Powder Technology 253 (2014) 522–536

Contents lists available at ScienceDirect

Powder Technology

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

In-depth system parameters of transition flow pattern between turbulent and fast fluidization regimes in high solid particle density circulating fluidized bed reactor

Benjapon Chalermsinsuwan [a,b,*], Dimitri Gidaspow [c], Pornpote Piumsomboon [a,b]

[a] Fuels Research Center, Department of Chemical Technology, Faculty of Science, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand
[b] Center of Excellence on Petrochemical and Materials Technology, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand
[c] Department of Chemical and Biological Engineering, Illinois Institute of Technology, 10 West 33rd Street, Chicago, IL 60616, United States

ARTICLE INFO

Article history:
Received 9 June 2013
Received in revised form 2 December 2013
Accepted 7 December 2013
Available online 14 December 2013

Keywords:
Circulating fluidized bed
Computational fluid dynamics
Flow regimes
Experiment
Hydrodynamics
In-depth system parameter

ABSTRACT

The transition flow between turbulent and fast fluidization regimes in high solid particle density circulating fluidized bed reactor was successfully investigated in this study. In addition, the effects of computational cell and simulation time were compared. Then, the comprehensive explanation and in-depth system parameters were discussed. The results showed the unique system characteristics. For the transition flow between turbulent and fast fluidization regimes, lower range of energy spectrum than the turbulent and fast fluidization regimes was found. As the increasing of gas inlet velocity, the axial and radial solid particle dispersion coefficients increased and decreased, respectively. For the axial and radial gas dispersion coefficients, both of them were increased with the increasing of gas inlet velocity. At the near wall region, the transition flow had no obvious power spectrum peak except the one below 0.2 Hz, while, at the center region, the maximum peak at 5.0 Hz was found. All the velocities had no significant correlation with solid volume fraction. The solid particle–solid particle interaction or collision then governs the system characteristic. The transition flow between turbulent and fast fluidization regimes will be an alternative choice for applying with circulating fluidized bed reactor in other new physical and chemical processes.

1. Introduction

Fluidized beds are a type of reactor mechanisms that can be used to carry out a variety of physical or chemical multiphase flow processes. Currently, there is much interest in the potential of this reactor for sorption or cleaning up greenhouse gases e.g. carbon dioxide (CO_2) from power generation to attenuate the greenhouse effect [1–4]. In this type of reactors, a fluid is passed through solid particles at high enough velocities to suspend the solid particles and causes them to behave as a fluid [5,6]. If the major fraction of solid particles leaving the reactor is captured by a solid separator and re-circulated back to the system, the process is called circulating fluidized bed. In this study, the focus is purely on gas–solid particle fluidization. Different gas–solid particle phenomena or fluidization regimes can occur when the gas inlet velocity passing through the solid particles is increased. At a very low velocity, the solid particles remain stationary on the bottom. The column hence is laid on a fixed bed operation. At an adequately high velocity, the fluidized bed operation starts and the solid particles transport out of the column. With changes in gas velocity, the solid particles move from one state or fluidization regime to another. These fluidization regimes, arranging in order of increasing velocities, are bubbling, turbulent, fast fluidization and pneumatic transport. Each fluidization regime has distinct characteristics. The occurrence of bubbles is the major characteristic of the bubbling fluidization regime [7]. In the turbulent fluidization regime, it is characterized by two different coexisting regions which are a dense bubbly flow at the lower region and dilute dispersed flow at the upper region [8,9]. In the fast fluidization regime, the flow structure can be characterized by dilute upwards of solid particles in the core or center region and downwards movement of aggregates or particle clusters in the annulus or wall region [10]. The main characteristic of the pneumatic transport fluidization regime is that all the solid particles will be carried up the column as individual dispersed solid particles [11]. However, the fluidization regime diagrams in the literature is major concerned with the low solid particle density or low flux system [12–14].

One of the emerging hydrodynamic problems in circulating fluidized beds is how to increase the solid volume fraction or solid holdup inside the reactor and to be more uniform in both axial and radial directions. The uniform distribution of the solid holdup is believed to have a positive effect on the reactor efficiency or chemical reaction conversion [2,15]. To date, only few research studies have been done to solve this problem. The high solid particle density or high flux system operation is one of the alternative choices to cure this problem. In the literature,

---

* Corresponding author at: Fuels Research Center, Department of Chemical Technology, Faculty of Science, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand. Tel.: +66 2218 7682; fax: +66 2255 5831.
E-mail address: benjapon.c@chula.ac.th (B. Chalermsinsuwan).

0032-5910/$ – see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.powtec.2013.12.018

most of the related research studies are focused on the finding of new methodology to operate this reactor in denser mode [15–17] and the measurement of denser general system hydrodynamics such as system pressure, velocity or solid volume fraction distributions [18–20]. However, the obtained system hydrodynamics are still unsatisfied [2, 15,21–24]. Zhu and Zhu [17] successfully operated the dense circulating fluidized bed reactor using secondary gas injection. Still, the new system flow structures or new operating methodologies with their explanation are needed with only primary gas injection.

