Powder Technology 318 (2017) 350–362

Contents lists available at ScienceDirect

Powder Technology

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

Computational particle-fluid dynamics simulation of gas-solid flow in a circulating fluidized bed with air or O_2/CO_2 as fluidizing gas

Mukesh Upadhyay, Hoon Chae Park, Jae Gyu Hwang, Hang Seok Choi *, Ha-Na Jang, Yong-Chil Seo

Department of Environmental Engineering, Yonsei University, Wonju, Gangwon-do 26493, Republic of Korea

ARTICLE INFO

Article history:
Received 6 January 2017
Received in revised form 17 May 2017
Accepted 6 June 2017
Available online 09 June 2017

Keywords:
Oxy-fuel
Circulating fluidized bed
Hydrodynamics
Computational particle-fluid dynamics
O_2/CO_2 mixture

ABSTRACT

Computational particle-fluid dynamics (CPFD) simulation was carried out to examine the influence of air versus O_2/CO_2 as the fluidizing gas on the hydrodynamics of gas-solid flow in a cold-mode circulating fluidized bed. The CPFD simulation results were compared to the experimental data at constant superficial gas velocity, using air or mixed O_2/CO_2 in three different concentrations as the fluidizing gas. The simulation results showed that the model successfully captured the experimentally observed trends. A detailed statistical analysis was carried out on the transient pressure data, and results were found to vary depending on whether air or combustion gases (O_2/CO_2) were used for fluidization. In all cases, the flow exhibits a typical core-annular flow structure, although for O_2/CO_2 gas the solid volume fraction increases near the wall region. The CPFD results provided insights into the gas-solid flow behavior in a fluidized bed combustor riser under an oxy-fuel fluidizing atmosphere.

© 2017 Elsevier B.V. All rights reserved.

1. Introduction

In recent years, many researchers have investigated oxy-fuel combustion technology as a means to achieve effective carbon capture and storage (CCS) objectives with commercial potential [1]. Oxy-fuel fluidized bed combustion combines the advantages of both oxy-firing and fluidized bed. Especially, oxy-circulating fluidized bed (CFB) combustors have several advantages and considerable potential for medium-scale units (typically <300 MWe) in terms of utility requirements [2]. These advantages include the ability to use a wider range of solid fuels and operating flexibility [3]. Moreover, oxy-CFB combustors operate at slightly above the atmospheric pressure, thereby avoiding the leakage issue [4]. Another advantage of these combustors is the low NO_X and SO_X formation compared to conventional CFB combustors [5]. Especially, they can easily switch from air mode to oxy-mode combustion, because the CFB contains inert material with a cyclone to recirculate solid particles and help to maintain the bed temperature. All these factors make this technology a viable option in the energy sector.

Over the years, simulation tools such as computational fluid dynamics (CFD) have been immensely useful for improving reactor engineering by replacing expensive and difficult experimental procedures. CFD-based methods are now routinely applied to understand the macroscopic behavior in fluidized beds [6,7]. CFD simulation of gas-solid flow can be divided into two categories: the Eulerian-Eulerian two-fluid model (EE-TFM) and the Eulerian-Lagrangian (EL) method. Eulerian- Eulerian method has become popular for simulating gas-solid flows due to its lower computational costs [8,9,10]. In this method, the gas and solid phases are described as two interpenetrating continua based on the kinetic theory of granular flows (KTGF) [11], thus the transient characteristics of individual particles cannot be captured clearly [12]. Additionally, TFM lacks the ability to incorporate specific particle types and particle size distribution (PSD), factors that clearly influence the gas-solid flow behaviors [13]. On the other hand, the Eulerian-Lagrangian method provides the trajectory of each particle. EL-based approaches, such as the discrete phase method (DPM) and the discrete element method (DEM), are generally considered reliable tools for investigating the dynamic behavior of gas-solid fluidized beds [14]. However, the high computational cost makes DPM and DEM unsuitable for large-scale systems [15]. In recent years, the advent of high-performance parallel computing based on graphical processing units (GPUs) has brought great potential for rapid simulation. Especially, rapid simulation of commercial-scale processes has received a boost from improved Eulerian-Lagrangian-based simulation with hybrid simplification, such as the coupling of lattice Boltzmann method (LBM) with the DEM framework [16,17], the smoothed particle hydrodynamics (SPH) method [18,19], and the multiphase particle-in-cell (MP-PIC) method [20,21]. A MP-PIC-based commercial computational particle fluid dynamics (CPFD) code, Barracuda, has the advantages of including full particle size distribution and fast, accurate solution. It has become a specialized tool to predict gas-solid flows, and is used in the current work.

