in straight square ducts

Since the pioneering experiment by Nikuradse [2], the single phase turbulent square duct flow has been the subject of several experimental studies [3,4]. The secondary flow of Prandtl's second kind consists of four pairs of counter-rotating vortexes normal to the streamwise direction, statistically these eight vortexes distribute symmetrically about the bisectors of the walls and the diagonals of the square cross-sections. The streamwise flow is flattened by the secondary motions with transferring the fluid momentum from the centerline to the duct corners. Besides the experimental studies, the adoption of numerical simulations can assist people in making decisions on trial conditions. Previous simulations of the turbulent square duct flow are broadly divided into three categories: Direct Numerical Simulation (DNS) [5–11], Large-Eddy Simulations (LES) [12,13] and the solution of the Reynolds-averaged Navier–Stokes equations with turbulent models (RANS) [14,15]. In this paper, we focus on the DNS part, a brief summary

* Corresponding authors.
E-mail addresses: cttc@cttc.upc.edu (A. Oliva), d.yang@leeds.ac.uk (D. Yang).

http://dx.doi.org/10.1016/j.powtec.2014.08.070
0032-5910/

of this work is presented here. DNS aims at solving Navier–Stokes equations with the whole range of spatial and temporal scales of the turbulent relevant. Gavrilakis [5] performed a DNS of the turbulent square duct flow at $Re_\tau = 300$ providing a detailed description of the mean flow in the transverse plane and turbulent statistics along the wall bisectors. Huser and Biringen [6] expanded the database by simulating turbulent square duct flow at $Re_\tau = 600$ in which the corner influences on turbulent statistics and on the origin of the secondary flows were explained. Then, Huser et al. [7] assessed all the terms in the Reynolds stress transport equation. The DNS results of Uhlmann et al. [8] indicated that with the rise of the Reynolds number there is a marginal state before fully turbulent one. The marginal Reynolds number is around $Re_\tau = 160$ below which the flow exhibits totally different secondary flow structure alternating in time. Joung et al. [9] and Pinelli et al. [11] performed a series of DNS at $Re_\tau = 300$ to pursuit the basic physical mechanisms that are responsible for Prandtl's secondary motion of the second kind. From the survey of the literature, the DNS has been proven to be a robust method to investigate the turbulent duct flow at a relatively low Reynolds number.

1.2. Particle-laden turbulent flow in a square duct

It is a challenging work to track a large number of particle trajectories in a turbulent flow and obtain dynamic information of them by physical experimentation at this stage of development. Experimental investigations of the particle-laden turbulent flow in square ducts are very limited and mainly focused on the airborne particles such as that on the particle resuspension rate in ventilation ducts [16], the deposition in horizontal ducts [17] and dispersion in a vertical ventilation duct [18].

Zhang and Ahamad [19] carried out a DNS coupled with a Lagrangian Particle Tracking (DNS–LPT) method to study the aerosol particle transportation and deposition in vertical and horizontal turbulent duct flows, they found that the wall coherent structure plays an important role in the particle deposition process. Winkler and Rani [20] conducted a LES–LPT simulation ($Re_\tau = 360$) to investigate the preferential concentration of particles in a downward fully developed turbulent square duct flow. They concluded that the particles tend to accumulate in the regions of low vorticity magnitude. However, near the wall, the particles show a tendency to accumulate in regions of high vorticity with this phenomenon increasing with the response time. Winkler et al. [21] also indicated that the lift forces due to secondary flow velocity gradients are not as important as those due to streamwise velocity gradients in a square duct flow.

The particle deposition in turbulent duct flow is of particular interest under the effect of the gravitational force. Yao and Fairweather [22] performed a LES–LPT simulation to investigate the particle resuspension mechanism in relatively high $Re_\tau = 10550$ turbulent flows where the role of drag force and lift force on the particle resuspension were studied, the carried fluid was assumed to be water. Four kinds of particles with different sizes were considered in the work [22] and they found that the drag force dominates small (5 μm) particle resuspension while for the large (500 μm) particles the lift force is also a contributing factor especially close to the duct wall. Then, this work was compared by a RANS–LPT simulation [23] indicating that the main difference between LES– and RANS–LPT simulation is in the magnitude of the resuspension rate with RANS predicting a greater variability across the duct.

