ARTICLE IN PRESS
PTEC-12850; No of Pages 14
Powder Technology xxx (2017) xxx–xxx
Contents lists available at ScienceDirect
Powder Technology
journal homepage: www.elsevier.com/locate/powtec

CPFD study of a full-loop three-dimensional pilot-scale circulating fluidized bed based on EMMS drag model

Qiuya Tu [a], Haigang Wang [a,b,*]

[a] Institute of Engineering Thermophysics, Chinese Academy of Sciences, PO Box 2706, Beijing 100190, China
[b] University of Chinese Academy of Sciences, Beijing 100190, China

ARTICLE INFO

Article history:
Received 25 January 2017
Received in revised form 13 September 2017
Accepted 24 September 2017
Available online xxxx

Keywords:
Circulating fluidized bed
CPFD
EMMS
Fluidization regime
Power spectrum
Particle volume fraction

ABSTRACT

For the purpose of process optimization and scale-up, coupling energy-minimization multi-scale (EMMS) drag model with computational particle fluid dynamics (CPFD) is a good option for large scale three dimensional full-loop circulating fluidized bed (CFB) simulation. In this study, such integration is applied for a complex CFB with six cyclones, based on Barracuda platform, to investigate the gas-solids flow dynamic characteristics in scaled-up system, especially in the dense flow region, and to characterize different fluidization regimes. The simulation results are compared with those obtained from Wen-Yu model, in terms of axial pressure distribution, solid volume fraction, particle recirculation, etc., and are evaluated against available measurement data. The result demonstrates that EMMS drag model can predict the axial pressure distribution well, especially in the dense region, which gives an S-shape solid volume fraction in the axial direction and two-core-annulus structures ('M' shape) in the radial direction. Meanwhile, EMMS drag model provides smaller drag force between the gas phase and the solids that the particle recirculation fluxes inside and outside the riser is relatively smaller. With the increase of superficial gas velocity, three different fluidization regimes, including multiple bubble regime, exploding bubble regime and turbulent fluidization regime, are identified based on EMMS drag model. The results show that the dense region is clear at the multiple and exploding bubble regimes; while the particle volume fraction and mass flow rate in the riser top are larger than those in the middle at the turbulent fluidization regime, which gives a guidance that the riser height is limited that particles do not have enough space to distribute.

1. Introduction

Due to its good gas-solid mixing and excellent mass and heat transfer ability, circulating fluidized bed (CFB) has been widely used in the industry for coal combustion, gasification, and fluid catalytic cracking (FCC) etc. [1]. But also due to its complex gas solid flow characteristics and difficulty to quantify, the gas-solids flow in CFB has been paid lots of attention and extensively investigated by experiments and numerical simulation [2–6].

Experiments can provide macro and local flow information, but is generally time- and cost-consuming, especially for large scale facilities. With the increase of computational power and improvement in modeling methods, numerical simulation, e.g. computational fluid dynamics (CFD), enabling more detailed hydrodynamics description, is emerging as a popular tool. In CFD simulation, two approaches, Eulerian-Eulerian and Eulerian-Lagrangian, are commonly used, but these two are usually limited to small scale CFBs with low density [3,7–10]. For large scale or industrial scale CFBs with high density, an alternative new approach, the computational particle fluid dynamics (CPFD) numerical scheme, developed by Snider [11], can be used.

CPFD is an Eulerian-Lagrangian model for gas-solid flows, and is considered as a supplement to the Eulerian-Eulerian and conventional Eulerian-Lagrangian methods, which employs multi-phase particle-in-cell (MP-PIC) [12–14] methodology and numerical-particle concept to handle and track a great amount of particles. Comparatively, CPFD requires less computational resources [15], and has become as a feasible and efficient approach for CFB riser simulation [15–17], especially the one in industrial scale. However, the optimal CFB simulation should be with the full loop, which is considered as the most accurate and valuable way to fluidized bed design and operation [4,18]. Currently, due to the complexities in geometry and flow physics, only a few investigations have focused on the full loop simulation, which requires large computational resources [19–22], while CPFD can finish it effectively in three dimensions [23–25].

In the gas-particle two-phase flow of a CFB, drag dominates the particle transport and internal recirculation in the riser, so numerical accuracy of CFB simulation is sensitive to the drag force [26]. The most commonly used drag models are conventional drag models, which are developed on the basis of homogeneous systems, without

* Corresponding author at: Institute of Engineering Thermophysics, Chinese Academy of Sciences, PO Box 2706, Beijing 100190, China.
E-mail address: wanghaigang@iet.cn (H. Wang).

https://doi.org/10.1016/j.powtec.2017.09.045
0032-5910/

considering the effects of meso-scale structures [27] in the riser, thus the heterogeneous structures cannot be predicted accurately. Currently, two models begin to appear which can resolve the heterogeneity in the CFB, one is based on filtered two-fluid models (TFMs) with constitutive relations for filtered fluid-particle drag coefficient and filtered stresses [28], the other is the energy minimization multi-scale (EMMS) drag model [29,30]. Introduced by Li and Kwauk [31], EMMS scheme has been developed into many improved versions by taking into account the effect of heterogeneous characteristics on the momentum exchange coefficient. Generally, EMMS drag models are coupled with two-fluid model (TFM), and used in the investigation of small/pilot scale CFB risers [3,26,32,33], only a few are used for the large scale full-loop setup [4,5,22], due to demanding computing source requirements. Comparatively, CPFD is efficient and affordable. Therefore, coupling EMMS model with CPFD is a promising method for three dimensional full loop CFB simulations.

