Fuel 234 (2018) 189–198

Contents lists available at ScienceDirect

Fuel

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

Full Length Article

Numerical study of particle deposition in turbulent duct flow with a forward- or backward-facing step

Hao Lu[a,b], Wenjun Zhao[c,*]

[a] Key Laboratory of Enhanced Heat Transfer and Energy Conservation of Education Ministry, School of Chemistry and Chemical Engineering, South China University of Technology, Guangzhou 510640, China
[b] Faculty of Engineering, The University of Nottingham, Nottingham, United Kingdom
[c] Faculty of Architecture, The University of Hong Kong, Hong Kong, China

ARTICLE INFO

Keywords:
Particle deposition
Forward-facing step
Backward-facing step
Numerical simulation

ABSTRACT

Particle deposition in duct air flow over a forward- or backward-facing step was investigated by Reynolds stress model with velocity fluctuation correction combined with discrete particle model. The mean air velocities in duct with a backward-facing step and particle deposition velocities for smooth duct both agreed well with related literature results. The deposition velocities of the duct with a forward-facing step are significantly higher than those of smooth duct, especially for small particles ($d_p < 10\,\mu$m). Particle deposition velocities in the duct with a backward-facing step are higher for small particles ($d_p < 10\,\mu$m) but lower for large particles ($d_p > 10\,\mu$m), compared with smooth duct case. Air velocity and turbulent kinetic energy fields in the duct with a forward- or backward-facing step are quite different from smooth duct case, which modify the particle deposition characteristics significantly. Furthermore, a particle deposition efficiency ratio was defined and analyzed by considering flow drag of the duct. The results showed that the peak deposition efficiency ratio of the duct with a forward-facing step can reach 60 for particles of 1 μm. The duct with a forward-facing step can enhance deposition of small particles more significantly ($d_p < 10\,\mu$m), compared with the duct with a backward-facing step.

1. Introduction

Particulate matter (PM) deposition in turbulent duct flow is usually encountered in energy and environmental engineering. Forward-facing step (FFS) and backward-facing step (BFS) are commonly used in the above gas-particle two-phase devices, such as expansion or contraction of building ventilation duct, the external casing of air cleaner, pulverized coal combustor and gas-particle reactor [1–3]. Moreover, FFS and BFS are basic and important configurations to investigate particle motion behaviors in geometry-induced flow separation. Better understanding of particle deposition characteristics and mechanisms in FFS or BFS flows is significant for efficiency improvement of above related equipment [4–7]. Although a large amount of studies were conducted on particle deposition in smooth duct flow, very limited attention has been paid for deposition behaviors of particles in FFS or BFS flow. Therefore, particle deposition in FFS and BFS duct needs to be investigated in details for better design of many related devices.

In the last several decades, particle deposition in smooth duct flow has been studied by theoretical analysis, experimental measurement and numerical simulation [8–12]. It has been concluded that deposition velocity profile of particles would firstly fall, then significantly rise and finally keep constant as particle relaxation time increases [13–15]. Generally, particle deposition in vertical smooth duct could be separated into three regimes [16]. Particle deposition is determined by turbulent vortex and Brownian diffusion in turbulent particle diffusion regime. With increase of particle diameter, Brownian diffusion effect reduces while particle inertia becomes more important for particle deposition in eddy diffusion-impaction regime. Particle deposition behaviors are mainly dominated by particle inertia due to high mass in inertia-moderated regime. Moreover, Lai et al. [17] developed an empirical model to estimate particle deposition velocity in smooth duct flow. Turbulent particle diffusion, Brownian diffusion and gravity effect were considered in the model. Zhao and Wu [18–19] improved the three-layer model by considering turbophoresis to model particle deposition behaviors in smooth and rough duct flows.

Recently, CFD simulation becomes a main way to predict particle deposition behaviors in duct flow [20–22]. The Reynolds-averaged Navier-Stokes (RANS) model and discrete particle model (DPM) are

* Corresponding author.
E-mail address: zhaowenjunhku@gmail.com (W. Zhao).

https://doi.org/10.1016/j.fuel.2018.07.033
Received 11 March 2018; Received in revised form 14 June 2018; Accepted 7 July 2018
0016-2361/

usually employed to simulate particle deposition in turbulent duct flow [20–22]. Because this Eulerian-Lagrangian method can effectively reflect particle motion behaviors in turbulent flow fields. Tian [23] compared particle deposition velocity in smooth duct by using various RANS models. The results showed Reynolds stress model (RSM) has better accuracy on particle deposition simulation than other RANS models. Zhang et al. [24] found that near-wall treatment is crucial for accurately model particle deposition in smooth duct flow.

Although particle deposition in smooth duct was well investigated, the researches for non-smooth duct case are very limited. Sippola et al. [25–36] experimentally investigated particle deposition rates in S-shape connector and bent duct. Particle deposition velocity in non-smooth duct is obviously higher compared with smooth duct. Lustfeld et al. [27] studied multilayer deposition of particles in FFS duct by experimental measurement. They found that gravitational settling and inertial impaction are the main mechanisms for particle deposition. Benedetto et al. [28] investigated dust dispersion in turbulent flow fields within a 20 L bomb. The results showed that turbulent vortices produce dead volumes for the dust and the dust is pushed toward the walls of the sphere. Moreover, Sarli et al. [29,30] further investigated effects of different nominal particle concentrations and particle diameter on dust dispersion inside a 20 L bomb. They found that sedimentation prevails when the dust nominal concentration is increased and dust is mainly concentrated at the vessel walls with increase of dust diameter. The authors [31–34] previously investigated particle deposition in expanding, contracting and ribbed ducts. We found that particle deposition characteristics in non-smooth duct are greatly different from smooth duct. In consequence, particle deposition in FFS or BFS ducts has been seldom studied and the detailed deposition mechanisms remain unclear. Thus the objective of the study is to examine particle deposition behaviors and mechanisms in FFS and BFS duct flows. The RSM model with user-defined function (UDF) and DPM model were used to simulate turbulent flow and particle movement respectively. Turbulent flow fields, particle deposition velocities and mechanisms in FFS and BFS ducts were studied in details. Moreover, particle deposition efficiency considering flow drag for FFS and BFS ducts was also obtained and compared with smooth duct case, as FFS or BFS may be a potential way to enhance particle deposition in related devices.

