Fuel 192 (2017) 187–200

Contents lists available at ScienceDirect

Fuel

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

Full Length Article

Numerical investigation of particle transport hydrodynamics and coal combustion in an industrial-scale circulating fluidized bed combustor: Effects of coal feeder positions and coal feeding rates

Massoud Massoudi Farid [a], Hyo Jae Jeong [b], Keun Ho Kim [c], Jongmin Lee [d], Dongwon Kim [d], Jungho Hwang [a,*]

[a] *Mechanical Engineering Department, Yonsei University, 50 Yonsei-ro, Seodaemun-gu, Seoul 03722, South Korea*
[b] *Corporate R&D Institute, Doosan Heavy Industries & Construction, 10 Suji-ro 112beon-gil, Suji-gu, Yongin 16858, South Korea*
[c] *Yeosu Power Plant, Korea South-East Power Corporation, 727 Yeosusandan-ro, Yeosu 59613, South Korea*
[d] *Green Power Generation Laboratory, Korea Electric Power Corporation Research Institute, 105 Munji-ro, Yeoseong-gu, Daejeon 34056, South Korea*

HIGHLIGHTS

- Particle transport was simulated in an industrial scale CFB combustor in 3D.
- Coal combustion was simulated in an industrial scale CFB combustor in 3D.
- Dense discrete phase model (DDPM) with several UDFs was used for the simulation.
- Two different coal feeder positions and coal feeding rates were investigated.
- Solid volume fraction, pressure, gas temperature, and gas species were reported.

ARTICLE INFO

Article history:
Received 26 September 2016
Received in revised form 6 December 2016
Accepted 9 December 2016

Keywords:
Circulating fluidized bed
Coal combustion
3D Computational fluid dynamics
Dense discrete phase model
Gas-particle hydrodynamics

ABSTRACT

This study investigates particle transport hydrodynamics and coal combustion in an industrial-scale circulating fluidized bed (CFB) combustor using the dense discrete phase model (DDPM). DDPM is an extension of the discrete phase model (DPM); however, unlike the standard formulation of DPM, DDPM considers the solid volume fraction when solving the Navier–Stokes equations for the gas phase. In the DDPM, the kinetic theory of granular flows is used to calculate the particle interaction in the Eulerian frame of reference. This interaction is then mapped to the particles in the Lagrangian frame of reference. In this study, user defined functions (UDFs) were used to extend the ANSYS FLUENT original code. These UDFs were used to reinject particles into to the combustor (cyclones were not modeled), calculate the pressure drop, circulation rate, and combustor mass load control. Various operation indexes such as distributions of gas temperature, solid volume fraction, pressure, and mass fractions of combustion products were displayed, and the selected indexes were compared with operating data obtained from a 340 MWe CFB combustor located in Yeosu, South Korea. The effects of both coal feeder positions and coal feeding rates on operation indexes were investigated.

1. Introduction

While worldwide interest in utilizing fluidized beds in various industrial processes such as coal combustion, biomass gasification, drying systems, granulation, catalysis, and coating has increased in recent decades, a complete and comprehensive knowledge of the phenomena occurring in fluidized beds is becoming increasingly necessary. One of the main applications of fluidized beds in industry is to produce energy using low-grade solid fuels in circulating fluidized beds (CFBs). A CFB, in its simplest form, consists of a riser (in which solid particles are fluidized), a cyclone (by which particles are separated from gas), and a return leg (through which particles return to the riser). In addition to considerable combustion efficiency, capturing sulfur emissions in a combustor using sorbents, and low nitrogen oxide emissions owing to low combustion temperature are the main reasons for the dramatic growth in the use of CFBs since the first CFB combustors were developed in the late 1970s [1]. Moreover, CFBs are flexible to use various low-grade fuels, including biomass and waste-derived fuels. Large-scale CFB units with new combustion processes that utilize various

---

* Corresponding author.
E-mail address: hwangjh@yonsei.ac.kr (J. Hwang).

http://dx.doi.org/10.1016/j.fuel.2016.12.025
0016-2361/

fuels have been developed recently. Therefore, an effective modeling tool is required to study solid–gas hydrodynamics and the mixing of different species in a combustor in order to optimize the design of the combustor layout for performance and emissions. Owing to the non-uniform feeding of fuel and air into the combustor, fluidization and combustion are spatially non-uniform processes, and hence need to be modeled three-dimensionally. Considering the factors mentioned above, a numerical simulation is an effective tool for determining different aspects of solid–gas hydrodynamics and fuel combustion in CFBs three-dimensionally.

There are two approaches for modeling gas-particle interaction numerically in dense beds: the Eulerian two fluid model (TFM) approach and the Eulerian–Lagrangian approach. In the TFM approach, solid and gas phases are both treated as interpenetrating continua, and the kinetic theory of granular flows (KTGF) approach is used to calculate particle interaction by solving an additional equation for solids fluctuating energy or granular temperature. The main disadvantage of the TFM approach is the long calculation time required for evaluating the time-averaged solution. The consideration of particle size distribution can significantly increase this time, as an additional dispersed phase for each particle size must be used [2]. In the Eulerian–Lagrangian approach, the gas phase is treated as a continuum while the solid phase is solved by tracking a large number of particles through the flow field calculated from the Navier–Stokes equations for the gas phase. All the forces applied on the particles are calculated to find their new positions and velocities. The solid phase can exchange momentum, mass, and energy with the gas phase.

