Chemical Engineering Science 109 (2014) 85–97

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Modelling and experimental investigation of the full-loop gas–solid flow in a circulating fluidized bed with six cyclone separators

Yu Jiang [a,b], Guizhi Qiu [a,b], Haigang Wang [a,*]

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

HIGHLIGHTS

- CPFD model is used to simulation the full-loop of a circulating fluidized bed with six cyclone separators.
- Simulation results are compared with Electrical capacitance tomography and pressure measurement.
- The CPFD model combined with ECT technology provides a possibility to optimize the design for large scale circulating fluidized bed.

ARTICLE INFO

Article history:
Received 22 November 2013
Received in revised form
15 January 2014
Accepted 25 January 2014
Available online 31 January 2014

Keywords:
Circulating fluidized bed
CPFD
Cyclone Separator
Full-loop
Process tomography

ABSTRACT

In the literature, there are few reports on the full-loop gas–solid flow in a circulating fluidized bed (CFB) with large scale and complex cyclone arrangement. In this paper, a new approach based on computational particle fluid dynamic (CPFD) method combined with electrical capacitance tomography (ECT) is used to investigate the hydrodynamic behavior of gas–solid flow in a CFB with six cyclone separators in order to improve the design and performance of a large scale CFB boiler. The full-loop CFB system for the simulation includes the CFB riser, cyclone, standpipe and U-loop. Two types of cyclone arrangement, i.e. axis and point based symmetric arrangement, are used for the CPFD simulation and ECT measurements. To validate the CPFD simulation, ECT is applied to measure the solids concentration in the standpipe with eight electrodes mounted on the outside of the standpipe. Key parameters including pressure, solids recirculation flux and velocity profile along different positions based on the CPFD simulation are analyzed and compared with experimental results. The CPFD simulation shows that the gas–solid flow is non-uniform among the six parallel cyclones. The solids concentration of four cyclones at the corner of the riser is higher than that of the others. The location of cyclone as well as the inlet angle of the cyclone needs to be optimized. The study shows that the presented approach based on CPFD simulation and ECT measurements can be used to optimize the arrangement of cyclone separators in a supercritical pressure circulating fluidized bed system.

1. Introduction

Circulating fluidized bed (CFB) is one kind of clean coal combustion technologies which plays an important role in the power generation and coal gasification industry (Reh, 2003). Large thermal capacity and high steam pressure is a tendency for the development of CFB boilers (Lv et al., 2007, Fan et al., 2008, Chen et al., 2008). To meet the demands for high steam parameter and large thermal capacity, high efficiency of gas–solid separation is a key to achieving high combustion efficiency, reducing limestone consumption and NO*x* emission (Koornneef et al., 2007).

With the scaling up of a CFB boiler, the dimension of the cyclone is increased accordingly and the separation efficiency decreases due to a reduction in the centrifugal force. To overcome the above issue, a large cyclone is replaced by numbers of smaller cyclone with the increase of boiler size to reduce the cyclone size. Different arrangement of cyclones on the top of the CFB riser is provided and patented (Hack et al., 2008). Experimental research has been carried out and methods related with cyclone arrangement have been patented (Armistead et al., 2002, Lv et al., 2007, Zhou et al., 2012). However, there is a non-uniform solids mass flux distribution among cyclones with a maximum difference of 17% (Morin, 2003, Chen et al., 2008, Zhou et al., 2012). For a CFB boiler with multi-cyclone separators, it is important to investigate the gas–solid flow in the whole loop including the CFB riser, cyclones as well as standpipe and U-loop.

* Corresponding author. Tel.: +0086 10 8254 3140.
E-mail address: wanghaig@hotmail.com (H. Wang).

http://dx.doi.org/10.1016/j.ces.2014.01.029
0009-2509

Computational fluid Dynamics (CFD) simulation provides detailed information for the investigation of fluidization characteristics with large-scale CFB boilers (Reh, 2003). Research has been reported using the CFD approach to investigate the three dimensions gas–solid flow in a CFB boiler (Zhang et al., 2008, 2010, Ahuja and Patwardhan, 2008, Hartge et al., 2009). In dealing with gas–solid fluidized beds, two approaches, i.e. the Eulerian-Eulerian two-fluid model (TFM) (Gidaspow and Ettehadieh, 1983, Lun et al., 1984) and the Eulerian-Lagrangian discrete particle method (DPM), are commonly used. The TFM model treats solids as a continuous phase which interacts with the gas phase by momentum exchange. Conservation equations for each phase have similar terms and are solved together with a set of constitutive equation derived by experiment. The TFM model has been widely used in multiphase flows simulation. However, it has limitations, such as not applicable to particle size distribution and inter-particle forces (Makkawi et al., 2006). The DPM method describes the discrete phase by tracking numerous particles trajectories which exchange mass, momentum and energy with the gas phase all through the whole simulation field. The DPM model takes into account the particle size distribution as well as particle–particle interactions. However, it is difficult to simulate dense gas–solid flow with solids volume fraction above 5% due to large amount of particle numbers. In general, the particle number is under the order of 2×10^5 in the DPM model and it is often applied to two-dimensional simulation. Recently, an Eulerian–Lagrangian model called CPFD (computational particle fluid dynamics) has been used to model the gas–solid flow in a fluidized bed (Abbasi et al., 2011, Chen et al., 2013). This methodology incorporates the multi-phase-particle-in-cell (MP-PIC) method for calculating a dense gas–solid flow (Andrews and O'Rourke, 1996, Snider, 2001). In the CPFD approach, the gas phase is modeled as a continuous fluid and particles as a discrete phase which can handle particle size distribution. Particles are classed into numerous of computational parcels. Each parcel represents a number of physical particles which have a same velocity and material property in the computational domain. With this scheme, billions of particles can be simulated much more efficiently.

