Journal of the Taiwan Institute of Chemical Engineers 000 (2016) 1–9

Contents lists available at ScienceDirect

Journal of the Taiwan Institute of Chemical Engineers

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

Effect of the jet direction of gas nozzle on the residence time distribution of solids in circulating fluidized bed risers

Hoanju Yoo[a], Hokyu Moon[b], Seungyeong Choi[c], Yong-Ki Park[d], Hyung Hee Cho[c,*]

[a] *Aircraft Research & Development Division, Korea Aerospace Industries LTD., 78, Gongdan 1-ro, Sanam-Myun, Sacheon, Gyeongnam 52529, Republic of Korea*
[b] *Tokamak Engineering Department, National Fusion Research Institute (NFRI), 169-168 Gwahak-ro, Yuseong-gu, Daejeon 34133, Republic of Korea*
[c] *Department of Mechanical Engineering, Yonsei University, 50 Yonsei-ro, Seodaemun-gu, Seoul 03722, Republic of Korea*
[d] *Green Chemistry Process Research Division, Korea Research Institute of Chemical Technology, Ganjeongro 141, Daejeon 34114, Republic of Korea*

ARTICLE INFO

Article history:
Received 29 July 2016
Revised 27 October 2016
Accepted 17 December 2016
Available online xxx

***Keywords*:**
Circulating fluidized bed
Gas nozzle
Jet direction
Solid residence time distribution
Numerical simulation

ABSTRACT

The authors investigate the effects of the direction of the gas jet on the solid residence time distribution in a CFB riser. Tracer technique was employed to calculate the RTD of solids. A Eulerian–Eulerian model with kinetic theory of granular flow and species transport was used to simulate the motion of tracer particles in a CFB riser. For a comparative analysis of the direction of the gas jet, simulations of vertical, horizontal and hybrid jets were carried out. The direction of the gas jet significantly influenced the axial and radial structure of bed, and hence affected the RTD for solid particles. The mean residence time of solids was changed, and the results showed that 16.3 s, 14.8 s, and 11.4 s with vertical, horizontal and hybrid jets nozzles, respectively.

1. Introduction

Circulating fluidized beds (CFBs) are widely used in industrial applications including petrochemicals, power generation, and environment-related processes, because of the favorable heat and mass transfer and excellent gas–solid contact efficiency [1–5]. Various chemical reactions may occur in CFB reactors, and the operating conditions are primarily determined by the nature of these chemical reactions. The chemical reactions in CFB reactors depend on the reaction rate and residence time. The reaction rate is function of the chemistry and thermodynamics of the system; however, the residence time is determined mainly by the hydrodynamics of the gas–solid flows in the CFB reactor [6].

In general, chemical reactions in CFB reactors can be divided into two groups: catalytic gas-phase reactions and gas–solid reactions. With catalytic gas-phase reactions, the residence time of solids is relatively short, as shown in Fig. 1 [7,8]. The solid particles act as catalysts or heat transfer media, and the target/desired products are mostly gas phase and organic chemicals. Catalytic gas-phase reactions require short contact times (of the order of a few seconds), and solid backmixing results in inefficient gas–solid contact because of the rapid deactivation of the catalyst particles. For these reasons, CFB reactors for catalytic gas-phase reactions typically operate at high superficial gas velocities and solid feed rates. Gas–solid reactions are typically employed for generating heat or solid-phase products, and have slower reaction rates. The solid residence time for gas–solid reactions should therefore be longer, of the order of seconds to minutes (see Fig 1). Solid backmixing results in long residence times with a uniform temperature distribution to prevent hot spots from forming. Owing to the different characteristics of chemical reactions, an understanding of the solid residence time and backmixing is required in the design or development of the conversion processes using CFB reactors.

Injection of gas *via* nozzles is frequently used for highly reactive or high-temperature applications. There have been reports that the geometry of gas nozzles significantly influences the gas–solid structures in CFB risers. For example, Cheng et al. [9] investigated the arrangement of the open area of the main gas nozzle and found that the geometry of the gas affects primarily the lower part of the riser. Peng et al. [10] varied the location of the gas nozzles to increase the uniformity of the radial distribution of solids in a CFB riser. Chalermsinsuwan et al. [11] studied the effects of the gas inlet channel size on the solid hydrodynamics in high-flux solid CFB risers. Cheng et al. [9] and Peng et al. [10] have researched the effect of gas nozzle in the CFB riser through 2-dimensional simulation, which although computational cost was low, there were limitation to quantitative comparison with actual cases. Generally, 2-dimensional simulation is only recommended for qualitative

* Corresponding author.
E-mail addresses: hhcho3@gmail.com, hhcho@yonsei.ac.kr (H.H. Cho).

http://dx.doi.org/10.1016/j.jtice.2016.12.018
1876-1070/

Nomenclature

Symbols

C	concentration of tracer particles
d_p	mean diameter of particles [m]
g	acceleration due to gravity [m/s^2]
g_0	radial distribution coefficient
G_S	circulation rate of solids [kg/m^{2}s]
H	enthalpy
r/R	dimensionless radial coordinate
p	pressure [Pa]
Re_S	Reynolds number of particles
t	time [s]
T	temperature [K]
$\vec{v}$	velocity vector [m/s]
U_s	superficial velocity [m/s]
Y_{tracer}	mass fraction of tracer particles
θ_s	granular temperature
α	volume fraction
σ	standard deviation
ρ	density [kg/m^3]
μ	viscosity [Pa s]
$\overline{\overline{\tau_s}}$	stress tensor
K_{gs}	interchange momentum exchange coefficient

