Powder Technology 275 (2015) 188–198

Contents lists available at ScienceDirect

Powder Technology

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

Effects of fluidization velocity on solid stack volume in a bubbling fluidized-bed with nozzle-type distributor

Son Ich Ngo [a], Young-Il Lim [a,*], Byung-Ho Song [b], Uen-Do Lee [c,d], Jeung-Woo Lee [e], Jae-Hun Song [e]

[a] Center of Sustainable Process Engineering (CoSPE), Department of Chemical Engineering, Hankyong National University, Jungangno 327, Anseong-si, Gyeonggi-do 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 Hansol SeenTec Tower, 74-6 Sangnam-dong, Seongsan-gu, Changwon City, Gyeongnam 642-831, Republic of Korea

ARTICLE INFO

Article history:
Received 6 November 2014
Received in revised form 2 January 2015
Accepted 8 February 2015
Available online 16 February 2015

Keywords:
Bubbling fluidized-bed
Nozzle-type distributor
Fluidization velocity
Computational fluid dynamics (CFD)
Eulerian two-fluid model
Solid stack volume (SSV)

ABSTRACT

Hydrodynamic characteristics in a wind-box and bubbling fluidized-bed (BFB) gasifier zone of a dual fluidized-bed (DFB) were investigated by a two-phase three-dimensional computational fluid dynamics (CFD) model. The gas and solid phases were treated by an Eulerian–Eulerian two-fluid model, coupled with the realizable *k*–epsilon turbulence model and the kinetic theory of granular flow (KTGF) describing the random motion of solid particles. Pressure drops obtained from the cold-rig CFD simulation were validated with experimental data which were measured in a pilot-scale BFB using air as a fluidization agent and sand as heat carrier particles at an operating temperature of 800 °C. Hydrodynamics of the fluidized-bed with a uniform gas distributor (Ideal case) and a nozzle-type gas distributor (Real case) were evaluated in terms of the pressure drop, solid volume fraction (SVF), uniformity index (UI), and solid stack volume (SSV) for three inlet air flow rates (low, medium and high). Similar behaviors were shown for both the two cases in pressure drop along the gasifier height. However, significant differences were observed in SVF, UI, and SSV. A threshold changing the slope of SSV to the air flow rate was found at a fluidization index (u/u_{mf}) of 2.9.

1. Introduction

Dual fluidized-bed (DFB) has been considered as one of the most suitable technologies for biomass gasification to produce middle heating value gas free of serious dilution by N_2 of combustion air [1,2] and for an excellent ability to scale-up [3,4]. It was well demonstrated that the dual fluidized-bed gasifier (DFBG) is a good combination of a bubbling or turbulent fluidized-bed gasifier and a riser combustor both to facilitate gasification reactions and to suppress tar formation [5]. There are many parameters affecting the gasification system such as feedstock, geometry, circulating bed materials, operating temperature, hydrodynamics and heat transfer [6].

The importance of the gas distributor design (critical pressure drop ratio, hole size, geometry and spacing) was addressed in the light of uniform gas distribution, bubble formation, pressure drop, dead zones, and particle attrition [7]. Experimental and numerical studies were performed for a nozzle-type distributor (tuyere with horizontal holes) to identify the main source of pressure drop across the distributor [8]. Here, the pressure drop was measured and calculated according to the tuyere configuration.

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

http://dx.doi.org/10.1016/j.powtec.2015.02.017
0032-5910/

The hydrodynamics of gas and solid flows have been investigated for appropriate DFB design and operating conditions. Among various studies, computational fluid dynamics (CFD) has been shown as one of the most useful tools for investigating the complex gas and solid flows inside the system [9,10]. There are two general approaches for CFD modeling and simulation of DFB: The Eulerian–Eulerian (EE) and Eulerian–Lagrangian (EL) methods [9]. In the EE approach, both gas and solid phases are treated as continuous flows, where the solid phase is represented as pseudo-particles. The kinetic theory of granular flow (KTGF) [11] is usually applied to the EE models for accounting the flow property of the pseudo-particles.

In the EL approach, the gas phase is continuous but the solid particles are discrete. The interface between the gas and solid phases is computed by an average value of the area bound by a number of particle trajectories. The EL approach is normally limited to a relatively small number of particles because of computational expense [12]. Taking into account the computational capacity, the EE approach is an alternative for the preliminary investigation of gas and solid flow characteristics. Moreover, various sub-models have been introduced into the EE approach, making it possible to represent the gas–solid interaction with better accuracy.

In studies of Nguyen and Seo et al. [10,13], the gas and solid flow characteristics were analyzed and compared with experimental data for a pilot-scale DFB gasifier. Our recent study [9] reported

