Powder Technology 327 (2018) 17–28

Contents lists available at ScienceDirect

Powder Technology

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

The influence of pressure and temperature on gas-solid hydrodynamics for Geldart B particles in a high-density CFB riser

Yawen Zhang [a,b,c], Fulin Lei [b,c,*], Yunhan Xiao [b,c]

[a] Collaborative Innovation Center of Chemical Science and Engineering (Tianjin), School of Chemical Engineering and Technology, Tianjin University, Tianjin 300072, China
[b] Key Laboratory of Advanced Energy and Power, Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China
[c] University of Chinese Academy of Sciences, Beijing 100049, China

ARTICLE INFO

Article history:
Received 3 August 2017
Received in revised form 29 October 2017
Accepted 9 December 2017
Available online 12 December 2017

Keywords:
High-density CFB
Two-fluid model
EMMS
Pressure effect
Temperature effect

ABSTRACT

The effects of operating pressure and temperature on the flow behavior in the riser of a high-density circulating fluidized bed have been numerically studied using the two-fluid model coupled with the EMMS-based drag model. When the operating pressure is below 0.4 MPa, the flow regime in the riser experiences the fluid dominated (FD) zone, the transition zone from FD to the particle-fluid compromising (PFC), and the PFC zone with the increase of solids circulating flux. Under the same solids circulating flux, the solids axial velocity increases while the solids volume fraction decreases with the increase of operating pressure. In terms of axial profiles of solids volume fraction and axial velocity, there exists a critical operating pressure, above which the axial profile changes little with further increase of operating pressure, while it is hard to find a critical pressure in terms of radial profiles. When the gas density is kept the same, the increase of gas viscosity caused by higher temperature has only minor effect on the axial profiles of solids volume fraction, but it makes the solids axial velocity increase in the core zone and decrease slightly near the wall.

1. Introduction

Circulating fluidized bed (CFB) reactors operated under fast fluidization regime have been widely used in coal combustion and gasification process [1,2]. A large number of high-density circulating fluidized bed (HDCFB) reactors operated with high superficial gas velocity and high solids circulating flux are employed in the field of Fluid Catalytic Cracking (FCC) [3,4]. Based on the work of Grace et al. [5], the HDCFB's boundaries for Geldart A particles should satisfy that the solids flux is >200 kg/m^2 s and the solids volume fraction is >0.1 in the riser. The flow regime in the riser of HDCFB is characterized as the dense suspension upflow (DSU) [2,6]. Zhou et al. [7] investigated flow behavior under ambient condition in the riser of HDCFB with the EMMS-based two-fluid model, and they mainly validated the predicted results with experimental data. Wang et al. [8] also performed a numerical study of flow behavior in the riser of HDCFB under ambient condition. The pressurized gasification technology has great potential in improving the gasification efficiency and the scale-up of the gasifier [9–11]. High-density CFB reactors have been employed in the pressurized gasification [12,13], and these reactors are operated under elevated pressure and temperature. The reaction behavior is highly dependent on gas-solid flow hydrodynamics in the riser, thus it is important to understand the influence mechanism of pressure and temperature on the gas-solid flow behavior.

Yin et al. [14] carried out an experimental study to investigate the effect of operating pressure on the flow behavior in a HDCFB, where the operating pressure was up to 0.5 MPa. Richtberg et al. [15] studied the effect of the solid/gas density ratio on the flow pattern in a pressurized circulating fluidized bed. The CFB system was built inside a high-pressure vessel, and the maximum operating pressure could reach 5.0 MPa. These studies reveal that it is difficult to build a pressurized CFB system in a laboratory, and it is challenging to fully characterize the gas-solid flow behavior with the experimental methods. To avoid these problems in the experimental study, several researchers adopted dimensional analysis method to study the flow behavior in a pressurized CFB, considering that the effect of operating pressure can be attributed to gas density. Louge et al. [16] employed the dimensional analysis method to simulate the pressurized condition with different gases in an experimental system operated under ambient pressure and temperature. Based on previous study, Brieout and Louge [17] further validated the dimensional analysis method in investigating the hydrodynamics under pressurized conditions.

The effect of temperature on gas-solid hydrodynamics was mainly reported on spouted bed. Ye et al. [18] investigated the hydrodynamics parameters such as the minimum spouted velocity for different sizes of silica sand in a spouted bed under the operating temperatures up to 880 °C. Wu et al. [19] investigated the flow behavior in a spouted bed

* Corresponding author at: Institute of Engineering Thermophysics, Chinese Academy of Sciences, No. 11 North Fourth Ring Road West, Haidian District, Beijing 100190, China.
E-mail address: leifulin@iet.cn (F. Lei).

https://doi.org/10.1016/j.powtec.2017.12.040
0032-5910/

Nomenclature

Symbols

C_{D0}	standard drag coefficient for a particle
d_p	particle diameter, m
e	particle-particle restitution coefficient
$\mathbf{g}$	gravitational acceleration, m/s^2
G_s	solids circulating flux, kg/m^{2}s
H_D	heterogeneity index
p	pressure, Pa
Re	Reynolds number
T	Temperature K
$\mathbf{u}_{silp}$	real slip velocity, m/s
$\mathbf{v}$	velocity, m/s

Greek letters

β	drag coefficient
ε	volume fraction
μ	viscosity, kg/(m s)
ρ	density, kg/m^3

Subscripts

g	gas phase
p	particle
s	solid phase

under elevated temperature, and three different gases including air, helium and methane were adopted to study the effects of gas density and gas viscosity separately. However, it is difficult to separate the effects of gas density and gas viscosity with experimental method, thus the numerical method can play an important role in the study of these effects separately.

