WEP Warwick Evans Publishing

International Journal of Mechanical and Electrical Engineering

ISSN: 3005-9615 (Print), ISSN: 3005-7132 (Online) | Volume 1, Number 1, Year 2023
DOI: https://doi.org/10.62051/ijmee.v1n1.08
Journal homepage: https://wepub.org/index.php/IJMEE/index

CFD-based study of inlet flow rate variation on flow field of micro-cyclone

Bohan Li, Zhishan Bai, Huiqing Luo, Xinle Meng

School of Mechanical Engineering, Sichuan University of Science & Engineering, Yibin 643000, China

ABSTRACT

In this study, the operational parameters of the micro-cyclone were investigated. A 10 mm micro-cyclone suitable for waste drilling mud treatment was designed and the effect of the inlet on the flow field within the micro-cyclone as well as the total separation efficiency was investigated using CFD techniques. It was concluded that the pressure gradient inside the micro-cyclone increased with increasing inlet flow; axial and tangential velocities increased with increasing inlet flow; the peak turbulent kinetic energy increased with increasing inlet flow; and the total separation efficiency increased and then decreased slightly with increasing inlet flow.

KEYWORDS

Waste drilling mud; Micro cyclones; Separation efficiency; Pressure; Velocity.

1. INTRODUCTION

In drilling operations for oil and gas exploration and production, drilling fluids are used in large quantities for controlling formation pressure, suspending rock cuttings, stabilizing the wellbore, cooling and lubricating drilling tools, and sealing the formation [1]. As a result, a large amount of waste drilling mud is generated. According to incomplete statistics, each drilling a well produces about 300m3 of waste drilling mud to, this calculation, the global annual amount of waste drilling fluids up to tens of millions of cubic meters, and is still a slowly rising trend [2]. Since waste drilling mud is mainly composed of large rock chips, small suspended solids, oil, organic chemical treatments, inorganic salts, and heavy metal ions, the discharge of this waste drilling mud into the environment without treatment can cause serious environmental problems in the terrestrial, aquatic, and aerial environments, such as decreasing the fertility of the soil or negatively affecting the flora and fauna and causing health problems [3,4]. As the environmental protection of countries around the world continues to be strengthened, the treatment of waste drilling mud generated during oil drilling is also receiving more and more attention, so the development of a green waste drilling mud treatment technology from an environmental and economic point of view is an urgent problem to be solved.

A hydro-cyclone is a device that utilizes the velocity and pressure of the fluid itself as it enters the interior of the cyclone to generate rotational motion, usually for the separation of non-homogeneous multiphase materials. Since the 1950s, hydro-cyclones have been used on a large scale for more than 70 years. Hydrocyclone is mainly composed of five parts: cylindrical section, inlet, cone section, overflow pipe, underflow pipe. The specific working principle of hydro-cyclone is to have a certain density difference of non-homogeneous multi-phase mixture at a certain speed or pressure is continuously input into the hydro-cyclone, and in the cyclone internal high-speed rotary movement to produce a strong centrifugal force. In the centrifugal force as well as the mixture's own gravity under the action of rapid stratification, the density (or diameter) of the mixture of components (heavy phase) is thrown to avoid, and in the continuous outward as well as downward motion to form the external cyclone, and ultimately discharged by the underflow pipe; and the density (or diameter) of the components of the smaller (light phase) is constantly inward and upward rotary motion to form the internal cyclone, and ultimately discharged by the overflow pipe [5~7]. At present, hydro-cyclones are very common in industrial applications due to their simple structure and high separation efficiency, which makes the research on hydro-cyclones has been a hot research issue. However, at present, the research on hydro-cyclone mostly focuses on the research on hydro-cyclone with larger diameter, while the research on micro hydro-cyclone with small diameter is still less. Meanwhile, the computational fluid dynamics (CFD) technique is widely used in the study of the internal flow field characteristics and particle separation process of hydro-cyclone because of its advantages of not requiring any test and detection of the side backs, no need for repetitive validation tests, no negative impact on the flow field, and clear and comprehensive simulation results [8~10].

In this study, CFD technology is adopted, based on the fluid simulation software Fluent, the RSM model is used to calculate the turbulence and the Mixture model is used to simulate the results of particle separation, and the effect of the inlet flow change of a micro-cyclone with a diameter of 10mm on its internal flow characteristics and separation efficiency is systematically investigated.

2. MODEL SELECTION AND VALIDATION

2.1. physical model

The ϕ10 mm micro-cyclone is selected as the simulation object, and its mesh is divided as shown in Fig. 1, with the coordinate origin at the center of the circle at the top of the overflow port. The entire computational domain is meshed hexahedrally by ICEM software, and the meshes at the wall and the overflow pipe are locally encrypted to enhance the wall effect. Meanwhile, in order to ensure the accuracy of the simulation results, a mesh-independence check of the model is required before the formal simulation to determine the influence of the mesh on the simulation as well as to determine the appropriate mesh size for the subsequent formal simulation. Since the fluid domain mesh of the microcyclone in this paper is generated by ICEM plotting, the number of meshes can be adjusted in it. Three different numbers of grids, 565462, 712941, and 962837, were generated with a grid quality of 0.5 or higher. The three meshes are simulated separately, the simulation medium is air, and the simulation results are shown in Figure 2. It can be seen that the simulation results are basically the same when the number of grids is greater than 712941. Therefore, considering the simulation accuracy and simulation calculation time, this paper selects the grid number of 712941 for simulation calculation.

