Powder Technology 281 (2015) 34–48

Contents lists available at ScienceDirect

Powder Technology

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

Evaluation of multifluid model for heat transfer behavior of binary gas–solid flow in a downer reactor

Zhan Shu [a,b], Junwu Wang [a,*], Quan Zhou [a,b], Chuigang Fan [a], Songgeng Li [a]

[a] State Key Laboratory of Multiphase Complex Systems, Institute of Process Engineering, Chinese Academy of Sciences, Beijing 100190, PR China
[b] University of Chinese Academy of Sciences, Beijing 100049, PR China

ARTICLE INFO

Article history:
Received 26 November 2014
Received in revised form 30 January 2015
Accepted 24 April 2015
Available online 4 May 2015

Keywords:
Fluidization
Gas–solid flow
Downer reactor
Multifluid model
Multiphase flow
Heat transfer

ABSTRACT

The heat transfer characteristics of different particles during coal pyrolysis process are of fundamental importance in a downer reactor, where hot sand particles often serve as the heat carrier for heating cold coal particles. Despite its urgent practical demands, fundamental studies on this topic were still very limited so far. To this end, this work carried out computational fluid dynamics (CFD) investigations of heat transfer behavior of binary gas–solid flow in a downer reactor using multifluid model. A modified gas–solid drag model and a modified Gunn's gas–solid heat transfer coefficient model were used to address the critical role of particle cluster structure. Furthermore, the effects of constant or temperature-dependent air properties, particle–particle drag force, and particle–particle heat transfer as well as the different choices of kinetic theories of granular flow were systematically evaluated, and then the optimized models were identified. CFD simulations with the optimized models show that CFD simulation has the ability to qualitatively capture the key heat transfer features in downers, based on the fact that a fairly good agreement with the available experimental data in the literature can be obtained and be further improved by taking the specific shape of inlet distributor into account.

1. Introduction

With deeper understanding of gas–solid flow behavior in fluidized beds in recent years, downer reactor has been increasingly applied in energy industry, primarily due to its prominent advantages of more uniform gas–solid flow, lower solid back mixing and shorter residence time, when compared with riser flows [1–4]. These important characteristics offer the possibility of devolatilizing coal particles under a rapid and uniform heating condition, which makes the downer to be a very attractive reactor for coal pyrolysis [5–7].

In coal pyrolyser, inert particles (such as sand) and reactant particles (coal powder) coexist and therefore form a binary mixture system. The inert particles are usually used as solid heat carriers to transport heat to coal particles during pyrolysis reaction process, and meanwhile as coke carriers to carry generated coke away from reactor wall [5,8,9]. Thus, the heat transfer characteristics between hot and cold particles are of great importance in pyrolysis reaction. However, a great amount of attention in the past has been mainly paid to the hydrodynamic behaviors of binary mixture in downer reactors [10], segregation/mixing behavior [11–13], and other accompanying aspects [14]. The studies focusing on the heat transfer in downer reactors with binary mixture have seldomly been reported. To the best of our knowledge, only Fushimi et al. [11] have investigated the solid–solid mixing behavior between the injected hot particles and circulating cold particles in a downer through a mixing index based on the temperature distributions. Although the primary purpose in their study was to measure the mixing quality of particles by experimentally measured temperature, the detailed temperature distributions at different heights were also reported.

The existence of particle cluster structures in downer reactors has been widely recognized [6,15,16], in spite of its more common occurrences in riser flows [17–22]. Most experimental studies have shown that particle clustering plays a key role in determining not only the hydrodynamics but also heat and mass transfer characteristics of downer reactors [6,15,16]. Therefore, it is necessary to consider the effects of particle cluster structure in computational fluid dynamics (CFD) studies of downer reactors, which undoubtedly have become the powerful research tools in chemical engineering. In fact, CFD has been extensively employed to study the hydrodynamics of downer reactors. For example, the hydrodynamic behaviors of particles and particle axial and radial solid holdup distribution in downers have been studied using a CFD–DEM coupled approach [23,24]; particle clustering phenomena and particle residence time distribution in both riser and downer have also been examined by Eulerian–Lagrangian CFD models [25] and Eulerian–Eulerian CFD models [26]; and flow properties of both monodisperse system and binary mixture system have been well-predicted by Eulerian–Eulerian CFD models for gas–solid flow in downer reactors [10,27,28]. More recently, our previous works have used CFD methods

---

* Corresponding author. Tel.: +86 10 82544842; fax: +86 10 62558065.
E-mail address: jwwang@ipe.ac.cn (J. Wang).

http://dx.doi.org/10.1016/j.powtec.2015.04.055
0032-5910/

as a promising approach to investigate the hydrodynamic behavior and segregation/mixing behavior of binary mixture system in downer reactors [12,29,30].

In this work, a multifluid model was chosen to investigate the heat transfer characteristics between cold and hot particles in a coal pyrolyser. The main purpose is to evaluate the effects of various available model inputs, including temperature-dependent air properties, gas–solid and particle–particle drag coefficient models, gas–solid and particle–particle heat transfer coefficient models and kinetic theories, and then find the optimal CFD models. The established model was further validated quantitatively against the experimental data from Fushimi et al. [11], such as temperature distributions.

