Computers and Chemical Engineering 92 (2016) 180–188

Contents lists available at ScienceDirect

Computers and Chemical Engineering

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

Circulating fluidized bed combustion reactor: Computational Particle Fluid Dynamic model validation and gas feed position optimization

R.K. Thapa[a,*], A. Frohner[b], G. Tondl[b], C. Pfeifer[b], B.M. Halvorsen[a]

[a] Institute for Process, Energy and Environmental Technology, Telemark University College, 3901 Porsgrunn, Norway
[b] University of Natural Resources and Life Sciences, Institute of Chemical and Energy Engineering, Muthgasse 107, A-1190 Vienna, Austria

ARTICLE INFO

Article history:
Received 7 March 2015
Received in revised form 21 April 2016
Accepted 15 May 2016
Available online 24 May 2016

Keywords:
CFB
Biomass gasification
Dual fluidized bed
CPFD

ABSTRACT

A 3D Computational Particle Fluid Dynamic (CPFD) model is validated against experimental measurements in a lab-scale cold flow model of a Circulating Fluidized Bed (CFB). The model prediction of pressure along the riser, downcomer and siphon as well as bed material circulation rates agree well with experimental measurements. Primary and secondary air feed positions were simulated by varying the positions along the height of the reactor to get optimum bed material circulation rate. The optimal ratio of the height of primary and secondary air feed positions to the total height of the riser are 0.125 and 0.375 respectively. The model is simulated for high-temperature conditions and for reacting flow including combustion reactions. At the high temperature and reaction conditions, the bed material circulation rate is decreased with the corresponding decrease in pressure drop throughout the CFB for the given air feed rate.

1. Introduction

Circulating fluidized bed reactors are widely used in various industrial applications including oxyfuel combustion, gasification and combustion of biomass or other carbonaceous feedstock. One of the applications of CFB in gasification processes is heating bed materials by combustion of fuels and then transporting them to the gasification reactor (Pfeifer et al., 2009). Fluid dynamic properties of the reactor, including gas-particle mixing and residence time, depend on the gas velocity and particle circulation rate for a given bed inventory (Ludlow et al., 2008). Gas velocity and bed material circulation rate are significant parameters determining the performance of the reactor. The solid circulation rate is also crucial for reaction kinetics. The solid circulation rate determines the fluid – solid contact time, heat transfer and overall performance of the CFB as a rector (Roy et al., 2001). The solid circulation rate and solid distribution over the circulating system are determined by the fluid dynamics (Lei and Horio, 1998) of the reactor. To obtain a proper distribution of the solids throughout the CFB, proper pressure balance is required (Kaiser et al., 2001). Improvement of performance of CFB mainly needs optimum fluid dynamic properties of the bed. Many researchers have studied the fluid dynamic behavior of CFB. Yerushalmi et al. have shown the transition between packed bed, bubbling bed, turbulent and fast fluidization regimes in the plot of bed voidage against superficial gas velocities (Yerushalmi et al., 1976). Flow regime maps of gas-solid flow are also developed plotting gas velocity against the solid flux (Leung, 1980). Takeuchi et al. performed experimental measurements to define the boundaries of fast fluidization (Takeuchi et al., 1986). Hirama et al. extended the flow diagram to transition from high velocity to low-velocity fluidization regimes (Hirama et al., 1992). Bi and Grace proposed unified flow regime diagram based on the experimental findings. They have shown the relationship between flow regimes for both gas-solid fluidization and co-current upward transport (Grace, 1986; Bi et al., 1993; Bi and Grace, 1995). In the all of the mentioned studies, the experiments are carried out at the ambient conditions. One of the significant factors affecting overall fluid dynamic properties of the bed is particle size distribution. The particle size distribution is not included in all the studies mentioned above. When particles of larger sizes and lower densities are mixed with the particles of smaller sizes and higher densities, the minimum fluidization velocity changes (Thapa et al., 2011). Change in minimum fluidization velocity effects on the transport velocity and fast fluidization velocity. High-temperature gasses have lower density and higher viscosity. Change in density and viscosity changes flow behavior in fluidized bed.

Therefore, the study of fluid dynamics in CFB should include the particle size distribution and the effect of high-temperature con-

* Corresponding author at: Kjølnes ring 56, P.O. BOX 203 N3901, Porsgrunn, Norway.
E-mail addresses: rajan.k.thapa@hit.no, rajankumarthapa@hotmail.com (R.K. Thapa).

