Powder Technology 234 (2013) 97–106

Contents lists available at SciVerse ScienceDirect

Powder Technology

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

Hydrodynamics of cold-rig biomass gasifier using semi-dual fluidized-bed

Son Ich Ngo [a], Young-Il Lim [a,*], Byung-Ho Song [b], Uen-Do Lee [c,d], Chang-Won Yang [c,d], Young-Tai Choi [c], Jae-Hun Song [e]

[a] Lab. FACS, RCCT, Department of Chemical Engineering, Hankyong National University, Gyonggi-do, Anseong-si, Jungangno 167 456-749, Republic of Korea
[b] Department of Chemical Engineering, Kunsan National University, Gunsan, Jeonbuk 573-701, Republic of Korea
[c] Energy System R&D Group, Korea Institute of Industrial Technology (KITECH), Cheonan 331-825, Republic of Korea
[d] Department of Green Process and System Engineering, University of Science and Technology (UST), Cheonan 331-825, Republic of Korea
[e] 1501 SeenTec Tower 74-6 Sangnam-dong, Seongsan-gu, Changwon City, Gyeongnam 642-831, Republic of Korea

ARTICLE INFO

Article history:
Received 21 August 2012
Accepted 14 September 2012
Available online 23 September 2012

Keywords:
Gasification
Semi-dual fluidized-bed (sDFB)
Solid circulation rate
Internal mixing
Computational fluid dynamics (CFD)
Eulerian–Eulerian model

ABSTRACT

This study reports a semi-dual fluidized-bed (sDFB) biomass gasifier, which is a novel design of dual fluidized-bed (DFB) with the internal mixing of solid particles between riser and gasifier to enhance the heat and mass transfer. A cold-rig experiment of sDFB (0.8 m width × 0.2 m depth × 3.85 m height) was performed to investigate fluid hydrodynamics and solid circulations. Pressures were sampled at 43 points along the sDFB gasifier. An external circulation rate of sand was measured for 60 s after 2 min of the operating time. In order to estimate the amount of direct back-mixing particles through the gasifier–riser interconnection area, an Eulerian–Eulerian two-dimensional computational fluid dynamics (CFD) model was developed for the cold-rig sDFB. This CFD model included the kinetic theory of granular flow and the k– dispersed turbulence model. The CFD simulation results were validated with the experiment data. About 17% back-mixing of particles through the gasifier–riser interconnection area were obtained from the CFD simulation. This indicates that the sDFB has a possibility of having higher heat and mass transfer than the conventional DFB.

1. Introduction

Nowadays, interest in biomass gasification has an accelerative development, since it is renewable, sustainable, abundantly available everywhere in the world, and the increased use of biomass can reduce the petroleum dependence [1,2]. Biomass gasification is the process by which organic matter is thermally devolatilized, followed by secondary reactions of the resulting products [3]. The chemical energy of the solid fuel is converted into both the thermal and chemical energy of the gas. Chemical energy contained within the gas is a function of chemical composition. Thus the chemical composition of the producer gas determines its quality as a fuel [4]. Furthermore, the producer gas of biomass gasification is used as feedstock in some upgrading systems for generating energy and fuels in a much cleaner manner. This is suitable for energy demand in the future, owning to reduce the net of carbon dioxide emission while increasing environmental safety [1]. However, biomass gasification has been known as a complex process due to the complicating nature of biomass composition.

Fluidization has been widely used industrially because of its continuous handling ability of solid particles and its good heat and mass transfer characteristics [5]. In the conventional dual fluidized-bed (DFB) system, the heat required for endothermic reactions in the gasifier is provided with solid particles (sand) transported from the combustion zone (riser) to the gasification zone (gasifier). Thus, the amount of circulating solids indicates the energy demand for the gasification process [6].

Recently, researchers have addressed the relationships between the solid circulation rate and other factors in DFB gasifiers such as the heat efficiency [6,7], breakage and attrition effects [8], and the stability of the loop-seal [9]. Using a gasifier with a higher capacity requires more energy and a higher solid circulation rate. However, Shen et al. suggested that more solid circulation rate could lead to more breakage and attrition by the hot circulating particles [8]. Seo et al. reported that the solid circulation rate has to be maintained above a certain amount for the loop-seal to be stable [9]. From our experience [6,7], the solid circulation rate should be carefully selected by considering many performance criteria such as heat efficiency, lower heating value, and additional fuel ratio.

