Crimson Publishers
Wings to the Research

Opinion

Numerical Simulation on Hydraulic Circulation Chamber in a Deep Clarification Device for Beneficiation Circulating Water

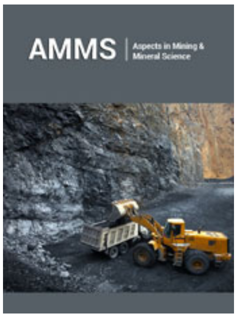
Kai Tian[1,2,3], Jiahong Li[1,2], Wentao Hu[1,2,3]*, Siyi Liang[3] and Huajun Wang[1,3]

[1]Key Laboratory of High-Efficient Mining and Safety of Metal Mines Ministry of Education (USTB), University of Science and Technology Beijing, China

[2]State Key Laboratory of Mineral Processing of China, China

[3]National Technical Center of Circular Economy, MCC Capital Engineering & Research Incorporation Limited, China

ISSN: 2578-0255



***Corresponding author:** Wentao Hu, Key Laboratory of High-Efficient Mining and Safety of Metal Mines Ministry of Education (USTB), University of Science and Technology Beijing, Beijing, 100083, China

Submission: November 01, 2021

Published: November 10, 2021

Volume 7 - Issue 5

How to cite this article: Kai Tian, Jiahong Li, Wentao Hu, Siyi Liang, Huajun Wang. Numerical Simulation on Hydraulic Circulation Chamber in a Deep Clarification Device for Beneficiation Circulating Water. Aspects Min Miner Sci. 7(5). AMMS. 000672. 2021.
DOI: 10.31031/AMMS.2021.07.000672

Opinion

The beneficiation plants generally process the production wastewater of the whole plant through natural concentration and clarification, and then feedback to each working procedure through the circulation pump house. This part of water referred is called circulating water in beneficiation plants. However, the circulating water in some beneficiation plants contains an excess suspended solids, which fail to meet the process demand of mineral separation [1-4]. Thus, it is necessary to conduct further treatment on the circulating water to increase water quality. The suspended solid in the circulating water is fine-grained, and difficult to precipitate due to separate particles is found uneasy to collide and forming flocs. The current investigation proposed a new type of deep clarification device to remove suspension solids, which was mainly composed of a hydraulic circulation chamber and a sludging and settling chamber. The schematic diagram of the device is shown in Figure 1. The hydraulic circulation chamber is mainly based on the principle of finite space jet flow [5]. As shown in Figure 1, the hydraulic chamber is mainly implemented on the basis of the principle of limited space jet sludge circumfluence to achieve improved suspended particles collide. With the increase of suspended solids concentration of backflow water in the bottom part of device throat, the collision possibility of the solids with particles carried by the ejected raw water from the nozzle promoted, and the particle size of the backflow water was different from that of the raw water, hence the flocculation effect was significantly improved. The separation chamber of suspended solids and water was majorly designed to promote the settling effect of particles and improve the solid-liquid separation effect by the function of the inclined plate. The appearance of Computational Fluid Dynamics (CFD) provides great convenience for understanding the flow pattern inside the device [6-8]. CFD technology is used to conduct numerical simulation research, in order to reveal the hydraulic characteristics of the hydraulic circulation area of the device.

The CFD numerical simulation of hydraulic circulation chamber is shown in Figure 2. As shown in (Figure 2a), it found that the flow jet out from the nozzle and through the throat, afterwards flow across the upward side flap and then dispersed on both sides of the hydraulic circulation area. Some of the water flowed from both sides to spate separation chamber, and the other part flow back to the throat from the bottom part of itself, thereby formed a state of continues circulation. At the same time, it found that obvious vortices were formed at the upper part of the hydraulic circulation area and the lower part of the diversion flow on both sides. The existence of such water circulation and vortex flow state was conducive to the adequately mixed of water and coagulant [9] and created conveniences for the subsequent flocculation reaction. As shown in (Figure 2b), it found that the pressure energy was converted into kinetic energy through the contraction section of the nozzle [10], making the velocity of the raw water flow at the end of the nozzle increase sharply, when the water flows into the nozzle at a certain speed. In the contact stage of raw water and reflux water, the velocity of the jet out raw water was fast, while the velocity of coiling reflux water was slow, thus the boundary layer of the two fluids was obvious. The constant velocity flow at the center of the pipe became a conical flow, and the velocity decreases, while the velocity of the backflow near the side wall of the pipe gradually increased, with the continuous contact between the mixed raw water and the backflow. It indicated that radial turbulent diffusion occurred in the throat and the raw water, and the returned water mixed violently, resulting in the energy exchange. It provided good dynamic conditions for the collision of particles carried by the raw water and reflux water, and promotes the occurrence of contact flocculation, in the fierce mixing process of raw water and reflux water in the throat pipe. As the water flow downstream, energy continues to be dissipated so that the overall flowed velocity of the water decreased. The mixed water spreads evenly after it flowed through the upper baffle, and the velocity and uniform flow state of the water trickle reduced the particle collision rate. The flocculation condition generated in the hydraulic circulation chamber was found conducive to the further growth of the micro flocs and made the latter denser and firmer [11]. In the current investigation, the distribution of fluid, including water and suspended solids, in the water circulation area of the device was clearly revealed, and it provided a new reference for the subsequent structure optimization of the deep clarification device of circulating water.

