Powder Technology 275 (2015) 30–38

Contents lists available at ScienceDirect

Powder Technology

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

Numerical investigation of performance of a fast gas–solid separator

Mengxi Liu, Chan Zhou, Jun Xie, Chunxi Lu *, Zhuan Wang

State Key Laboratory of Heavy Oil Processing, Faculty of Chemical Engineering, China University of Petroleum, Changping, Beijing 102249, PR China

A R T I C L E I N F O

Article history:
Received 31 August 2014
Received in revised form 4 January 2015
Accepted 12 January 2015
Available online 15 January 2015

Keywords:
Fast separator
Flow field
Separation efficiency
Computational fluid dynamics
Reynolds stress model

A B S T R A C T

In this study, the performance of a fast gas–solid separator was simulated based on the commercial software Fluent 6.2. The tangential velocity dominated the three-dimensional gas velocity, and decreased with the increase of the circumferential angle and radial position. In the separator housing, the swirling gas flow is similar to a modified or quasi free vortex. In the gas outlet tube, gas swirls with a rotation direction reverse to that in the separator housing. The ratio of circulation gas flow rate over the inlet gas flow rate changed only slightly with the increase of the inlet gas velocity, remaining at approximately 0.49. There was a stagnant particle layer on the separator base, inducing particle re-entrainment by circulation gas. In the present study, the separation efficiency of fine particles decreased with increasing inlet gas velocity, approximately remaining a constant as the particle diameter is greater than 12.5 μm.

1. Introduction

Gas–particle separators play a key role in a gas–solid fluidized bed by effectively collecting and recycling particles, and thus have been widely used in most industrial processes involving gas–solid fluidization. In recent years, collection efficiency and pressure drop have been considered to be the most important characteristics of a gas–particle separator [1–3], and sometimes even determine the long-term operation of the fluidized bed reactor. However, greater concern should be given when a separator is employed in a specific industrial process in which the reaction time is elaborately controlled, such as the fluid catalytic cracking (FCC) process. The FCC process converts heavy feeds into lighter products by cracking large molecules into smaller ones over the solid catalyst. It is expected that catalytic cracking reactions only occur in the riser reactor with a reaction time of less than 3 s [4]. Therefore, the residence time of lighter products in the separators mounted at the end of the riser reactor should be critically restricted to reduce unwanted side reactions and byproducts [5,6]. The most commonly employed scheme consists of multiple stage cyclones, providing high gas–solid separation efficiency, but requiring a long and only slightly controlled gas residence time of 15–45 s [4].

As an alternative of the cyclone, the fast separator has been attracting the attention of more and more researchers due to its many advantages, such as short residence time of gas and particles, low pressure drop, simple configuration, and small equipment size. Fast separators are usually located at the end of the riser or downer reactors, and are also known as riser or downer terminators. Conventional fast separators do not work well on gas–solid separation; for example, the T-type ballistic riser terminators offer a rough separation efficiency of only about 60%. Recently, a number of novel fast separators have been proposed to achieve high gas–solid separation efficiency. Donsí et al. [7] presented a separator with a U-bend channel, which presented a residence time of about 4 ms and a high separation efficiency of about 99% for coarse particles. However, the collective efficiency of Donsí's separator is sensitive to particle mass flow rate, significantly decreasing as the particle concentration at the inlet is lower than 4 kg/m^3, and the fraction of fine particles smaller than 40 μm also led to a drop of the separation efficiency. Andreus et al. [4] presented a short residence time separator (fast separator), which was capable of centrifuging gas and particles in a half-turn elbow with particle collection efficiency close to 95%. Variation of the back pressure exerted on the outlet of the dipleg has a slight influence on the particle collection efficiency, but a significant effect on the gas collection efficiency. Letzsch et al. [5] and Ross et al. [6] proposed a novel separator known as the ram horn. Gas and particles were separated in a half-turn elbow, then gas was discharged from a horizontally disposed gas opening and particles flow into the dipleg. It was found that one gas opening induced dramatic variation and rearrangement of the flow field, thereby decreasing the particle collection efficiency. More details on fast gas–solid separator technologies can be found in the review by Huard [8]. The present study evaluates a short residence time separator with multiple gas openings uniformly disposed on the gas outlet tube. This separator provides many significant advantages, such as remarkably small size compared to cyclone, simple configuration, high collection efficiency, low pressure drop, and short gas residence time.

The modern study of cyclone separators has taken advantage of several sophisticated experimental and numerical techniques. Pitot tubes and hot-wire anemometry [9] were used to measure the flow field of separators, but at the same time error was also induced by the intrusion of the flow field. Furthermore, measurement based on heat

* Corresponding author at: 18 Fuxue Road, Changping, Beijing, PR China, Postcode 102249. Tel./fax: +86 10 89733803.

http://dx.doi.org/10.1016/j.powtec.2015.01.023
0032-5910/

exchange may suffer from inaccuracy when dealing with unsteady flow [10]. The particle image velocimetry (PIV) [11] and laser Doppler velocimetry (LDV) [12–14] may be the prominent technologies in investigating flow field within separators, despite the fact that the need of seeding of gas with small particles, which is too prone to separation, results in some intrinsic difficulties [10]. Recently, with the improvement of the related mathematical models, computational fluid dynamics (CFD) has been proven to be a strong tool for describing the anisotropic and strongly swirling gas flow, particle concentration distribution, particle trajectories, grade efficiency, and so on [10,15,16]. However, the precise of simulation results strongly depends on the choice of turbulence model and numeric techniques. Based on the commercial software Fluent 6.2, the present work investigated the performance of a gas–solid separator, involving gas flow field, as well as pressure drop and separation efficiency.

