Powder Technology 322 (2017) 471–489

Contents lists available at ScienceDirect

Powder Technology

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

Understanding the separation of particles in a hydrocyclone by force analysis

Yumeng Zhang [a], Pu Cai [a], Feihua Jiang [b], Kejun Dong [c], Yunchao Jiang [a], Bo Wang [a],*

[a] *Key Laboratory of Western China's Environmental Systems (Ministry of Education), Key Laboratory for Environmental Pollution Prediction and Control, Gansu Province, College of Earth and Environmental Sciences, Lanzhou University, Lanzhou 730000, PR China*
[b] *Sinopec Catalyst CO., LTD., Changling Division, Yueyang 414012, PR China*
[c] *Center for Infrastructure Engineering, Western Sydney University, Penrith, NSW 2751, Australia*

ARTICLE INFO

Article history:
Received 10 March 2017
Received in revised form 3 September 2017
Accepted 17 September 2017
Available online 22 September 2017

Keywords:
Statistical analysis
Spatial distribution
Hydrocyclone physical separation
Computational fluid dynamics

ABSTRACT

The multiphase flow in a hydrocyclone is studied using computational fluid dynamics (CFD). The Reynolds stress model (RSM) is used to describe the turbulent characteristics of the gas-liquid flow. The motions of particles are simulated by the Lagrangian particle tracking model (LPT). By the numerical simulation, the forces on particles are obtained and their spatial and statistical distributions are analyzed. The separation mechanisms of different sized particles are proposed based on the analysis. The results show that for small particles the main force is the fluid drag force. As the particle size increases, the fluid drag force decreases exponentially, while the effects of the centrifugal force and pressure gradient force are enhanced. The fluid drag forces in the axial and tangential directions are both rather random, while with the increase of particle size the randomness of the radial force gradually decreases. The particle motion is governed by the inward pressure gradient force, the outward centrifugal force and the fluid drag force with strong randomness. For different sized particles, the value of the outward centrifugal force is essentially about 2.5 times of the inward pressure gradient force, so that large particles get into the downstream and are collected in the spigot. As particle size decreases, the magnitude of the fluid drag force increases. In addition, the direction of the fluid drag force becomes more random, though it is generally inward. Under the inward drag force, a portion of particles are pushed to the central axis and escape through upstream. When the particle size is smaller than a certain value, the fluid drag force is significantly greater than the centrifugal force and pressure gradient force. In such case, the movements of particles are largely random, and the particles in different directions are uniformly distributed.

1. Introduction

Hydrocyclone is a highly efficient separation device for heterogeneous liquid mixtures [1]. Since its advent in 1891, it has been widely used in many industries due to its various advantages, including simple design and operation, high production capacity, low operation and maintenance costs, compact structure, small volume and light weight [2]. Although the structure of a hydrocyclone is very simple, the high shear flow of fluid, the layered distribution of particles and the interactions between multiple phases make the flow in the hydrocyclone very complicated.

The separation performance of a hydrocyclone is closely related to the flow field and the separation mechanism of particles. In particular, the research on the separation mechanism is helpful to deepen the understanding of the separation process and optimize the design of hydrocyclones. Owing to that the multiphase interactions in a hydrocyclone are extremely complex, there are no general theories that can accurately describe the separation mechanism. In the past 60 years, many scholars had put forward different theoretical models, such as the balanced orbit model [3,4], the residence time model [5] and the congestion model [6]. However, these models are based on various hypotheses on the flow in a hydrocyclone, and hence cannot accurately predict the performance of the hydrocyclone if the assumed flow is not fully consistent with the real situation. Computational fluid dynamics (CFD) method can overcome the deficiency by precisely predicting the flow under different operational conditions and hence to investigate the effects of different parameters on the separation performance. Moreover, the detailed flow field from CFD simulation is also very helpful to the understanding of the separation mechanism.

With the development of CFD, the research on the internal flow field of a hydrocyclone has been greatly advanced by numerical simulations. As the swirling flow in a hydrocyclone is an anisotropic and highly developed turbulent flow, it is of key importance to simulate the turbulent flow precisely in the CFD model. Normally there are three methods

---

* Corresponding author at: College of Earth and Environmental Sciences, Lanzhou University, PR China.
E-mail address: wangbo@lzu.edu.cn (B. Wang).

