Applied Thermal Engineering 99 (2016) 928–937

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

A study on wall-to-bed heat transfer in a conical fluidized bed combustor

Hamada Mohamed Abdelmotalib, Dong Guk Ko, Ik-Tae Im *

Dept. of Mechanical Design Eng., Chonbuk National University, 567 Baekjedaero, Duckjin-gu, Jeonju, 561-756, Republic of Korea

ARTICLE INFO

Article history:
Received 21 April 2015
Accepted 12 January 2016
Available online 21 January 2016

Keywords:
Renewable energy
Biomass
Computational fluid dynamics
Conical fluidized bed
Heat transfer coefficient
Hydrodynamics

ABSTRACT

In this study, the flow characteristics and wall-to-bed heat transfer in a conical fluidized bed combustor (FBC) of height 0.8 m and cone angle of 30° were analyzed numerically and the results were compared to experimental ones. A two fluid Eulerian–Eulerian model coupled with kinetic theory of granular flow (KTGF) was used to simulate both hydrodynamic characteristics and heat transfer in a conical FBC. Hydrodynamic characteristics such as sand volume fraction, bed expansion, and pressure drop between two points at the cone part as well as heat transfer coefficient were compared to experimental data obtained under various operating conditions such as different superficial gas velocities and granular temperature models. Both heat transfer coefficient and pressure drop increased with increasing gas velocity. Use of a phase property model for granular temperature with slip conditions at the wall resulted in no clear effect in case of heat transfer coefficient, whereas there was better agreement between the experimental and numerical results for bed pressure drop when a partial differential equation model was used.

1. Introduction

Important and dramatic technical, economic, and operational advances in renewable energy technologies have been made over the past decade. There are several renewable energy sources such as biomass, wind, solar and geothermal energy for which lots of research effort has been made [1]. Combustion, pyrolysis and gasification are three main thermochemical conversion methods for converting biomass to energy [2]. Fluidized bed combustion technology is known to be the most efficient and environmentally friendly technology for conversion of energy from a wide diversity of solid fuels ranging from coal to agricultural residues to hazardous waste because of excellent gas–solid mixing, temperature homogeneity and ability to control emissions [3].

A fluidized bed combustor with a cone shape bed is the most suitable fluidized bed combustion technique for testing new alternative bed materials. A conical bed has been widely applied in many industrial processes such as biological treatment of waste-water, incineration of waste-materials, catalytic polymerization, coal gasification and liquefaction, coating of nuclear fuel particles, and fluidization of cohesive powder [4]. Compared to a columnar fluidized-bed combustion system fired with biomass, a conical fluidized bed combustor (FBC) has several benefits such as using a relatively small amount of inert bed material, shorter start-up time of the combustor and lower pressure drop across the fluidized bed for similar bed material and static bed height. Similar to any fluidized bed combustion system with a cylindrical or prismatic shape, the conical FBC ensures high combustion efficiency and acceptable levels of gaseous emissions when burning various types of biomass [5–7].

Heat transfer in fluidized beds has been the subject of intense research, and quite a few mechanistic and empirical models for bed-to-wall or wall-to-bed heat transfer have been proposed. However, mechanistic and empirical models have limitations because of the assumptions on which they were based or narrow range of the experimental data. Computational fluid dynamics (CFD) is a good method to overcome the limitations of mechanistic or empirical models. By using a CFD approach, numerical calculations of bed-to-wall (or vice versa) heat transfer are carried out by solving Navier–Stokes equations along with the thermal energy balance equation [8,9]. Modeling FBC hydrodynamics and heat transfer has become of interest over the past two decades because of the widespread industrial applications of FBC. Development of high-performance computers and advances in numerical techniques over the past several decades makes CFD analysis of multiphase systems possible, and CFD analysis is now considered a powerful tool for the design and development of FBCs [10].

Patil et al. [11] studied the wall-to-bed heat transfer in a cylindrical fluidized bed reactor using Eulerian–Eulerian model. Two types of equations for the solid phase rheology were considered, the kinetic theory of granular flow and the constant viscosity model. The predicted local instantaneous heat transfer coefficients were in good

* Corresponding author. Tel.: +82 63 270 4761; fax: +82-63-270-4761.
E-mail address: itim@jbnu.ac.kr (I.-T. Im).

http://dx.doi.org/10.1016/j.applthermaleng.2016.01.054
1359-4311/

agreement with the experimental data for different jet velocities as well as for different particle sizes, provided that the near wall porosity profile was used. Armstrong et al. [12] investigated the heat transfer from a heated wall in a gas-fluidized bed using the Eulerian–Eulerian approach. The local heat transfer coefficients were compared against experimental data for two different drag models, the Gidaspow and the Syamlal–O'Brien drag models. Both drag models results were in a good agreement with experimental data. In addition, the results showed that, the heat transfer coefficient gradually decreased and multiple bubbles were formed with increasing of inlet gas velocity. Dong et al. [13] studied the heat transfer in fluidized bed using two-fluid model coupling with kinetic theory for granular flow. The effect of effective thermal conductivity and different tube shapes on heat transfer from immersed tube to fluidized bed was studied. The simulations results showed that introducing solid particles around immersed tube strongly affected the heat transfer process. There were good agreement between simulations and experimental results. Hamzehei et al. [14] investigated heat transfer and hydrodynamics in a 2-dimensional fluidized bed reactor using Eulerian model coupling with k-ε turbulence model and kinetic theory for granular flow. Three exchange drag models namely, Syamlal–O'Brien, Gidaspow, and Cao–Ahmadi were used in the study. The simulations results such as temperature distribution, pressure drop, time-average local voidage profiles, and bed expansion ratio were in a good agreement with experimental results. The simulation and experimental results showed that the gas temperature decreases as it moves upward in the reactor, while the solid particle temperature increases. The results showed that, the numerical model was able to predict heat transfer and hydrodynamics in fluidized bed reactor with reasonable accuracy. Shu et al. [15] investigated the heat transfer of binary gas–solid flow in a downer reactor using multi-fluid model. The influence of constant air properties, particle–particle drag force, particle–particle heat transfer, and the different choices of kinetic theories of granular flow was estimated and the optimized models were identified. Computational fluid dynamics simulations with the optimized models showed that, CFD simulation had 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 could be obtained and be further improved by taking the specific shape of inlet distributor into account.