For the purpose of process optimization and scale-up, it is necessary to simulate the flow behavior in large scale CFBs with high density and high flux. Scale-up of fluidized beds is difficult due to the significant hydrodynamics differences between small scale and large scale fluidized beds, like the bubble behavior change, flow regime change, etc. These differences can lead to process yields that can be inferior to results obtained in smaller units [34]. Successful scale-up of CFBs typically proceeds through three to four phases or steps, from the lab scale (bed diameter ~ 25 mm), pilot-plant scale (bed diameter ~ 200–600 mm), to demonstration plant scale (bed diameter ~ 800–1500 mm) and commercial scale (bed diameter ~ 1–16 m) [34]. According to the effect of bed diameter on scale-up parameters [34], at certain point, the scale-up parameters approach to constant values, then, the corresponding model can be used to estimate the behavior of larger models.

In this study, a complex full-loop CFB system, with a rectangular riser and six cyclone separators, scaled down from a 600 MWe supercritical pressure CFB following the scaling rules proposed by Glicksman, Hyre and Woloshun [54], was simulated by CPFD Barracuda platform coupling with EMMS/matrix drag model, to investigate the gas-solids flow dynamics in the scaled-up system, especially in the riser dense region, and to characterize different fluidization regimes. Firstly, at one condition, the simulation results were compared with Wen-Yu based drag model simulation, in terms of pressure profiles, solids distribution, non-uniform distribution of solid fluxes inside and outside the riser, etc., and evaluated against available experimental data. Then the system is investigated at three different fluidization regimes to characterize the flow dynamics.

2. Experimental set up

Experimental measurements were conducted in a cold CFB facility, as mentioned before, it is scaled down from an industrial-scale CFB model, the scheme is shown in Fig. 1. The main dimensions and parameters are presented in details by Jiang et al. [23].

In the facility, silica sand is used as bed material, whose actual density is 2620 kg/m^3, and bulk density is 1703 kg/m^3. Its particle size distribution is shown in Fig. 2, covering the range from 200 μm to 1002 μm, with average diameter of 475.43 μm.

Along the riser, three regions' (i.e. the dense region, middle region and dilute region) pressure drops are measured by the differential pressure transducers and collected by Agilent data acquisition unit, and at each cross section, six pressure transducers are located around. When the system reaches steady state, the particle accumulation height in each dipleg is measured after the particle fluidization air at the bottom of the corresponding U-loop is switched off for a certain period of time, then the particle recirculation flux is calculated by

$$G_p = \rho_b \cdot \frac{\pi d_0^2}{4} \cdot \Delta h / \Delta t \tag{1}$$

where ρ_b is the particle bulk density, d_0 is the dipleg diameter, Δh is the particle accumulation height in the dipleg, Δt is the switching off period of particle fluidization air at the bottom of U-loop.

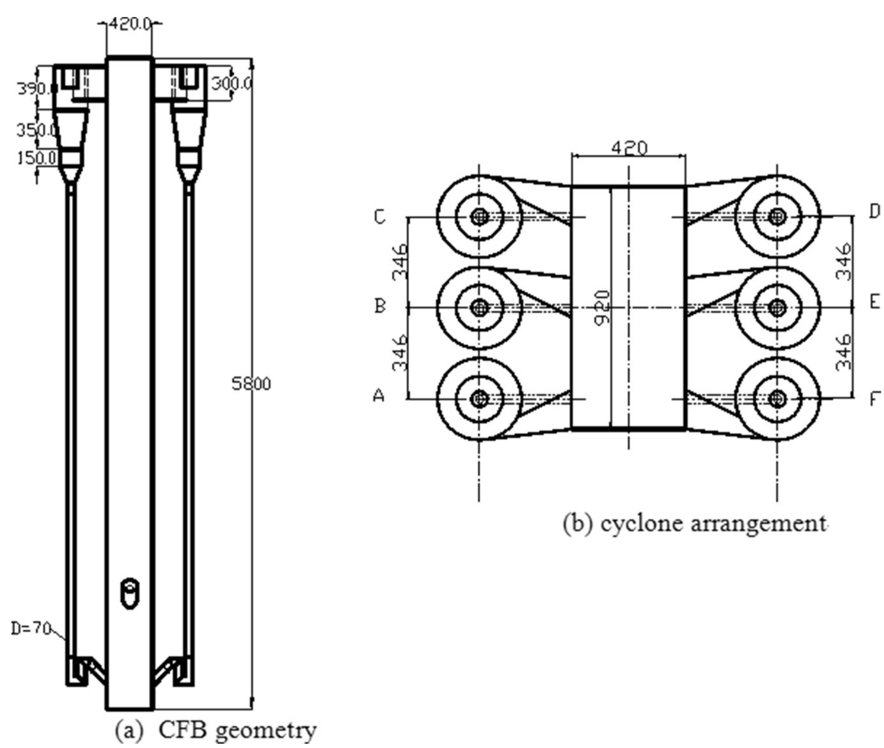

Fig. 1. CFB schematic (mm). (a) CFB geometry, (b) Cyclone arrangement.

Please cite this article as: Q. Tu, H. Wang, CPFD study of a full-loop three-dimensional pilot-scale circulating fluidized bed based on EMMS drag model, Powder Technol. (2017), https://doi.org/10.1016/j.powtec.2017.09.045