https://doi.org/10.1016/j.powtec.2017.09.031
0032-5910/

to model turbulent flows, including direct numerical simulation (DNS), large eddy simulation (LES) and Reynolds average Navier-Stokes (RANS). In DNS, the N-S equations are directly solved without any empirical parameters, and the transient flow field as well as vorticity can be obtained at various scales. However, the computational effort of DNS is vast, which limits its application to small scale systems. For example, Petty et al. [7] and Zhu et al. [8] used DNS to simulate the turbulent flow in hydrocyclones and obtained the full scale velocity distributions, including some fine vortex structures. However, due to the limitation of the computational cost, the diameter of the cyclone is only 5 mm. LES model is less computational demanding than DNS, while Wilcox [9] shows that it has a strict requirement on the grid size. If the grid size does not meet its requirement, the calculation results are not accurate, besides the number of the grids needs to be in proportion to $Re^{9/4}$. LES model is also widely used to simulate turbulent flow. For example, Delgadillo et al. [10] and Lim et al. [11] used the LES model to simulate the turbulent flow in hydrocyclones, and the results are in good agreement with the experiments. The most cost-effective method is RANS, mainly including the eddy viscosity model and the Reynolds stress model (RSM). In the early studies, many scholars used the eddy viscosity model in simulating hydrocyclones, but the predicted flow fields were not very accurate [12–14]. Later, RSM received more attention. In RSM, instead of using the assumed eddy viscosity, the Reynolds stress transport equation is established directly. The numerical solution of turbulent flow field is obtained by solving Reynolds averaged Navier-Stokes (N—S) equations. It has been shown that RSM is more accurate than the eddy viscosity model [15], while the computational cost is much less than LES and DNS, so it is widely used in modelling turbulent flow in hydrocyclones. For example, Wang et al. [16] and Leeuwner et al. [17] used RSM to predict the velocity distributions in hydrocyclones which agreed well with the experiments. Bhaskar et al. [15] compared the velocity distributions of a hydrocyclone predicted by standard k-ε, RNG k-ε and RSM, and found that the k-ε model gives the poorest results, while the RSM model can accurately predict the velocity distributions as well as the gas-liquid interface.

With the flow field being accurately simulated, the separation of particles in hydrocyclones can be studied based on the forces acting on the particles. The balanced orbit theory proposed by Barth [18] is a classical model based on the forces, which assumes that the radially outward centrifugal force and the radially inward drag force on cutting particles constitute a pair of balanced forces. Many scholars directly introduced this theory in evaluating the separation performance of hydrocyclones [19,20], assuming that the particles with high centrifugal force move towards the wall and flow downward to the spigot, while the particles with relatively high inward drag force approach the apex and escape through the vortex finder [21–26]. However, Wang [27] found that the fluid drag force, centrifugal force and pressure gradient force on cutting particles are in the same order and the radial fluid drag force has random direction, which indicates that the balanced orbit theory [18] may not be able to be directly applied to hydrocyclones.

Actually, in CFD model, we can use Lagrangian Particle Tracking (LPT) model to track the trajectory of a particle by solving Newton's equation of motion once we know the forces on the particle. The governing equation for the particle can be given by [27]:

$$\frac{du_p}{dt} = \left(1 - \frac{\rho}{\rho_p}\right) g + F_D + F_x \tag{1}$$

where u_p and ρ_p are the velocity and density of the particle respectively, g is the gravitational acceleration, ρ is the density of the fluid, F_D is the drag force per unit mass, and F_x represents other forces per unit mass. While generally the forces between a particle and fluid include the drag force, pressure gradient force, added mass force, Saffman lift force, Magnus force and basset force [28]. In hydrocyclones, as mentioned earlier, particles are mainly subjected to the drag force [29], so earlier studies ignored F_x in the LPT model [30,31]. But according to the later study of Wang [27], the pressure gradient force is also comparable to the drag force and therefore it must be considered. Indeed, Hsu [32] used the LPT model to track the movements of particles in a hydrocyclone without considering F_x, and the results showed an apparent error. On the other hand, Suasnabar et al. [33] and Mousavian et al. [34] considered the drag force, pressure gradient force, virtual mass force and gravity in modelling heavy medium cyclones and hydrocyclones respectively. Zhu et al. [8] simulated the multiphase flow in a hydrocyclone by DNS and they considered the fluid drag force, pressure gradient force, virtual mass force and gravity on particles. In these studies, it has been shown that the forces on particles in hydrocyclones may play different roles which deserves a better understanding. So far, however, few researchers studied all kinds of forces on different sized particles to understand the separation mechanism of hydrocyclones.

In this paper, most kinds of fluid forces on particles in a hydrocyclone are analyzed to better understand the separation of particles, including the drag force, pressure gradient force, added mass force, Saffman lift force and Magnus force. According to the previous study [35], the influence of Basset force can be ignored under the simulation conditions of this paper. And in the sparse liquid-solid two-phase flow system discussed here, as the concentration of solid phase is very low, the interactions between particles can be neglected. Statistical methods are used to analyze all the considered forces in a much more detailed way than previous studies. The spatial distributions of the main forces are also analyzed. The separation mechanisms of particles are then depicted based on the analyses of forces.

2. Method description

2.1. Numerical model

2.1.1. Reynolds stress turbulence model (RSM)

Due to the non closure of the N-S equations for turbulence, a variety of turbulence models have been developed to close the equations. The Reynolds stress model (RSM) is used here as it gives up the assumption of isotropic eddy viscosity and closes the N-S equations by directly solving the Reynolds stress transport equation and the dissipation rate equation. RSM is much better than the eddy viscosity model in considering the rotating fluid, the streamline bending and the sharply changing of the strain rate, which hence is more suitable for hydrocyclones.

The Reynolds averaged N-S equations are given by [36]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \tag{3}$$

where ρ and u_i are the density of liquid phase and the velocity of liquid phase respectively, $\bar{u}_i'$ and $u_i{'}$ are the mean and fluctuating velocities respectively, x_i is the positional length and $\rho\overline{u_i' u_j'}$ represents the Reynolds stress tensor.

The Reynolds stress transport equation is [37]:

$$\frac{\partial}{\partial t}\left(\rho\overline{u_i' u_j'}\right) + \frac{\partial}{\partial x_k}\left(\rho u_k \overline{u_i' u_j'}\right) = D_{T,ij} + P_{ij} + \varnothing_{ij} + \varepsilon_{ij} \tag{4}$$