Computational fluid dynamics (CFD) method has been widely used for the modeling of the gas-solid flow in the riser of CFB under ambient temperature and pressure [20–22]. And these reported studies focus on developing the numerical methods and validating them with experimental data. Recently, Yin et al. [23] employed the Eulerian-Lagrangian method to simulate the gas-solid flow behavior in the riser of a pressurized high-flux CFB. They mainly performed parametric study of important model parameters on the flow behavior, the effect of operating pressure was not investigated further. Zhong et al. [24] developed the CFD method to investigate hydrodynamic characteristics of gas-solid flow in a spouted bed under elevated pressure and temperature. Recently, they studied the effect of pressure on gas-solid flow behavior in a spouted bed using the developed numerical method [25]. These studies reveal that the CFD method is useful to predict the gas-solid flow behavior under ambient conditions, elevated pressures, and high temperatures.

In this work, the two-fluid model coupled with the EMMS-based drag model is employed to predict the flow behavior in the riser of a HDCFB under high operating pressure and temperature. Firstly, the experimental cases performed on the HDCFB are selected to validate the CFD method. Secondly, the effect of operating pressure on the flow behavior is studied using the CFD method, and the effect of pressure is attributed to the variation of gas density. Thirdly, the effect of operating temperature on the gas-solid behavior is studied with the CFD method, and we mainly focus on the effect of gas viscosity by proper combination of the operating pressure and temperature.

2. Numerical method

The test rig of pressurized dense transport bed gasifier was established in our laboratory, and the gas-solid flow experiments with silica sand of Geldart group B under elevated pressure and temperature have been performed. The schematic diagrams for the experimental system and

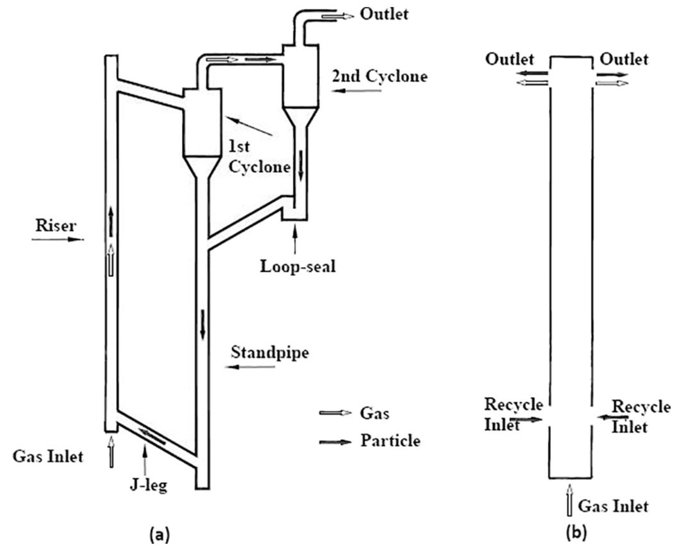

Fig. 1. Schematic diagram of (a) full-loop system and (b) two-dimensional riser of the HDCFB.

the computational geometry of the riser are shown in Fig. 1. The height of the riser is 18.3 m, and its diameter is 0.1 m. The 2D simplification of the riser geometry has been used in many studies [26,27], because it can give correctly the main gas-solid flow characteristics. In our previous work, a two-dimensional (2D) model has been validated reasonable in predicting the flow behavior under ambient condition with two-fluid model coupled with the EMMS-based drag model [22]. The comparisons between Gidaspow drag model and EMMS-based drag model have been performed, and it was found that the EMMS-based drag model could predict better axial and radial profiles. The comparison between 2D and 3D model indicated that 2D model could also predict well axial and radial profiles in the riser. In this study, two sets of gas-solid flow experiments under elevated pressure and temperature are chosen to further validate the numerical method. The riser temperature is assumed to be constant, considering that the temperature difference between combustion zone and gasification zone is lower than 100 °C due to large solids flux and excellent heat transfer characteristics in the riser. Furthermore, the flow behavior in the riser under high pressure and temperature will be studied by the variations of gas density and viscosity.

The air density is determined by the ideal gas equation,

$$\rho_g = 1.225\frac{p}{p_0}\frac{300}{T}\ \text{kg/m}^3 \tag{1}$$

The air viscosity is calculated by Sutherland's law,

$$\mu_g = 1.72\times10^{-5}\frac{273+111}{T+111}\left(\frac{T}{273}\right)^{1.5}\ \text{Pa}\cdot\text{s} \tag{2}$$

Table 1
Operating conditions and physical parameters used in the simulation of experimental cases.

Case	1	2	3
Superficial gas velocity (m/s)	8.6	8.0	4
Operating pressure (MPa)	0.1	0.9	0.22
Temperature (K)	300	300	363
Pressure drop (kPa)	33	23	34
Particle diameter (μm)	89	154	154
Particle density (kg/m^3)	2951	2620	2620
Maximum packing limit	0.57	0.6	0.6
Initial solids volume fraction	0.048	0.034	0.052