evaluation and 3-dimensional modeling is recommended for predictive simulations [12]. Furthermore, above-mentioned investigations only focused on the flow structure in gas–solid CFB risers which was affected by the gas nozzle, but they did not consider the residence time of the solid particles. Because the residence time is very important in chemical reaction, this study is conducted to understand of the effect of gas nozzles on the residence time in the CFB riser which is essential part of the design process of CFB. Here the authors investigate the effect of gas nozzles on the residence time of solids using computational fluid dynamics (CFD). The method was validated using the available experimental data. The authors then varied the direction of the jets of fluidizing gas, and carried out a comparative analysis. The authors determined the effect of the direction of the gas nozzles on the residence time of the solids, which can aid in the design of a CFB reactor, and the choice of jets depends on whether a catalytic gas phase or gas–solid reaction is employed.

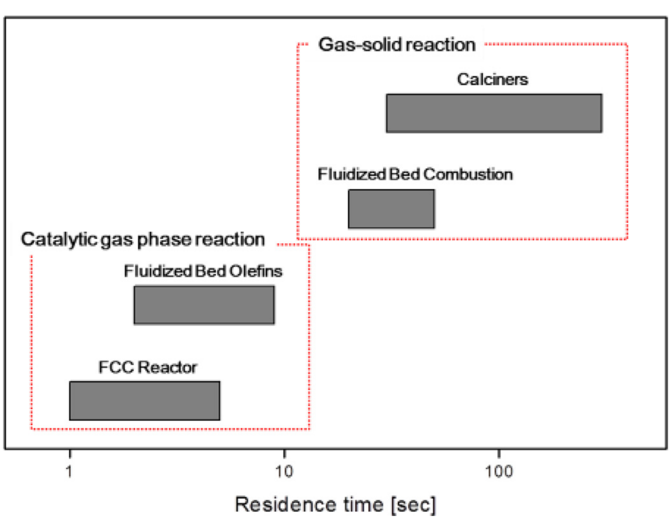

Fig. 1. Typical residence time of solids in catalytic gas phase reactions and gas–solid reactions [1,2].

2. Research method

The objective of this work is to investigate the influence of the directions of the gas jets on the residence time distribution of solids. The tracer technique was used to calculate the residence time distribution (RTD) of the solid. A comparative study using experimental data was carried out to validate the numerical method, and subsequently the authors determined the distribution of the solid RTD for various jet directions of the fluidizing gas into the CFB riser.

2.1. Numerical simulation

For multi-phase flow analysis, there are two approaches; Eulerian–Eulerian approach and Eulerian–Lagrangian approach. In Eulerian–Eulerian approach, gas and solid phase are treated mathematically as interpenetrating continua. To describe the motion of solid, the kinetic theory of granular flow (KTGF) is incorporated in an Eulerian framework. The advantage of this approach is that computational cost is lower. However, the disadvantage is that it is difficult to simulate the discrete characteristics like size and shape of the solid particles [13]. In Eulerian–Lagrangian approach, solid particle phase is treated as discrete phase. The movement, translation and rotation of every solid particles is descried by Newton's equation. However, numerical simulation based on Lagrangian approach is that computational cost is tremendously expensive, as the number of solid particles increases [14].

In this study, an unsteady simulation was used to simulate the gas–solid multiphase flow in the CFB riser using an Eulerian–Eulerian approach with kinetic theory of granular flow implemented using ANSYS Fluent version 14.0. A fixed time step of 5×10^{-4} s was used. The model treats gas and solid phases as interpenetrating continua, which means that the presence of two phases is possible in a single control volume by introducing fractional volume variables. Mass, momentum and species were conserved individually for each phase. Details of governing equations are listed in Table 1 [15]. The CFB riser operates with a turbulent flow field, so that the Eulerian–Eulerian model with turbulence is more accurate than a laminar model [16]. In this study, $k-\varepsilon$dispersed turbulence model, which is appropriate for a dilute secondary phase [15,16].

The solid phase was assumed to be a continuum, so that constitutive correlations were required for conservation of momentum; these were derived from the kinetic theory of granular flow [13]. The granular temperature was introduced, which is proportional to the kinetic energy associated with the random motion of the particles. The pressure and viscosity of the solid were described as a function of the granular temperature to close conservation of momentum. The pressure of the solid is equal to the normal forces on the solid phase that arise from particle–particle collisions. The coefficient of restitution for particle collisions was set $\varepsilon_{ss}=0.9$. A radial distribution function was used to describe the probability of particle collisions, which prevents the maximum packing limit $\alpha_{\max}$ from being exceeded. The solid shear viscosity is given by the sum of the collisional, kinetic and frictional components, and describes the tangential forces that result from translation and collision of particles. The solid bulk viscosity represents the resistance of the particles to compression and expansion. Table 2 lists the constitutive correlations.

Forces due to drag acting on the particles in the gas–solid flow are represented by the final term on the right-hand side of the momentum conservation relation. It is composed of the product of the interphase momentum exchange coefficient, K_{gs} and the slip velocity, $\vec{v}_g-\vec{v}_s$. The Gibilaro drag model was used, which has been shown to accurately describe the gas–solid flow characteristics in fluidized beds under turbulent flow conditions [17,18].

Please cite this article as: H. Yoo et al., Effect of the jet direction of gas nozzle on the residence time distribution of solids in circulating fluidized bed risers, Journal of the Taiwan Institute of Chemical Engineers (2016), http://dx.doi.org/10.1016/j.jtice.2016.12.018