Computational fluid dynamics (CFD) modeling has become a viable tool for investigation on hydrodynamics of various processes with the aid of increasing computational capacity. However, CFD is still at the verification and validation stage for modeling multiphase flow systems such as fluidized-beds. More improvements regarding the flow dynamics and computational models are required to make CFD suitable for fluidized-beds modeling and scale-up [10,11].

In CFD, the modeling of gas–solid hydrodynamics is generally divided into two main approaches. The Eulerian–Lagrangian approach is also called discrete particle modeling. The gas phase is calculated

* Corresponding author. Tel.: +82 31 670 5207; fax: +82 31 670 5445.
E-mail address: limyi@hknu.ac.kr (Y.-I. Lim).

0032-5910/$ – see front matter
http://dx.doi.org/10.1016/j.powtec.2012.09.022

using an Eulerian framework and the trajectories of particles are treated in a Lagrangian framework. The interface between the gas and solid phases is computed by an average value of the area bounded by a number of particle trajectories. Hence, a large number of particle trajectories should be simulated to obtain a meaningful average of all quantities [11]. The Lagrangian model is normally limited to a relatively small number of particles because of computational expense [10]. Taking into account the available computational capacity, this approach is suitable for modeling dispersed multiphase flows containing a low volume fraction of solid particles [11].

The second approach, Eulerian–Eulerian model, also called the granular flow model, is considered to be the most common approach for fluidized-bed simulation [12]. Both phases are treated as an Eulerian framework based on the interpenetrating continuum assumption. The equations employed are a generalization of the Navier–Stokes equations for interacting continua [13]. Owning to the continuum presentation of particulate phases, the Eulerian models require additional closure laws in terms of constitutive equations to describe the rheology of particles. In this approach, the trajectories of particles are obtained at a hypothetical level rather than at a physical level in comparison with the Lagrangian model. The Eulerian model makes it possible to be applied to multiphase flow processes containing a large volume fraction of solid particles [13–15], such as fluidized-bed gasifier.

Among the various attempts to formulate the particulate flow in the Eulerian framework, the kinetic theory of granular flow (KTGF) is commonly used in fluidization [16]. This theory is basically developed from an extension of the classical kinetic theory of gases [17] to be applied in dense particulate flows. The turbulent kinetic theory of particles is introduced to the granular temperature concept, which is like a thermal temperature in the kinetic theory of gases [5]. However, it measures the random oscillations of particles in the solid phase. The granular temperature can be expressed mathematically in terms of generation of fluctuations by shear, dissipation by kinetic and collisional heat flow, dissipation due to inelastic collisions, generation due to fluid turbulence, and dissipation due to interaction with the fluid [13]. By introducing the KTGF, the models predicted the bubble formation, the distribution of time-averaged solid concentration in bubbling fluidized-beds [18–21], cluster formation, the time-averaged solid concentration, and mass flux distributions in circulating fluidized-beds [22–26].

The CFD methods solve the Navier–Stokes equations of gas and solid flows. Fluidization is a kind of turbulent flow rather than a laminar flow. Among several turbulence models which can be applied to the Eulerian–Eulerian approach, the dispersed turbulence model based on the Wilcox k–ϵ model [27] was used in a fluidized-bed [28]. This multiphase dispersed turbulence model consists of the productions of the turbulence kinetic energy (k) and the specific dissipation rate (ϵ) for the continuous phase (gas) [27], the predictions of turbulence quantities for the dispersed phase (solid) [29,30], and the correlation between the instantaneous distribution of the dispersed phase and the turbulent fluid motion (turbulent drag force).