2. Model description

2.1. Gas flow and turbulent models

The Reynolds-averaged mass conservation equation can be written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho \mathrm{u}_i)}{\partial x_i}=0 \tag{1}$$

the momentum conservation equation is:

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}\left(\rho u_i \mathrm{u}_j\right)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{\mathrm{u}_i'\mathrm{u}_j'}\right) \tag{2}$$

These Reynolds stresses, $-\rho\overline{u_i'u_k'}$, must be modeled in order to close Eq. (2). The flow field in a fast separator is characterized by a strongly swirling flow and anisotropic turbulence. There are many turbulence models available in Fluent, among which the k–ε model, algebraic stress model (ASM) and Reynolds stress model (RSM) are commonly used. The k–ε model is based on the assumption of isotropic turbulence, clearly disagreeing with the widely existing anisotropic turbulent flow within a gas–solid separator. ASM failed to predict the recirculation zone and Rankine vortex in strongly swirling flow [17]. RSM forgoes the assumption of isotropic turbulence and solves a transport equation for each component of the Reynolds stress. It has been proven that RSM is very suitable for simulating separator flow, despite its disadvantage of computationally expensive requirement [15,18,19].

In the RSM, the transport equation is written as

$$\frac{\partial}{\partial t}\left(\rho\overline{\mathrm{u}_i'u_j'}\right)+\frac{\partial}{\partial x_k}\left(\rho u_k\overline{\mathrm{u}_i'\mathrm{u}_j'}\right)=D_{ij}+P_{ij}+\phi_{ij}+\varepsilon_{ij} \tag{3}$$

where the left two terms are the local time derivative of stress and convective transport term.

The right four terms are

the stress diffusion term :

$$D_{ij}=-\frac{\partial}{\partial x_k}\left[\rho\overline{u'_k u'_i u'_j}+\overline{p'\left(\delta_{kj}u_i'+\delta_{ik}u_j'\right)}\right]+\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u'_i u'_j}\right)\right]$$

$$\text{the shear term}: P_{ij}=-\rho\left(\overline{u'_i u'_k}\frac{\partial u_j}{\partial x_k}+\overline{u'_j u'_k}\frac{\partial u_i}{\partial x_k}\right)$$

$$\text{the pressure–strain term}: \phi_{ij}=\overline{p'\left(\frac{\partial u'_i}{\partial x_j}+\frac{\partial u'_j}{\partial x_i}\right)}$$

$$\text{the dissipation term}: \varepsilon_{ij}=-2\mu\overline{\frac{\partial u'_i}{\partial x_k}\frac{\partial u'_j}{\partial x_k}}$$

The turbulence dissipation equation is

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho u_i\varepsilon)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+\frac{\varepsilon}{k}\left(\frac{1}{2}C_{\varepsilon1}P_{ij}-C_{\varepsilon2}\rho\varepsilon\right) \tag{4}$$

where $C_{\varepsilon1}=1.44$, $C_{\varepsilon2}=1.92$, $\sigma_k=0.82$ and $\sigma_\varepsilon=1$.

The finite volume method was used to discretize the partial differential equations of the CFD model, which used the Simple method for pressure–velocity coupling. The third-order accuracy QUICK upwind interpolation scheme was employed, due to its advantages of high accuracy, low numerical diffusion error, and so on. In the simulation, the time step is 4 ms, and the gravity direction is the negative direction of the y axis.

2.2. Gas–solid two-phase flow model

To calculate the trajectories of particles in the flow, the discrete phase model (DPM) was used to track individual particles through the continuum fluid. The particle loading in the separator was small, making it reasonable to assume that the presence of the particles did not affect the flow field. Except near the wall region, the concentration distribution of solids throughout most of the volume separator can be calculated using the Lagrangian approach without considering the interaction of particles. The momentum equation of a particle in the two-phase flow can be expressed as follows [20]:

$$\frac{du_p}{dt}=\frac{1}{\tau}\left(u+u'-u_p\right) \tag{5}$$

$$\frac{dv_p}{dt}=\frac{1}{\tau}\left(v+v'-v_p\right) \tag{6}$$

$$\frac{dw_p}{dt}=\frac{1}{\tau}\left(w+w'-w_p\right) \tag{7}$$

where u_p, v_p and w_p are particle velocity components, u, v and w are time-averaged gas velocity components, and u', v' and w' are gas fluctuation velocity components. τ stands for the relaxation time of particles and is calculated by the following [20]:

$$\tau=\frac{\rho_p d_p^2}{18\mu}\frac{24}{C_D\mathrm{Re}_p} \tag{8}$$

Here, Re_p is the particle Reynolds number and is defined as follows:

$$\mathrm{Re}_p=\frac{\rho d_p\left|\vec{u}_p-\vec{u}\right|}{\mu} \tag{9}$$

where u is the fluid phase velocity, u_p is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_p is the density of the particle, and d_p is the particle diameter. The drag coefficient, C_D, can be obtained from the following:

$$C_D=a_1+\frac{a_2}{\mathrm{Re}_p{}^p}+\frac{a_3}{\mathrm{Re}_p{}^{2p}} \tag{10}$$

where a_1, a_2 and a_3 are constants for smooth spherical particles over several ranges of Re_p given by Morsi and Alexander [21]. When the particle interacts with the fluid eddy, u', v' and w' are obtained by sampling from an isotropic Gaussian distribution with a standard deviation of $\sqrt{2k/3}$. Particle–eddy interaction time and dimension should not be larger than the lifetime and size of a random eddy.