International Journal of Mining Science and Technology 24 (2014) 123–127

Contents lists available at ScienceDirect

International Journal of Mining Science and Technology

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

Modeling the effects of mechanical parameters on the hydrodynamic behavior of vertical current classifiers

Arabzadeh Jarkani Soroush [a], Khoshdast Hamid [b,*], Shariat Elaheh [a], Sam Abbas [a]

[a] *Mining Engineering Department, Faculty of Engineering, Shahid Bahonar University, Kerman 76169-133, Iran*
[b] *Mining Engineering Department, College of Mining and Industry, Shahid Bahonar University, Zarand 77611-56391, Iran*

ARTICLE INFO

Article history:
Received 30 May 2013
Received in revised form 15 June 2013
Accepted 5 July 2013
Available online 4 January 2014

Keywords:
Hydraulic classifier
Modeling
Computational fluid dynamic
Cut size

ABSTRACT

This study modeled the effects of structural and dimensional manipulations on hydrodynamic behavior of a bench vertical current classifier. Computational fluid dynamics (CFD) approach was used as modeling method, and turbulent intensity and fluid velocity were applied as system responses to predict the overflow cut size variations. These investigations showed that cut size would decrease by increasing diameter and height of the separation column and cone section depth, due to the decrease of turbulent intensity and fluid velocity. As the size of discharge gate increases, the overflow cut-size would decrease due to freely fluid stream out of the column. The overflow cut-size was significantly increased in downward fed classifier compared to that fed by upward fluid stream. In addition, reforming the shape of angular overflow outlet's weir into the curved form prevented stream inside returning and consequently unselective cut-size decreasing.

1. Introduction

Vertical current hydraulic classifiers (VCHC) are well known as efficient devices for indirect classification of particles in water-base separation systems. They are used in many mineral applications such as the removal of clay fines from siliceous sands, particle size control in closed circuits with mills, fine control in taconite pellet washing, dewatering of coal tailing prior to centrifugation, silica removal from iron ores, cement purification, etc. [1–11]. With respect to the wide application of VCHCs, better understanding of the effects of operating parameters on their performance is of crucial importance. Several operating variables including washing liquid rate, size distribution of feed and overflow product, solid content of pulp and properties of washing liquid have been considered in literature, which are all related to the conditions of feed and overflow product [6,12,13]. In spite of development of tens of VCHC designs available, less attention has been paid to the effects of mechanical parameters.

The cut size of overflow (d_{50}), i.e., the size in which particles have found equal chance to transfer into overflow or underflow streams, is commonly used as a measure for performance evaluation of classifiers. This parameter is directly influenced by washing liquid upward velocity and the turbulent level of separation chamber. Thus, it would be possible to predict the cut size variations by modeling thereof. Present study is aimed to investigate the effects of mechanical and designing parameters on overflow cut size of a laboratory scale vertical current hydraulic classifier with a computational fluid dynamic (CFD) approach.

2. Modeling of VCH classifier

2.1. Theory of the model

It is well known that the best choice for modeling flow patterns in water-base separators is to apply incompressible Navier–Stokes equation in combination with a turbulent flow model. To predict the fluid flow pattern in VCH classifier, the governing equation consists of the continuity and momentum balance equations for the liquid phase as follow:

$$\nabla \cdot \rho v = 0 \tag{1}$$

$$\nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} \tag{2}$$

where ρ is the fluid density; g the gravity; v the velocity of fluid; and p the static pressure. The stress tensor τ can be calculated as below:

$$\vec{\tau} = \mu_{\text{effective}} \left[(\nabla v) - \frac{2}{3} \nabla \cdot \rho \vec{v}^2 \right] \tag{3}$$

$$\mu_{\text{effective}} = \mu + \mu_t \tag{4}$$

---

* Corresponding author. Tel.: +98 913 5325904.
E-mail address: khoshdast_hamid@yahoo.com (H. Khoshdast).

2095-2686/$ - see front matter © 2014 Published by Elsevier B.V. on behalf of China University of Mining & Technology.
http://dx.doi.org/10.1016/j.ijmst.2013.12.021

where μ and $\mu_{\text{effective}}$ are dynamic and effective, respectively [14,15]. The momentum equation can be solved using a turbulent flow (TF) model. The standard k–ε dispersed turbulence model is a TF model commonly used for engineering purposes. Variables k and ε characterize turbulent kinetic energy and turbulent dissipation rate, respectively. The k–ε model is solved based on equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial X_i}(\rho k u_i)=\frac{\partial}{\partial X_j}\left[\left(\mu+\frac{\mu_i}{\sigma_k}\right)\frac{\partial k}{\partial X_j}\right]+G_k-\rho\varepsilon \tag{5}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_i}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

where G_k is the kinetic energy due to velocity gradient; and μ_t the viscosity of turbulent flow. These parameters can be calculated as follows:

$$G_k=\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i} \tag{7}$$

$$\mu_t=\rho C_\eta\frac{k^2}{\varepsilon} \tag{8}$$

where u' is the velocity vector; and $C_{1\varepsilon}$, $C_{2\varepsilon}$, σ_k, σ_ε and C_n are constant values [16].