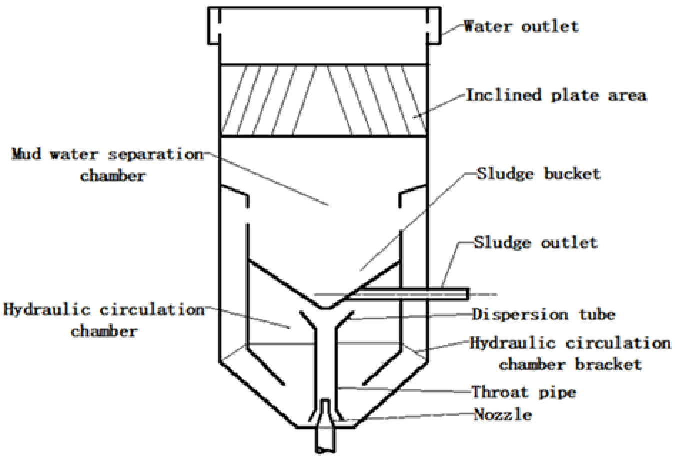


Figure 1: Schematic diagram of deep clarification device.

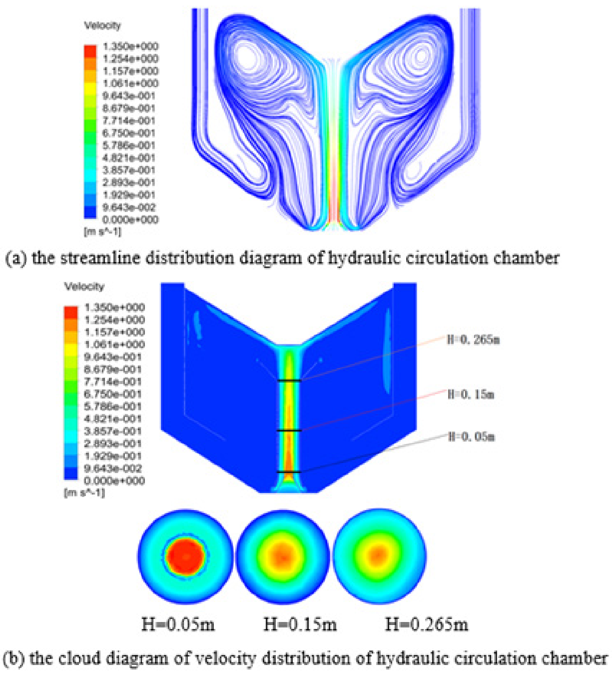


Figure 2: The CFD numerical simulation of hydraulic circulation chamber.

The authors gratefully acknowledge the support of the National Key Research and Development Project of China with the grant number 2018YFC1900604, the Open Foundation of State Key Laboratory of Mineral Processing of China with the grant number BGRIMM-KJSKL-2020-11, and the Fundamental Research Funds for the Central Universities with the grant number FRF-IP-20-02.

References

1. Liang QG, Zhao Q, Liu B, Du Z, Xia X (2020) Treatment and reuse of process water with high suspended solids in low-grade iron ore dressing. Journal of Cleaner Production 278: 123493.
2. Bicak O, Ozturk Y, Ozdemir E, Ekmekci Z (2018) Modelling effects of dissolved ions in process water on flotation performance. Minerals Engineering 128: 84-91.
3. Liu W, Moran CJ, Vink S (2013) A review of the effect of water quality on flotation. Minerals Engineering 53: 91-100.
4. Manono MS, Corin KC, Wiese JG (2013) The effect of ionic strength of plant water on foam stability: A 2-phase flotation study. Minerals Engineering 40: 42-47.
5. Ceglia G, Invigorito M, Chiatto M, Greco CS (2020) Flow characterization of an array of finite-span synthetic jets in quiescent ambient. Experimental Thermal and Fluid Science 119: 110208.
6. Goula AM, Kostoglou M, Karapantsios TD (2008) A CFD methodology for the design of sedimentation tanks in potable water treatment: Case study: The influence of a feed flow control baffle. Chemical Engineering Journal 140: 110-121.
7. Shah MT, Parmar HB, Rhyne LD, Kalli C, Utikar RP, et al. (2019) A novel settling tank for produced water treatment: CFD simulations and PIV experiments. Journal of Petroleum Science and Engineering 182: 106352.
8. Patziger M (2016) Computational fluid dynamics investigation of shallow circular secondary settling tanks: Inlet geometry and performance indicators. Chemical Engineering Research and Design 112: 122-131.
9. Sun Y, Zhou S, Chiang PC, Shah KJ (2019) Evaluation and optimization of enhanced coagulation process: Water and energy nexus. Water-Energy Nexus 2(1): 25-36.
10. Shi FG, Zhao J, Sun XG (2021) Numerical simulation of gas-liquid two-phase jet surface cleaning flow field. Journal of Central South University 52: 960-970.
11. Zhu W, Wang ZX, Han X (2019) Numerical simulation of air-carried vortex flocculation reactor. Journal of China University of Mining and Technology 48: 911-918.