In the discrete phase model (DPM), one of the models for the Eulerian–Lagrangian approach, a fundamental assumption is that the solid phase occupies a very low volume fraction, which is ignored in the Navier–Stokes equations for the gas phase. DPM can be used as either one way (fluid affects the particle motion only) or two ways coupled (particle motion affects the fluid motion as well) [3]. Additionally, In DPM the interaction between particles is neglected when solving the equation of motion for a particle. According to Elghobashi [4] a four way coupling is required already at even a low volume fraction of solids. Therefore, for modeling systems with particles of a high volume fraction such as those found in fluidized beds, DPM is inappropriate and instead a four ways coupled approach (fluid affects particle motion, particle affects the fluid motion, and particles affect each other [3]) is needed.

To consider the interaction between particles DPM can be extended for use in fluidized beds by considering the volume fraction of the solid phase in the Navier–Stokes equations for the gas phase. The volume fraction and velocity field of the solid phase come from the Lagrangian tracking solution, and the interaction between the solid particles is calculated at the Eulerian grid using solid stress–strain tensor, which is calculated using the KTGF approach. This method is considered under dense discrete phase model (DDPM) in ANSYS FLUENT [5]. In general, DDPM has common roots with the multiphase particle in cell (MP-PIC) method. The MP-PIC method is described in details in Refs. [6–10].

Several numerical simulations of reacting flow in fluidized beds are found in the literature. However, most of them employed 2D simulation, which cannot illustrate all features of the flow. Most of the 3D studies have simulated lab scale or pilot scale units and only a few have modeled industrial scale fluidized beds. Jung and Gamwo [11] developed a 2D TFM in-house code that included a reaction kinetics model for mimicking reactive flow behavior in a bubbling fluidized bed fuel reactor of a chemical looping combustor. Deng et al. [12] simulated fuel reactor flows and reactions in a bubbling fluidized bed reactor of a chemical looping combustor using the 2D TFM introduced in FLUENT. Papadikis et al. [13] studied the effects of different drag models on the fast pyrolysis of wood in a 2D simulation of a lab-scale bubbling fluidized bed reactor. The TFM in FLUENT was used to model the bubbling behavior of sand, which was treated as continuum, and the DPM was used to model discrete wood particles. Zhou et al. [14] applied a 2D TFM with a drag coefficient correction based on the extended energy-minimization multiscale (EMMS/matrix) model to simulate the air-coal two-phase flow and combustion characteristics of a 50 kW CFB combustor (riser). Reactions during coal combustion included moisture evaporation, devolatilization, volatile gas combustion, char combustion, and char gasification. The model predicted the main features of complex gas–solid flow, including cluster formation along the walls, and flow structures of upward flow in the core and downward flow in the annular region. FLUENT software was used for that simulation. Zhou et al. [15] utilized their model to predict coal combustion characteristics in an oxygen-fired CFB riser with wet flue gas recycling. Cornejo and Farías [16] simulated the coal gasification process inside a pilot fluidized-bed reactor (riser) using a 3D TFM, taking into account drying, volatilization, combustion, and gasification processes. Xie et al. [17] modeled the coal gasification process using a Lagrangian approach for both sand and coal particles in the same geometry used by Cornejo and Farías [16]. Adamczyk et al. [18] employed DDPM with KTGF in simulating particle transport and combustion in a pilot scale CFB, and showed that DDPM with KTGF can be used for predicting particle transport in fluidized beds working under the oxy-fuel combustion regime. Klimanek et al. [19] also used DDPM with KTGF to model a small-scale experimental facility in which coal was gasified in air and air/steam mixture. Adamczyk et al. [2] applied DDPM with KTGF to three-dimensionally model the dense gas–solid flow combined with a combustion process in a large-scale industrial CFB combustor. Adamczyk et al. [20] also modeled an industrial compact CFB combustor using DDPM-with-KTGF. The impact of geometrical model simplification on predicted mass distribution and temperature profiles was investigated. Moreover, some additional calculations were carried out to check the influence of radiative heat transfer on predicted temperature profiles within the CFB combustor. Distributions of species mass fractions were not illustrated in either Adamczyk et al. [2,20] studies.

In this study, a model was developed to investigate the transient 3D numerical simulation of air and sand particle hydrodynamics, along with coal combustion, in an industrial-scale CFB combustor. This study used the DDPM-with-KTGF approach to model the interactions of both sand (as an inert bed material) and coal particles with the gas phase. The ultimate goal was to investigate the effects of coal feeder positions and coal feeding rates, on particle transport hydrodynamics and coal combustion in an existing 340 MWe CFB combustor that is being operated by Korea South-East Power Corporation (KOSEP).

2. CFB combustor and coal

The 340 MWe CFB combustor of interest in this study is located in Yeosu, South Korea. The CFB combustor has four cyclones, as shown in Fig. 1(a). A mixture of hot sand particles, ash particles, and limestone particles are fluidized by primary air, which enters the combustor through nozzles located at the bottom of the combustor. Secondary air enters through side nozzles to improve fluidization, promote the combustion of coal particles which are injected into the combustor by coal feeders, and generate staged combustion in order to reduce NO_x emissions. Particles, which are carried by a mixture of air and combustion products to the top of the combustor, enter the cyclones and are separated from the combustion products. The sand particles then return to the combustor through return legs, while the combustion products