http://dx.doi.org/10.1016/j.compchemeng.2016.05.008

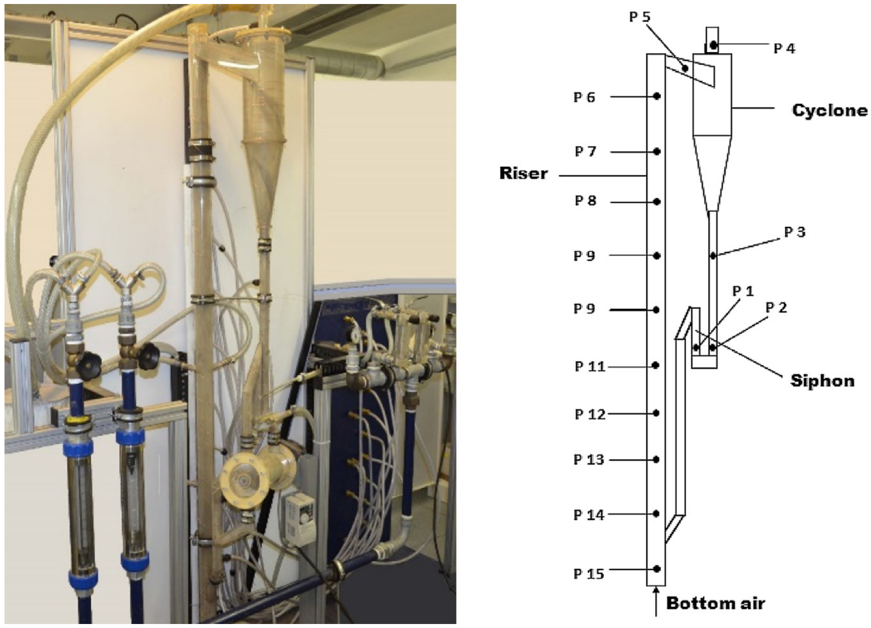

Fig. 1. (a) CFB cold model with air flow regulation and pressure measurement arrangements (b) pressure tapping points.

ditions. The gas-solid flow without chemical reaction differs from the reacting flow. The significance of those differences is not studied yet. The fluid dynamic properties of the CFB used in gasification of biomass is complicated due to the gas feed positions at different levels of the reactor. Air is fed to the reactor as bottom air at the bottom of the riser and primary and secondary air are fed along the height of the rector (Kaushal et al., 2008a,b). When the gas is fed at three positions of the bed, the feed position itself is expected to affect strongly on the fluid dynamics of the bed and the bed material circulation rate.

Design, scaling, operation and improvements of the circulating fluidized bed reactors require a good understanding of the fluid dynamic parameters affecting the performance (Gungor and Yildirim, 2013). Many of those parameters can be studied by the experimental investigations using laboratory models, pilot and demonstration plants. However, not all parameters are easy to study using experimental methods. For example, the study of the effect of the gas feed positions needs reconstruction of the reactor, which takes a lot of time and can be economically too costly. These facts indicate the usefulness of the computational models to overcome those types of challenges.

To overcome and/or substitute the experimental limitations, computer models have gained significant attention since the early 1990s. Computer models make it possible to study the fluid dynamics without disturbing the fluid flow field inside the reactor (Deen et al., 2007). The current work is therefore, focused on a validation of the CPFD (Computational Particle Fluid Dynamic) model against experimental measurements performed in a cold flow model of a CFB. The model is then used for further investigations of high temperature and reaction conditions as well as optimizing the feed positions of primary and secondary air.

2. Experimental set-up

The experimental rig is located at University of Natural Resources and Life Sciences (BOKU), Vienna, Austria. The set up consists of a cold model of a circulating fluidized bed as shown in Fig. 1. The model is made of a transparent Plexiglas, which makes it easier to visualize the fluidization and particle circulation during the experiments. The model is wrapped with conductive wires to avoid electrostatic forces at the wall. The fluidizing gas used in the experiment is ambient air supplied from a compressor. The fluidizing gas is fed as bottom air and primary air at two different stages of the reactor. The volume flow of primary and secondary air is measured through rotameters shown in Fig. 1a. The setup has 15 pressure tapping points which are connected to the pressure sensors. An industrial measurement and control system (B&R automation) is used to log the pressure data. The heights of pressure tapping points are shown in Table 1.

Table 1
Location of the pressure tapping points.

Labelling	Location	Height [mm]
P1	Siphon	665
P2	Siphon	665
P3	Downcomer	1010
P4	Exit Filter	1685
P5	Intersection Precipitator	1595
P6	Reactor	1535
P7	Reactor	1330
P8	Reactor	1170
P9	Reactor	1005
P10	Reactor	850
P11	Reactor	610
P12	Reactor	525
P13	Reactor	365
P14	Reactor	205
P15	Reactor	40

The siphon shown in Fig. 1b is also fluidized by air. The particles used in the experimental investigations are sand particles of density 2650 kg/m^3. The particle size distribution is presented in Fig. 2.

3. Computational model

In this work, a Computational Particle Fluid Dynamic (CPFD) model is used to simulate the gas-solid flow with heat transfer and chemical reactions. The commercial CPFD software Barracuda VR 15 is used for the simulations. The CPFD numerical methodology incorporates multi-phase-particle-in-cell (MP-PIC) method (Andrews and O'rourke, 1996; Snider, 2001). The gas phase is solved using the Eulerian approach and the particles are modeled