Different fundamental aspects of oxy-fuel combustion have been addressed in the literature, such as heat and mass transfer effects

* Corresponding author at: Department of Environmental Engineering, Yonsei University, 1 Yonseidae-gil, Wonju, Gangwon-do 26493, Republic of Korea.
E-mail address: hs.choi@yonsei.ac.kr (H.S. Choi).

http://dx.doi.org/10.1016/j.powtec.2017.06.021
0032-5910/© 2017 Elsevier B.V. All rights reserved.

under oxy-fuel condition [22,23], pollutant emissions [24,25], ash characteristics [26], and kinetic behavior for different solid fuels [27,28]. Further, hydrodynamic behavior under oxy-fuel fluidizing atmosphere is equally important factor, as it could affect the design and operation of the fluidized bed. Only a few researchers have investigated the hydrodynamic characterization under oxy-fuel conditions. Komorowski et al. [29] and Sarbassov et al. [30] investigated the effect of air and O_2/CO_2 fluidizing gas conditions on the axial profile. Their experiments showed that at low superficial gas velocity distinct axial profile observed between air and oxy (O_2/CO_2) operating conditions, while differences become less at higher superficial gas velocity. Guedea et al. [31] investigated the fluid dynamics using semi-empirical one-dimensional model in a bubbling fluidized bed, and reported that high CO_2 concentrations lead to higher bed voidage and expansion.