2. Mathematical model and simulation layout

A multifluid model, as summarized in Table 1 was used to investigate the hydrodynamic and heat transfer behavior of binary gas–solid flow in a downer in this work. The primary phase was air, and the secondary phases were sands with different inlet temperatures (283 K for cold sand and 323 K for hot sand). It should be noted that hot sand is treated as a substitution for coal. The properties of both particles and gas were tabulated in Table 2. In the heat flux term of energy conservation equations, the real thermal conductivities of gas and particles were used instead of effective conductivities as have been used in the study of Patil et al. [31]. The gas properties can either be constants or vary as a function of bed temperature by the correlation of Flamant [32] (see Section 3.1). From Table 1 it can be seen that the extended Gidaspow's drag model [33] as well as two cluster-based drag models [29,34] were used for calculating the drag coefficient (see Section 3.2). The Gunn's heat transfer model [35] which has been widely used in previous simulations [11,16], and a modified Gunn's model considering the effect of particle clustering were used to investigate the effect of gas–solid heat transfer (see Section 3.3). The effect of particle–particle heat transfer coefficient was investigated by the model of Chang et al. [36] (see Section 3.4). A KTGF developed for binary gas–solid system with its corresponding particle–particle drag coefficient [37] as well as the default KTGF available in commercial software FLUENT were tested (see Section 3.5), and the effect of particle–particle drag force was studied by the theory of Chao et al. [37] and the model of Syamlal [38] (see Section 3.6).

The simulated downer geometry was the same as that used in the experiment [11] as shown in Fig. 1. Cold sand (a substitution for coal) was fed into the system from the top of downer uniformly, while the air and hot sand were fed into downer through a horizontal nozzle in normal or tangential arrangement. The exit of downer was set as an atmosphere pressure outlet. The hexahedral mesh was finer near the wall than those in the center. Tetrahedral mesh was generated and converted into polyhedral mesh in 3D geometry of the nozzle and solid inlet regions. The detailed size information of downer and input parameters was exhibited in Table 3. No-slip boundary condition was set for gas phase and the Johnson and Jackson's model [39] was used as the solid-wall boundary condition, following the choices of our earlier simulation [12]. More detailed parameters used in FLUENT (ANSYS, Inc., Canonsburg, PA) were also listed in Table 3. In order to improve accuracy and reduce numerical diffusion, the discretization schemes were the same as in our previous studies [12, 30]. The second-order upwind scheme was used for solving momentum equations, energy equations and granular temperature equations. The volume fraction equations were discretized using the QUICK scheme. The maximum packing limit of each solid phase was 0.54, which was calculated from the ratio of particle bulk density to particle density. The air velocity in the nozzle was set to 20 m/s, calculated from the gas velocity of 1.25 m/s in downer with the inner diameter of 0.1 m. The velocity of hot particles was 19.85 m/s calculated from the Hinkle modified IGT model [40] for horizontal pneumatic transport of particles. The calculating method for velocity of cold particles followed our previous work [12] and the velocity was 4.9 m/s. The inlet solid concentration (ε_s) was calculated by $\varepsilon_s = G_s/(\rho_p u_p)$. Furthermore, the exact value of input parameters (restitution coefficients (e, e_w) and specularity coefficient (ϕ)) were hard to be determined [41–43], the present choice of those parameters followed our previous works [29,30]. Note that in principle any continuum model for gas–solid flow should have turbulent or turbulent-like model regardless of Reynolds number, because there always has velocity difference around particles, when they are averaged to obtain continuum model, a term similar to Reynolds stress always exists [44]. However, in our model, we have neglected the effect of this term as in most of the previous continuum modeling of gas–solid flow [12,29,45].

A standard case was set for the evaluation of different model inputs and settings, each aspect of multifluid model studied in Part 3 was based on the standard case and only the corresponding parameter was changed. In the standard case, the geometry was without inlet distributor; air properties were the default constant parameters in FLUENT; the Peng et al.'s gas–solid drag coefficient was chosen, with the use of particle–particle drag coefficient model of Syamlal [38]; and the Gunn's heat transfer coefficient and default KTGF in FLUENT were used.

3. Parametric study

3.1. The effect of air properties

Although the air properties were actually affected by temperature, in CFD simulation, they are usually set as constants. We therefore presented simulations using constants and temperature-dependent air properties (see Table 2) for evaluation. The effect of air properties on interphase momentum and heat exchange with and without temperature changes was assessed firstly. Figs. 2 and 3 plotted the effect of air properties on the Peng et al.'s drag coefficient and Gunn's heat transfer coefficient. In both figures, the drag coefficient and heat transfer coefficient are plotted as functions of slip velocity u_{si} (between 0 and 5 m/s) and operating bed temperature T (between 270 and 340 K) under a voidage of 0.98. Once air properties keep constant, the drag coefficient and heat transfer coefficient will only vary with slip velocity u_{si}. When the air properties vary with bed temperature, the effect of air properties on drag coefficient was not as obvious as on heat transfer coefficient as these figures show. It can be observed from Fig. 2 that the drag coefficient surface based on temperature-dependent air properties is slightly above the one based on constant air properties. Until u_{si} and T both reached relatively high values, an intersecting line can be seen clearly. Only when u_{si} is close to zero and T is relatively high, a visible difference can be found between the two drag coefficient surfaces. Different from the effect of air properties on drag coefficient, the effect on the heat transfer coefficient is somewhat significant as shown in Fig. 3. Around 10% higher heat transfer coefficient can be obtained by the temperature-dependent air properties than that with constant air properties. It can be explained that only two properties of air included in drag coefficient correlation, viscosity μ_g and density ρ_g, not vary so much with temperature, which leads to a nearly temperature-independent drag coefficient. Meanwhile, two additional air properties, thermal conductivity λ_g and specific heat C_{pg} included in the heat transfer coefficient correlation are more affected due to temperature change, which results in large differences observed in Fig. 3.

The analysis above has been confirmed by CFD simulation results shown in Fig. 4, where the temperatures of hot and cold sands are closer due to the higher heat transfer coefficient caused by the temperature-dependent air properties, whereas the solid holdups of both sands are not influenced so much. Therefore, temperature-dependent air properties will be chosen for modeling optimization instead of constant air properties, as they should be.