The purpose of this paper is to propose a novel semi-dual fluidized-bed (sDFB) gasifier. Since this sDFB gasifier has both an external solid circulation from the loop-seal to the gasifier and an internal solid circulation between the riser and the gasifier, the heat and mass reciprocations between riser and gasifier may increase under well-operated conditions. It is therefore expected to reduce the external solid circulation rate and the breakage and attrition of particles, keeping the same system energy demand. A cold-rig experimental apparatus was designed and constructed to investigate the hydrodynamics of the sDFB gasifier. A two-dimensional CFD simulation with an Eulerian–Eulerian two-fluid model using a commercial code was developed to evaluate the internal solid circulation rate. This rate is expressed by direct back-mixing particles through the gasifier–riser interconnection area. After the verification of the CFD model with experiment data of pressure distribution and external solid circulation rate, the internal solid circulation rate was estimated. This study shows that the sDFB has a potential for further development with its advantages of heat and mass transfer over the conventional DFB.

2. Semi-dual fluidized-bed gasifier

A cold-rig sDFB gasifier (0.8 m width × 0.2 m depth × 3.85 m height) was built. A rectangular internal hole (0.18 m width and 0.05 m height) between the gasifier and riser was located at 0.345 m from the gas distributor of the riser. The basic concept of the sDFB gasifier is illustrated in Fig. 1. Like the conventional DFB, the sDFB gasifier is divided into 5 zones: riser, gasifier, loop-seal, stand pipe and cyclone, in which the aerations are introduced to the riser, gasifier and loop-seal to make fluidizations [31–33].

The sand particles are used as the heat carrier in circulating fluidization systems [6,7,9] due to their good physical, mechanical, and handling properties. The feedstock (biomass) and sand particles are stored at the gasifier zone, herein, the gasification reactions take place. The riser burns char residue and additional fuel to heat up the sand particles. Flue gas and sand particles are then separated by the cyclone. Sand particles follow the stand pipe and come back to the gasification zone via the loop-seal.

The heat required at the endothermic gasification zone is provided by the hot sand particles which indirectly come from the loop-seal and directly come from the riser. The circulation rate of particles indirectly transported via the loop-seal is defined as the external circulation rate of solid particles. The rate of particles directly reciprocated between the riser and the gasifier through the internal hole is termed by the internal circulation rate of solid particles.

The cold-rig sDFB experiment was carried out to measure pressures and external solid circulation rates of the system. 43 points along the riser, gasifier, cyclone, stand pipe and loop-seal were used to measure the pressures. As illustrated in Fig. 1(a), those are 18 points in the riser and cyclone (P1–18), 16 points in the stand pipe and loop-seal (P19–34), and 9 points in the gasifier (P35–43). The pressure at the exit of the cyclone (P18) was used as the base pressure.

The inlet velocities of the riser, gasifier, and loop-seal were set to 0.85, 0.08 and 0.07 m/s, respectively. The size distribution of sand classified by the Geldart group B is shown in Fig. 2(a), which was measured by the laser diffraction technique (Mastersizer 2000, Malvern Instruments Ltd., UK). The mean particle size was 376 μm. In Fig. 2(b), the minimum fluidization velocity (v_{mf}) was determined as 0.067 m/s. The initial silica sand heights of the gasifier, combustor, and loop-seal were 0.36, 0.36 and 0.40 m, respectively. The sand properties and operating conditions for the experiment are summarized in Table 1.

After about 2–3 min of the operating time, the gas–solid flows inside sDFB reached a stable state. To obtain the average external solid circulation rate, the sand height accumulated for 60 s above the loop-seal (sampling points 23–25 in Fig. 1(a)) was measured without the inlet aeration of the loop-seal.

It is not trivial to measure the internal solid circulation rate by experiment because of the chaotic behaviors of sand particles at the interconnected hole between the gasifier and the riser. A CFD study was needed to estimate this value which plays a key role in the heat and mass transfer enhancement of sDFB.

3. Eulerian–Eulerian two-fluid CFD model

The Eulerian–Eulerian approach with the multiphase k–ϵ dispersed turbulence model consists of a set of continuity and momentum equations for the gas and solid phases (see Appendix A). Only one link between the two phases is drag coefficient. There are several drag models [16,34,35] of which the Gidaspow drag model [16] was used in this study due to its suitability for the present particles and flow regime, as shown in Appendix A.2. The kinetic theory of granular