To validate the CFD simulation results in a CFB boiler, it is necessary to verify the results with measurements. Electrical capacitance tomography (ECT) provides an option to investigate the gas–solid flow in a fluidized bed due to its no-intrusive and no-invasive nature (Dyakowski et al., 1997, 1999 , Makkawi and Wright, 2004, 2006, Du et al., 2005, Wang et al., 2006).

To understand the hydrodynamic behavior of a gas–solid circulating fluidized bed, a “cold” CFB test facility with six cyclone separators in the top of the riser and rectangular shape combustion chamber has been built in the Institute of Engineering Thermophysics, Chinese Academy of Sciences. CPFD is used to simulate the whole circulating loop. CPFD simulation results are compared with experimental results by ECT and pressure measurements. Two different arrangements of cyclones are used to compare the gas–solid fluid hydrodynamic behavior in the multi-cyclone CFB system. The objective of the research is to evaluate the applicability of CPFD method for the gas–solid flow simulation in a whole-scale CFB boiler and compare with experiment results. The CPFD simulation is based on a commercial code BARRACUDA. ECT was used to measure the solids concentration in the cross sections of the standpipe with eight electrodes mounted outside of the pipe and to validate the CPFD simulation results.

2. CPFD mathematical model

2.1. Governing equations

The CPFD methodology takes an Eulerian-Lagrangian approach to describe the gas–solid flow in three dimensions. The gas phase is described as a continuous phase with strong coupling to the discrete solids phase in mass and momentum equations. As the gas and solids phases are isothermal and the gas phase is incompressible, no volume averaged fluid energy equations are needed. In the CPFD scheme, a concept of numerical particle is introduced, which is a numerical approximation similar to the numerical control volume within which the fluid has a common property. The solids phase is modeled as numerical parcels each containing quite a number of physical particles with same properties (species, density, size, etc) in the same location. The flow fields of gas and solids phase are calculated by separated governing equations. For the gas phase, the governing equations are

$$\frac{\partial}{\partial t}(\rho_g\theta_g)+\nabla(\rho_g\theta_g v_g)=S_g \tag{1}$$

$$\frac{\partial}{\partial t}(\rho_g\theta_g v_g)+\nabla(\rho_g\theta_g v_g v_g)=-\nabla P+\nabla\theta_g\tau_g+\rho_g\theta_g g-F \tag{2}$$

where θ_g represents the volume fraction of gas, ρ_g and v_g stands for density and velocity of gas respectively, $S_g$is a source term, τ_g represents the gas stress tensor, p stands for the pressure of gas, g is the acceleration of gravity, F is the rate of momentum exchange per volume between the gas and solids phases. The momentum equation presented here neglects the viscous molecular diffusion in the fluid but retains the viscous drag between particles and fluid through an interphase drag force, F, which is

$$F=\iint fm\left(D_p(v_g-v_p)-\frac{\nabla P}{\rho_p}\right)dmdv \tag{3}$$

where D_p is the drag function, v_p and ρ_p represents particle velocity and density respectively, f is the probability distribution function which is calculated from Liouville equation as

$$\frac{\partial f}{\partial t}+\nabla(fv_p)+\nabla_{v_p}\left(f\frac{d}{dt}(v_p)\right)=0 \tag{4}$$

where $\frac{d}{dt}(v_p)$ is the particle acceleration, which is obtained by calculating all forces on the particles and is given by MP-PIC method (Andrews and O'Rourke, 1996, Snider, 2001) as following:

$$\frac{d}{dt}(v_p)=D_p(v_g-v_p)-\frac{\nabla P}{\rho_p}-\frac{\nabla\tau_p}{\theta_p\rho_p}+g \tag{5}$$

where τ_p is inter-particle normal stress, θ_p represents volume fraction of particles. The trajectory of a particle is solved by

$$\frac{dx_p}{dt}=v_p \tag{6}$$

where x is the location of the tracing particle.

2.2. Drag model

The Wen-Yu drag model is applicable to gas–solid flow with solids volume fraction lower than 0.61 while the Ergun drag model covers the range of 0.47–0.7. As the volume fraction of solids in the present study is less than 0.65 at close packing limit, the inter-phase drag function is defined by Wen-Yu model (Wen and Yu, 1966)

$$D_p=C_d\frac{3}{8}\frac{\rho_g}{\rho_p}\frac{|v_g-v_p|}{(3V_p/4\pi)^{1/3}} \tag{7}$$

where C_d is the drag coefficient. It depends on the Reynolds number, i.e..

$$\begin{aligned} C_d&=\frac{24}{\mathrm{Re}}(1+0.15\mathrm{Re}^{0.687})\theta_g{}^{-2.65} && \text{for } \mathrm{Re}<1000\\ C_d&=0.44\theta_g{}^{-2.65} && \text{for } \mathrm{Re}\geq 1000 \end{aligned} \tag{8}$$