2. Numerical methods and solution strategies

The commercial CFD software ANSYS FLUENT 15.0 was used to solve the Reynolds stress model (RSM) turbulence model and discrete particle model (DPM) in the simulation. Moreover, correction of turbulent velocity fluctuation as well as fully developed turbulent inlet velocity and turbulent kinetic energy profiles were imposed in the simulation to improve prediction accuracy by user-defined function codes.

2.1. Reynolds stress model with UDF correction

The Reynolds-averaged Navier-Stokes equation for turbulent duct flow can be written as follows,

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(\mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \overline{u_i' u_j'}\right), \tag{2}$$

In the above equations, $\bar{u}_i$ and $\bar{p}$ are velocity and pressure respectively. $\rho \overline{u_i' u_j'}$ is Reynolds stress tensor. Reynolds stress model (RSM) was used in the simulation, as it has been proven to have the best accuracy for predicting particle deposition in RANS models. The transport equation of Reynolds stress can be written by,

$$\frac{\partial}{\partial t}(\overline{u_i' u_j'}) + \bar{u}_k \frac{\partial}{\partial x_k}(\overline{u_i' u_j'}) = \frac{\partial}{\partial x_k}\left(\frac{\nu_t}{\sigma_k}\frac{\partial \overline{u_i' u_j'}}{\partial x_k}\right) - \left(\overline{u_i' u_k'}\frac{\partial \bar{u}_j}{\partial x_k} + \overline{u_j' u_k'}\frac{\partial \bar{u}_i}{\partial x_k}\right) - C_1 \frac{\varepsilon}{k}\left[\overline{u_i' u_j'} - \frac{2}{3}\delta_{ij} k\right] - C_2 [P_{ij} - \frac{2}{3}\delta_{ij} P] - \frac{2}{3}\delta_{ij}\varepsilon \tag{3}$$

Particle turbulent dispersion caused by turbulent instantaneous fluctuation is an important mechanism to affect particle deposition velocity. Discrete Random Walk Model (DRW) was commonly used to simulate particle turbulent dispersion. The DRW model allows successive encounter of particles with turbulent eddies by a Gaussian distributed random velocity fluctuation of fluids and a time scale of turbulent eddy [23]. The turbulent fluctuating velocity is given as,

$$u' = \zeta u'_{rms},\ v' = \zeta v'_{rms},\ w' = \zeta w'_{rms} \tag{4}$$

where ζ is normal distributed random number with zero mean and unit variance; and u'_{rms}, v'_{rms} and w'_{rms} are fluctuating velocities obtained by the RSM model respectively. However, as a time-averaged turbulence model, the RSM model cannot accurately predict particle turbulent dispersion. Thus the correction of turbulent velocity fluctuation is necessary to improve simulation accuracy. Tian et al. [23] numerical predicted particle deposition velocity in smooth duct flow. They found that wall-normal turbulent velocity fluctuation in the near-wall region is crucial for accurately predicting particle deposition velocity. Therefore, the DNS results of wall-normal velocity fluctuation by Kim et al. [35] was employed in particle deposition simulation for smooth duct, as follows,

$$\frac{v'_{rms}}{u^*} = C(y^+)^2,\ \text{for } y^+ < 4 \tag{5}$$

For complex wall turbulent flow, Eq. (6) was successfully used by Lecrivain et al. [36] to improve particle deposition simulation in duct flow with ribs. This equation was also adopted in the present study for FFS and BFS duct cases. The Eq. (6) can be written by,

$$\frac{v'_{rms}}{u^*} = \frac{a_1 y^{+2}}{1 + b_1 y^+ + c_1 y^{+2.41}},\ \text{for } y^+ < 30 \tag{6}$$

In the above equations, $C = 0.008$, $a_1 = 0.0116$, $b_1 = 0.203$ and $c_1 = 0.0014$ y^+ is described by,

$$y^+ = \frac{y\, u^*}{\nu} \tag{7}$$

Turbulent velocity field and TKE distribution have great influences on particle deposition behaviors. Particle deposition characteristics will be quite different in developing and fully developed turbulent duct flow. As particle deposition in fully developed turbulent flows have a lot of engineering applications such as building ventilation system and pneumatic conveying, this study focused on particle deposition in fully developed turbulent flow over a BFS or FFS. Moreover, most of previous studies on particle deposition in turbulent duct flow were in the condition of fully developed turbulent flow status. Thus a large number of experimental and numerical data can be used to validate present numerical methods and results.

Fully developed velocity distributions were obtained and applied at inlet, which can be addressed by [23],

$$U = U_{free}\left(\frac{y}{D/2}\right)^{1/7} \text{for } y \leqslant D/2 \tag{8}$$

$$U = U_{free}\left(\frac{h-y}{D/2}\right)^{1/7} \text{ for } y > D/2 \tag{9}$$

$$U_{free} = \frac{8}{7}U_{mean} \tag{10}$$

where D is duct inlet height. U_{mean} is air mean velocity. Pressure outlet and non-slip conditions were employed at outlet and the duct walls. Moreover, fully developed TKE distributions were used at inlet and can