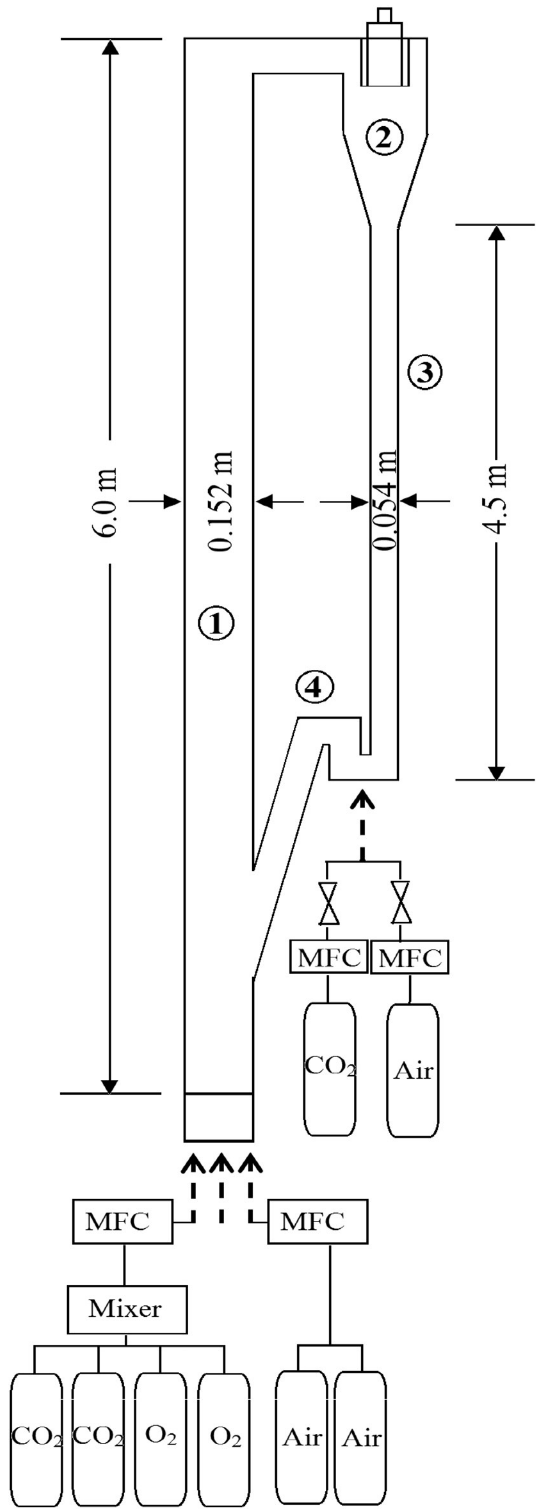

Fig. 1. Schematic diagram of the cold-mode CFB. (1—riser; 2—cyclone; 3—downcomer; 4—loop-seal).

Gas and solid phase hydrodynamics is widely recognized to influence the overall performance of a CFB riser [32]. Therefore, it is important to investigate the hydrodynamic difference between air and combustion gas (O_2/CO_2) in the design and analysis of CFBs. From the experiment, it is difficult to obtain the detailed information in the entire space of the bed. However, validation of CFD simulation with experimental results may provide accurate tempo-spatial information that cannot be accessed from experiments alone. Thus, the CPFD model in the present work was validated against experimental data obtained under different conditions, and the results were used as a benchmark to analyze the hydrodynamic flow pattern. Moreover, CPFD simulation of full-loop CFB has an advantage over that of only the riser section [33], because the full-loop simulation provides more detailed information, such as the influence of various non-uniform components and the associated complex physics. Hence, 3D full-loop CFB simulation was performed in the present study in order to investigate the hydrodynamic characteristics.

2. Experiment

The cold-mode CFB reactor (Fig. 1) has a riser with an overall height of 6 m and an inner diameter of 0.152 m. Solid particles were separated from the gas stream in the single-stage cyclone separator, and returned to the bottom section of the riser through the downcomer (i.d.: 0.054 m, height: 4.5 m) and loop-seal. The CFB system was operated under conventional air or oxy modes. For the oxy modes, O_2 and CO_2 gases were mixed prior to entering the reactor in the following ratios: O_2-20%/CO_2-80% (oxy-20%), O_2-30%/CO_2-70% (oxy-30%), and O_2-40%/CO_2-60% (oxy-40%).

Seven online pressure transducers (Sunyoung systec INC, Korea) were connected to pressure taps at different heights of the riser. Four pressure transducers were connected to the walls of the downcomer of CFB. A solid inventory of 13 kg was initially loaded into the CFB for the experimental run. In the experiment, silica sand particles (Geldart group B) were used as fluidizing media. The particle size distribution is shown in Fig. 2. The system geometry, material properties, and

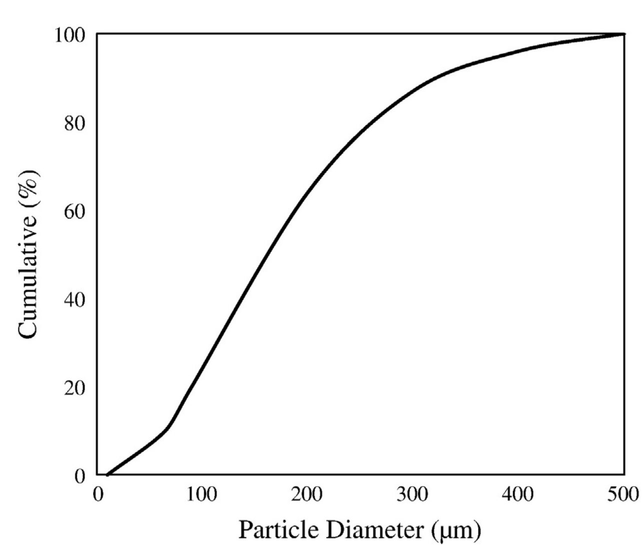

Fig. 2. Silica sand particle size distribution.

Download English Version:

https://daneshyari.com/en/article/4914935

Download Persian Version:

https://daneshyari.com/article/4914935

Daneshyari.com