The influence of particle size on hydrodynamics and heat transfer of gas–solid fluidized bed reactor were investigated by Hamzehei and Rahimzadeh [16]. They used both the numerical and experimental methods. Both temperature and pressure drop for simulations at different particle sizes were in good agreement with experimental results at superficial gas velocity higher than the minimum fluidization velocity. Furthermore, the results indicated that, for smaller particle size, solid-phase temperature increases and mean gas temperature decrease. Sau and Biswal [17] studied the hydrodynamics in con fluidized bed using two different bed materials, glass beads and dolomite. The Eulerian–Eulerian model was used to simulate gas–solid flow and the drag model of Gidaspow was used to calculate the momentum exchange coefficient. The experimental results of both pressure and bed expansion agreed well with those were estimated from the simulations. Kaewklum and Kuprianov [18] modeled the hydrodynamics in conical fluidized bed using the bed material of sand. The minimum fluidization, minimum velocity of full fluidization, and pressure drop across bed were calculated at different particle sizes, different static bed heights and con angles. The relative computational errors for both minimum fluidization and minimum velocity of full fluidization were found to be within 20% while for pressure drop were within 10–15 %.

Hosseini et al. [19] studied transient heat transfer and hydrodynamics in spouted regime using two-fluid model. The effect of different models of radial distribution function on simulations results was studied. The results show that, the radial distribution function had effect in both particle velocity and temperature distribution in dense region. In addition to, there was a good agreement between simulations and experimental results. Wang et al. [20] investigated heat transfer in a rectangular spouted bed using discrete element method. The results indicated that, with increasing inlet gas velocity, the convective heat transfer was strong in jet region and weak in annular region. Particle–particle conductive heat transfer in jet and fountain regions is enhanced by increasing inlet gas velocity. The temperature of both solid particles and gas were analyzed in detail. Simulation results on the particle cooling process agreed closely with that in the experimental results. Hosseini et al. [21] studied hydrodynamics in 2-dimensional conical spouted bed using two-fluid model. The effect of different values of coefficient of restitution, different types of friction stress model, and drag model was studied. Particles velocities in both the axial and lateral direction in spouted regime and granular temperature were calculated. The simulations results were in a quite good agreement with experimental results. Szafran and Kmiec [22] used a CFD simulation to study heat and mass transfer during grain drying in spouted bed dryer. An Eulerian–Eulerian model was used to descript gas–solid flow behavior. Simulations results were compared with experimental results and with those obtained from different correlations. CFD simulations predicted very well the mass-transfer rate but under predicted the heat-transfer rate. The heat and mass transfer rates obtained from the correlations and CFD simulations were usually of the same order of magnitude, but the values obtained from the simulations were closer to the experimental data. Fattahi et al. [23] studied gas to particle heat transfer for fluidized and spouted regimes using an Eulerian–Eulerian model. The effect of specularity coefficient on simulations results was studied. The specularity coefficient critically affected the particles behavior. The temperature distribution and variation of particle concentration in both spouted and fluidized regimes were compared with experimental data for different three regions of spout, fountain and annulus. The CFD model simulated the particles distribution properly for both regimes. The simulations results for temperature distribution for the spouted regime were in a better agreement with the measurements, when compared with that for the fluidized regime.

Determination of the heat transfer and hydrodynamic characteristics of conical FBC is very important for their design and operation. However, research into the heat transfer and hydrodynamic characteristics of conical fluidized bed is still very limited compared with that into columnar fluidized bed combustors [15]. In our previous studies [24,25] we have studied heat transfer and hydrodynamics in conical fluidized bed reactor using only numerical approach. In the first report [24] the heat transfer from a heated wall to bed and related hydrodynamics were investigated using an Eulerian–Eulerian two-dimensional model. The effect of inlet gas velocity, particle size, and different drag models on both heat transfer and hydrodynamics was studied. The simulations results were compared to those obtained from available correlations. The results showed that the heat transfer coefficient increased with decreasing particle size and increasing inlet gas velocity. In other study [25] the influence of particle–particle and particle–wall coefficients of restitution on heat transfer and bed hydrodynamics was studied. Both particle–particle and particle–wall coefficients of restitution had little effect on heat transfer. The bed pressure drop increased with increasing particle–particle restitution coefficient and decreasing particle–wall coefficient of restitution. In this study, our simulations results of heat transfer and hydrodynamics in a conical fluidized bed reactor have been compared with experimental results instead of using correlations. A multi-fluid Eulerian model incorporating kinetic theory for solid particles was applied to simulate two-dimensional gas solid fluidized bed reactors.