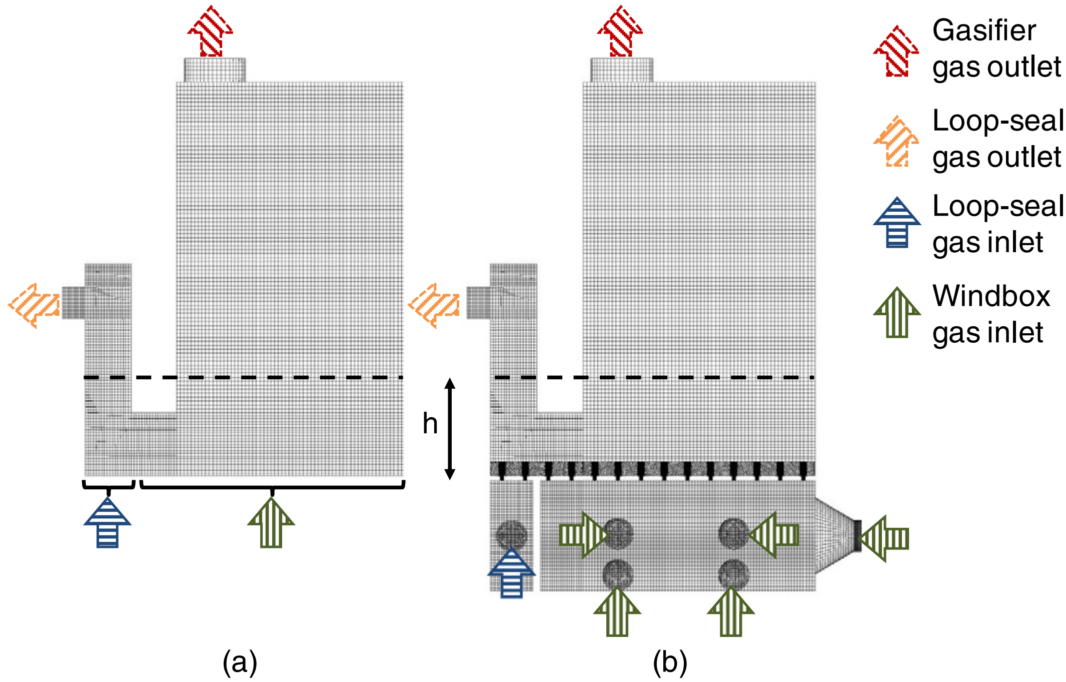

Fig. 1. Mesh structure of CFD geometric domain in front view: (a) Ideal distributor with uniform air injection, and (b) real distributor with nozzle-type air injection.

hydrodynamics in a semi-dual fluidized bed (sDFB) with internal and external solid circulations to improve heat transfer between the riser and gasifier and to regulate the external solid circulation rate. However, this research was performed in a 2D simplified domain, and the effects of the distributor structure on hydrodynamics were omitted. There remains a need for taking into account a 3D real

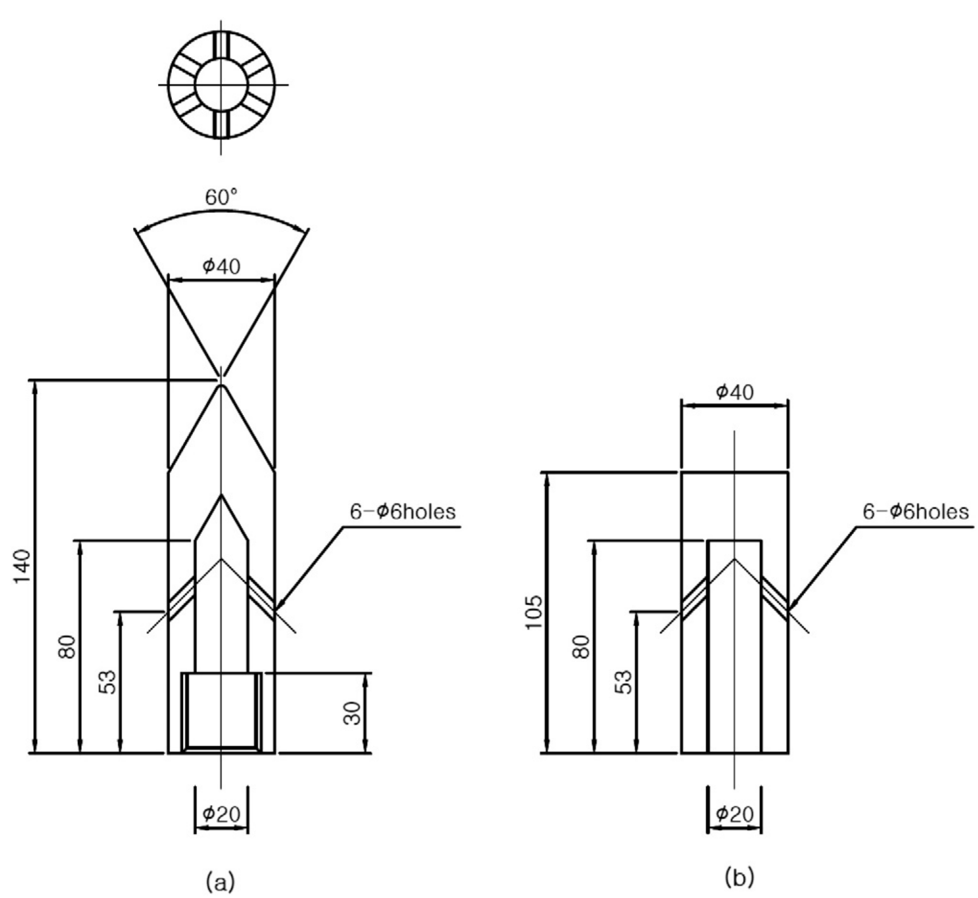

Fig. 2. Nozzle-type air injector: (a) Installed design, (b) CFD design.