2.2. Modeling procedure

This study simulated the effects of mechanical parameters including separation column height (162.5 and 267.5 mm) and diameter (90 and 180 mm), diameters of washing water inlet (15 and 20 mm) and overflow discharge gate (25 and 35 mm), depth of separation cone (45 and 90 mm), and the shape of overflow weir (angular and curved) on the flow pattern of a laboratory scale VCH classifier. The dimensions of the VCH classifier are shown in Fig. 1. The simulation used the steady state, pressure based, implicit formulation of Fluent 6.3 software which employs finite volume method and the 2D physical meshing of the classifier was constructed in pre-processor Gambit 2.3. The initial and boundary conditions were set on the basis of experimental data: water velocity 1.48 m/s, pulp density of 1094 kg/m^3 (15% solid content), and atmospheric pressure [17,18]. In order to approximate more accurate results, the residual convergence and iteration values were fixed at 1×10^{-5} and 15,000, respectively [19,20].

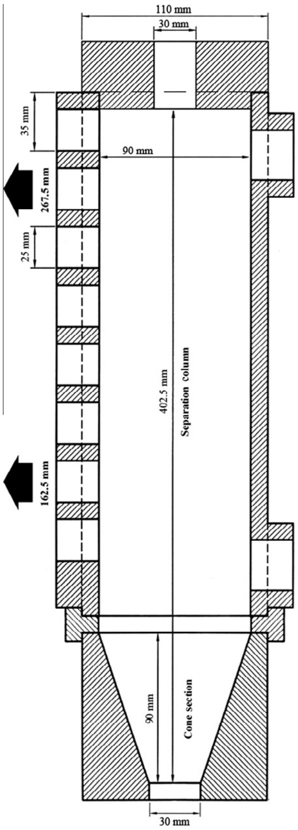

Fig. 1. Geometry of laboratory vertical current hydraulic classifier studied.

2.3. Validation of the model

The validation process of the model was done using the experimental data reported in our previous work [21]. In this regard, the water velocity in overflow discharge gate was plotted against predicted values. As shown in Fig. 2, the predicted values are in good agreement with experimental measurements. This confirms the accuracy of the model chosen.

3. Simulation results and discussion

3.1. Effect of separation column height

Fig. 3 shows the variation in turbulence pattern at different column heights. As seen in Fig. 3, despite the same turbulent intensity values, as column height decreases the fluid pattern approaches to turbulent condition. Fluid velocity patterns inside the columns are also modeled and showed in Fig. 4. Although both fluid velocity models are of similar velocity distribution, the overflow cut-size in shorter classifier is expected to increase. This is caused by turbulent condition which decreases retention time required for efficient separation process to segregate coarse particles from fines.

3.2. Effect of separation column diameter

Fig. 5 shows that turbulent intensity significantly decreases by increasing the column diameter. As shown in Fig. 6, the turbulence pattern approaches to steady conditions leading to the decrease in fluid velocity. Fig. 6 shows that a wide stationary zone dominates inside the column with higher diameter. This may result in a significant decrease in cut-size of overflow product since particles have enough time to be segregated.

3.3. Effect of cone section depth

Simulation results for turbulence and fluid velocity patterns for classifiers with different cone size are given in Figs. 7 and 8, respectively. Figs. 7 and 8 clearly show that both turbulent intensity and fluid velocity increase by decreasing of cone section depth. The

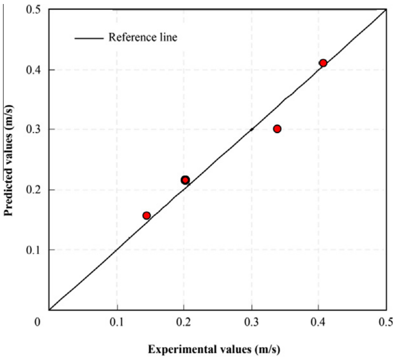

Fig. 2. Comparison between experimental data and simulation results.