Most of the research studies in the literature review are conducted using experimental method. However, there is another method called computational fluid dynamics simulation which is gaining attention. Computational fluid dynamics is an important tool for design and optimization of chemical processes. It is one of the fluid mechanic branches that uses numerical algorithms to analyze phenomena. The basic principle of the computational fluid dynamics is the calculation of mass, momentum and energy conservation equations, simultaneously. There are now many commercial and non-commercial computational fluid dynamics programs for example IIT code, MFIX and ANSYS FLUENT. For multiphase flow systems, two different approaches might be used for the calculation, namely the Lagrangian and the Eulerian approaches. The Lagrangian approach is limited by the overall number of solid particles in occupying space. For the circulating fluidized bed reactors, the Eulerian approach thus is suitable for the calculation. This approach separately solves the conservation equations for each phase. Among the various attempts to close the gas–solid particle flow, the kinetic theory of granular flow has found the widest use as a constitutive equation. This theory is basically an extension of the classical kinetic theory of gases, to dense gas–solid particle flows, with a description of the solid particle collisions by means of the restitution coefficients. However, the theory is ideally valid for solid particle velocity fluctuations having normal distributions. In real situation, strongly deviation from normal distribution may observe. Also, the assumption of no friction between solid particles is employed. These will be the numerical model limitation for using this numerical model. Although computational fluid dynamics has a promising future, there are currently no universal computational fluid dynamics models available [25–27].

In this research study, the transition flow pattern between turbulent and fast fluidization regimes inside high solid particle density circulating fluidized bed reactor was investigated using both experiment and computational fluid dynamics simulation. The computational fluid dynamics model thus could be validated its correctness and accuracy with the obtained experimental results. In addition, the comprehensive explanation and in-depth system parameters in high solid particle density circulating fluidized bed were discussed such as energy spectrums, power spectrums and dispersion coefficients.

2. Experimental apparatus and methodology

Experimental apparatus was performed in a cold-flow Plexiglas circulating fluidized bed reactor as displayed in Fig. 1. The three-dimensional circulating fluidized bed system with thin depth was used for clearly observing hydrodynamics phenomena inside the system. The riser, downer and separating and returning systems were three main parts of this circulating fluidized bed reactor. The riser or upflowing system part had 2.00 m height, 0.15 m width and 0.05 m depth. The downer or downflowing system part had 1.00 m height, 0.30 m width and 0.05 m depth. The separating and returning systems used 200 US standard mesh sizes to split the solid particles from the gas at the top of the reactor and at the inserted tubes in the downer. As the solid particles hit the separating system, they were fallen back to the returning system. Subsequently, the solid particles were fed to the riser at 0.05 m height from the bottom of the reactor through the controlling ball valve. At the base of the riser, a 200 US standard mesh size was used to support the solid particles. Then, a gas distributor was located directly below the mesh to disperse the gas, uniformly.

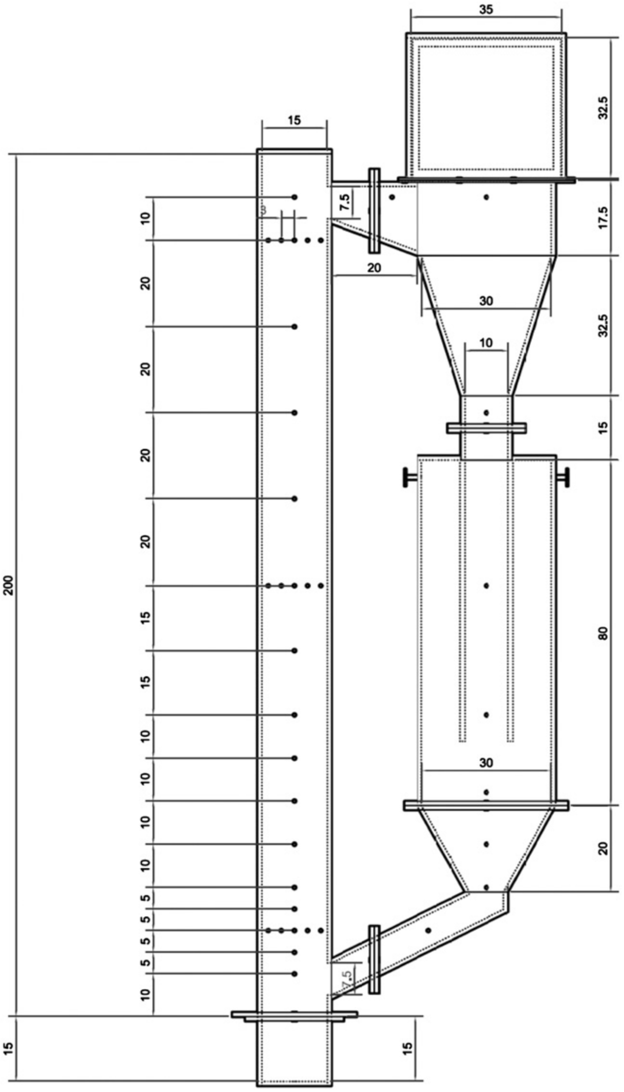

Fig. 1. Schematic drawing of circulating fluidized bed reactor.

To verify the computational fluid dynamics simulation results, the quantitative and qualitative experimental results were used for the comparison. Three gas inlet velocities were selected from a group of experiment results. These gas inlet velocities were corresponded to the transition flow between turbulent and fast fluidization regimes which

Table 1
The system operating conditions and modeling parameters used in this study.

Description	Value
Width of circulating fluidized bed riser (m)	0.15
Height of circulating fluidized bed riser (m)	2.00
Depth of circulating fluidized bed riser (m)	0.05
Width of circulating fluidized bed downer (m)	0.30
Height of circulating fluidized bed downer (m)	1.00
Depth of circulating fluidized bed downer (m)	0.05
Gas density (kg/m^3)	1.20
Gas viscosity (kg/m·s)	2×10^{-5}
Solid particle density (kg/m^3)	2650
Solid particle diameter (μm)	380
Gas inlet velocity (m/s)	0.75, 1.25 and 5.00
Initial solid particle inside the circulating fluidized bed reactor (kg)	21.00
Outlet system pressure (Pa)	101,325
Specularity coefficient (−)	0.01
Restitution coefficient between solid particle and wall (−)	0.90
Restitution coefficient between solid particles (−)	0.90