Sharma and Phares [24,25] carried out DNS–LPT simulations ($Re_\tau = 300$) to study the effect of the secondary flow on the transport and deposition of different inertia particles with the gravity effect ignored. They found that the higher-inertia particles accumulate close to the sidewalls during transportation and concentrate close to the corners in deposition.

Fairweather and Yao [1] investigated the mechanisms of particle dispersion in a turbulent square duct flow via LES–LPT and showed that the secondary flows within the duct dominate small particle dispersion, whereas gravity promotes the deposition of large particles on the duct floor. Yao and Fairweather [26] used the same code to further investigate the particle deposition mechanism in turbulent duct flows at various Re_τ. They mainly concluded that high inertia particles tend to deposit near the side wall in high $Re_\tau = 10500$ turbulent duct flows whereas the low inertia ones tend to deposit close to the centers of the duct floor. All the sizes of particles deposit in the center region in relative low $Re_\tau = 600$ turbulent duct flows. The particle deposition process in [26] can be described by the free-flight model.

1.3. Motivation and summary of the present work

The afore-mentioned simulations have mainly focused on the hydrodynamic interaction between the turbulent flow and the solid particles whereas the inter-particle collisions were neglected. The research of Tanaka and Tsuji [27] has shown that the effect of inter-particle collisions cannot be ignored even in dilute flow. They performed a numerical simulation to investigate gas–particle flow in a vertical pipe and found that the effect on the diffusion in the direction normal to the mean flow is especially important. Winkler et al. [28] studied the effect of the inter-particle collisions on the particle deposition in vertical duct turbulent flow, they noted that the collision effects can decrease the deposition velocity due to the fact the direction of the gravity is along the streamwise direction. Sommerfeld and Kussin [29,30] made a detailed analysis of collision effects for turbulent gas–particle flow in a horizontal channel where the particle interaction was based on the stochastic collision model and thus the location and motion of neighboring particles were not required. They found that the wall roughness has an important effect on the horizontal particle mean velocity, the fluctuating components and the wall collision frequency. The inter-particle collisions play an important role in the redistribution of the particle phase fluctuating motion, namely a decrease of the streamwise component and an enhancement of the lateral component. To the best knowledge of the authors, no research has been reported on the effect of the inter-particle collisions on the particle behavior in horizontal duct turbulent flows or on the particle resuspension rate. However, similar studies by means of other geometries have been conducted such as the research of Yan et al. [31] on particle dispersion in a turbulent jet, Afkhami et al. [32] and Yamamoto et al. [33] in a channel.

In this study, we performed DNS coupled with the Discrete Element Method (DEM) [34] to study the effect of inter-particle collisions on the behavior of particle-laden turbulent flow in a square duct. The friction Reynolds number adopted is $Re_\tau = 300$ which is defined as $Re_\tau = hu_\tau/\nu$ based on the duct hydraulic diameter h, the mean friction velocity u_τ and the kinematic viscosity ν. The DEM, as a typical Lagrangian method, has been widely accepted as an effective method of addressing engineering problems containing granular and discontinuous materials [35–37]. The method cannot only track the trajectories of the particles but also provide the transient forces acting on individual particles [38] that are very difficult to observe on-line by the physical experiments. The considered forces, both direct-contact and non-contact forces, can be directly exerted on the particles numerically which enables multiphysics [39,40] and multiphase [41,42] coupling simulations based on DEM. In this study, the interaction between the solid particle and particle/wall is based on the theoretical contact mechanics thereby it is possible to directly use the material properties of the particle in the calculation. A small overlap between the rigid particles is allowed to represent the physical deformation that takes place between the contacting elements. The DEM under this assumption is called soft-sphere DEM.

The rest of the paper is organized as follow: Section 2 describes the governing equations and the numerical procedures for DNS and DEM. In Section 3, the predicted single phase flow fields are compared with the former DNS results reported by Gavrilakis [5]. Then in Section 4, the numerical simulations of the multi-phase flow are conducted and