International Journal of Heat and Mass Transfer 83 (2015) 372–381

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Effects of preferential concentration on collision and erosion between solid particles and tube bank in a duct flow

Kun Luo, Fan Wu, Kunzan Qiu, Zeli Wang, Jianren Fan *

State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China

ARTICLE INFO

Article history:
Received 23 February 2013
Received in revised form 25 September 2013
Accepted 12 October 2014

Keywords:
Solid particle erosion
Impact
Power generation
Wear modeling

ABSTRACT

The impact and erosion on the stainless steel heat exchanger tubes located in the middle row of a 10 × 10 aligned tube bank caused by coal ash particles are investigated numerically by means of direct numerical simulations (DNS) to study the influence of preferential concentration on particle–tube interactions. The immersed boundary technique is applied to account for the coupling effect on the flow by the immersed tubes. A two-way coupled Lagrangian solver is developed for particle tracking. The experimentally validated particle–wall impact and erosion models are used to describe the collision between particles and tubes and the resulting erosion. The results demonstrate that the particles Stokes number has significant effects on particle dispersion patterns as well as collision and erosion characteristics. The global tube erosion of the first tube increases with the increment of the particle size, but the particles with an intermediate Stokes number of 1.6 cause the most erosion to the other downstream tubes with lower collision frequency, due to the preferential concentration. The maximum local tube erosion happens in the front side surface of each tube with certain angle region for all particles where more attention should be paid to prevent from erosion caused by coal ash particles.

1. Introduction

Turbulent flows laden with particles occur very often in nature and industrial applications, and have attracted much attention since 1960s. So far, one of the most striking features identified in turbulent flows laden with inertial particles has been the very strong inhomogeneity of the particle concentration field, namely the preferential concentration effect. There are numerous experimental, theoretical, and numerical based studies dedicated to this topic, and the mechanism of preferential concentration has been established. For details, please refer to the recent comprehensive reviews [1–2].

Although the phenomenon and mechanism of preferential concentration have been well-known, there is little study on the effect of preferential concentration on related applications. One of such examples is the study of effect of preferential concentration on particle collision rate and coalescence, which is considered as a possible mechanism to understand rain initiation in warm clouds [3]. However, there are more situations where preferential concentration may play important roles in industrial applications, and need to be investigated. For example, particle-laden flows around bluff bodies are commonly encountered in industrial applications, such as the coal–air flows around a bluff-body burner, the particulate flows around immersed tubes in chemical reactors, and the burnt gas containing dispersed particles flows around tube bank of a heat exchanger in a coal-fired power plant. In these flows, the dispersed particles may collide with the bodies and cause a serious problem of material erosion that will certainly shorten the life of the equipment. It is thus interesting to study the role of preferential concentration in controlling particle–wall collision and resultant erosion in such particle-laden flows.

Many experimental studies have been carried out to investigate the mechanism of erosion by the impacting particles and develop anti-erosion methods [4–6]. Hutchings [7] reviewed relevant studies and pointed out that erosion occurs at certain flow regime due to particle impact, and the erosion rate depends on the flow velocity, the impact angle of particle striking the surface of material, the particle properties, and the materials properties of the eroded surface.

In recent years, investigating erosion caused by solid particle impingement and developing anti-erosion approach by means of numerical simulations has also been conducted [8–9] with the

* Corresponding author. Tel./fax: +86 571 87951764.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.10.027
0017-9310/

development of computational fluid dynamics (CFD). The RANS approach, especially the k–ε turbulence model, had been widely used in the previous numerical studies of solid particle erosion [8–12]. The LES offers higher accuracy than the RANS by resolving the large-scale energy-containing eddies [13]. The DNS solves the Navier–Stokes equations to all scales without any closure model, and is the most accurate method compared to RANS and LES. Although it has been widely used in single-phase or two-phase flows, the application to predict erosion by particle impact is very scarce.

In addition, while simple geometries discretized by uniform grids are efficiently solved, complex geometries in most industrial applications become research focuses. To deal with the coupling between complex geometries of practical applications and the underlying flow field, body-fitted curvilinear coordinate grids were traditionally used. Geometrical complexity combined especially with moving boundaries and high Reynolds numbers considerably increase the computational difficulties since they require, respectively, regeneration or deformation of the grid and turbulence modeling [14].

As an alternative, structured-grid based method, for instance, the immersed boundary method proposed by Peskin [15] is being more and more popular because of its flexibility in handling highly complex geometry and less computational demands due to the avoidance of remeshing [16–18]: In Peskin's case, a uniform, staggered Cartesian grid is used for the fluid phase, to which referred as the Eulerian grid; furthermore, irregular Lagrangian grid that is attached to and moves with the surface of the objects is used. The singular force on the Lagrangian coordinates at the immersed boundary was exerted on the flow field via Dirac delta function. To calculate the mutual interactions between solid boundary and fluid, some forcing schemes are proposed. Goldstein et al. [19] proposed a feedback forcing scheme to iteratively determine the magnitude of the force to satisfy the no-slip boundary condition on the immersed boundary. Saiki et al. [20] applied this feedback forcing scheme to compute the flow past a stationary and oscillating circular cylinder successfully. Fadlun et al. [14] proposed a direct forcing scheme to calculate the interaction force between immersed boundary and fluid. Uhlmann [21] incorporates the regularized delta function into a direct formulation of the fluid–solid interaction force to allow for a smooth transfer between the Eulerian girds and the Lagrangian points. In this paper, we use the immersed boundary method with multi-direct forcing scheme proposed by Wang et al. [17] to iteratively reinforce the satisfaction of the no-slip boundary condition on the immersed surface.

To describe particle–wall surface collision, three types of models are available in the literature. The first one takes the normal and tangential restitution coefficients of collision as constants. The second one takes the normal and tangential restitution coefficients of collision as a function of particle incident angle, of which the parameters are usually determined from experiments [22]. The third one is based on the particle impulse and momentum equations with some parameters determined from experiments [23]. For erosion by particle impact, some models were also proposed to predict the erosion rate [24–25].

Based on these backgrounds, direct numerical simulations with immersed boundary method are applied to study the collision and erosion between coal ash particles and the immersed stainless steel tubes consisting of a 10 × 10 aligned tube bank in a duct flow, as a first step to investigate realistic heat exchanger erosion in coal-fired burners. The emphasis is placed on the effect of preferential concentration on particle–tube collision and erosion. The impact and erosion characteristics on the tubes located in the middle of the duct by coal ash particles with different sizes are investigated and analyzed, which are believed to be helpful for erosion reduction.

2. Mathematical description

2.1. Flow configuration

Tube erosion is a severe problem in coal-firing power stations and chemical reactors. As a first step to provide insight into it, we investigate the impact and erosion on the stainless steel heat exchanger tubes located in the middle row of a 10 × 10 aligned tube bank in a cold flow caused by coal ash particles in the present study. The schematic view of the computational domain is shown in Fig. 1. The center of the first tube marked 1 is located at (4.5D, 9.5D). The flow has a Reynolds number of $\mathrm{Re}_D = 2 \times 10^3$ when the tube diameter is taken as the characteristic length. Four kinds of particles with different diameter are tracked, and the corresponding Stokes number defined as $St = \frac{\rho_p d_p^2/(18\mu)}{L/U}$ are 0.01, 0.1, 1.6 and 10, respectively. The volume fraction of the particles is set as 10^{-4} for each case. Other computational parameters are listed in Table 1. The impact and erosion characteristics on the tubes located in the middle of the duct as marked in Fig. 1 by different particles will be investigated and analyzed.

2.2. Governing equations for gas-phase flow

The dimensionless governing equations for the incompressible viscous flow in the entire computational domain are:

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

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u} \cdot \nabla \boldsymbol{u} = -\nabla P + \frac{1}{\mathrm{Re}} \nabla^2 \boldsymbol{u} + \boldsymbol{f}_d + \boldsymbol{f} \tag{2}$$

where $\boldsymbol{u}$ is the fluid velocity, P is the pressure and Re is the Reynolds number defined as $\mathrm{Re} = \frac{\rho_f \cdot U \cdot L}{\mu}$. Here ρ_f is the density of fluid, Reis the characteristic velocity of the flow field, Reis the characteristic length of the flow field ($L = D$) and μ is the fluid viscosity. f_d is the momentum coupling term between the fluid and the dispersed coal ash particles, expressed as:

$$\boldsymbol{f}_d = \sum_k^N \boldsymbol{f}_{dk} = \sum_k^N \frac{Z_m \boldsymbol{f}_p}{St} (\boldsymbol{V} - \boldsymbol{U}) \tag{3}$$

where N is the total number of the particle in the control volume, St is the stokes number, Z_m is the ratio of the single particle mass to the fluid mass in each grid, $\boldsymbol{f}_p$ is defined in Section 2.3, V and U are the instantaneous particle velocity and the fluid velocity at the particle position that is obtained by interpolation, respectively.

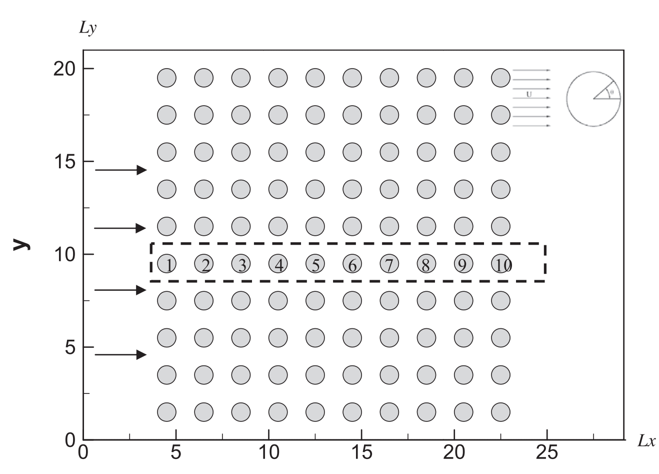

Fig. 1. Sketch map of the flow configuration and computational domain.