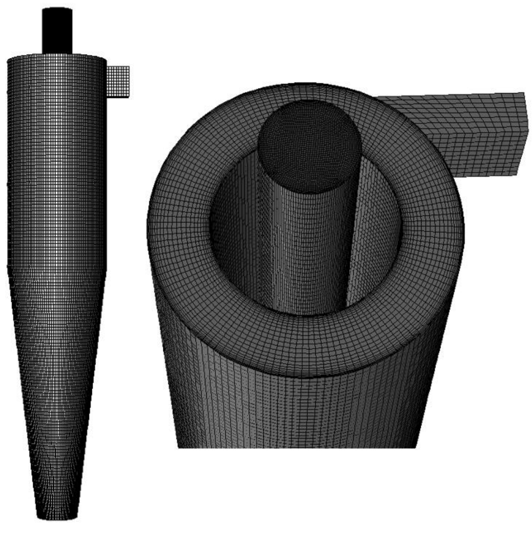

Fig.1 Micro-cyclone geometric model and structured mesh

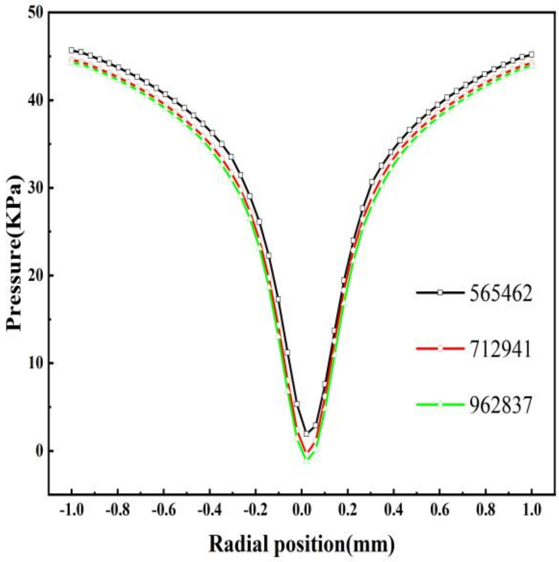

Fig.2 Grid independent verification of mini-hydrocyclone considered

2.2. Simulation parameterization and boundary condition setting

Since this paper studies the micro-cyclone for solid-liquid separation applied to waste drilling mud, and the solid phase particles are greater than 10%, the mixture model is chosen as the multiphase flow model, the water phase is set as the main phase, the particle phase is set as the secondary phase, and the particle phase particle size is set according to the response of the actual range of 0.168~244.9μm. The inlet of the micro-cyclone is set as the velocity inlet, the inlet velocity is converted by the inlet flow rate, the inlet flow rate is set to five grades of $0.1m^3/h$, $0.125m^3/h$, $0.15m^3/h$, $0.175m^3/h$, $0.2m^3/h$, the overflow port and the underflow port are set to free flow, the diversion ratio is controlled by the setting of the flow rate weighting, and the diversion ratio is set to 25% of the five variables, the wall is set as the main phase with no sliding wall. Walls were used with no-slip wall conditions. The QUICK format was used as the discrete phase format, PRESTO! format as the pressure interpolation format, and SIMPLEC algorithm as the pressure-velocity coupling algorithm to calculate the model.

2.3. Methods of analyzing simulation results

(1) Flow field analysis. The Y=0 cross section was selected to generate the flow field cloud. Based on these cloud maps, the variation of different inlet flow rates are analyzed in relation to the three-dimensional velocity field, pressure field, turbulent kinetic energy, and pressure drop.

(2) Total Separation Efficiency Analysis. The total separation efficiency of ultrafine particles is calculated for different diversion ratios, and the corresponding total separation efficiency curve is derived. The total separation efficiency is calculated as Eq (1).

$$\mu = m_u / m_i \tag{1}$$

where m_u is the mass flow rate at the bottom flow inlet and m_i is the mass flow rate at the inlet.

3. RESULT

3.1. Effect of inlet flow rate on pressure field

The cloud diagram of the pressure field inside the micr-ocyclone with different inlet flow rates is shown in Fig. 3, and the inlet flow rate gradually increases from a to e. The pressure field inside the microcyclone with different inlet flow rates is shown in Fig. 3. From the cloud diagram, it can be seen that the overall distribution pattern of the pressure cloud does not increase with the increase of the inlet flow rate. However, as the inlet flow increases, the pressure value and pressure gradient inside the microcyclone will gradually increase. For example, when the inlet flow rate is low at 0.1~0.15 m3/h, the color change of the pressure cloud is not obvious except for the other areas of the microcyclone except for the axial region, which indicates that the pressure change is small; when the inlet flow rate is greater than 0.175 m3/h, the red positive pressure and high pressure area (red area) starts to appear at the inner wall of the microcyclone, and with the further increase of the inlet flow rate, the area occupied by the red area is also increasing. With the further increase of the inlet flow rate, the area occupied by the red region is also increasing; when the inlet flow rate is further increased to 0.2 m3/h, the area occupied by the red region extends from the inner wall of the micro-selector to the vicinity of the outer wall of the overflow port. At the same time, the area occupied by the negative pressure and high pressure zone (dark blue area) in the axial region of the microcyclone gradually expands axially from near the bottom of the overflow opening to near the bottom flow opening as the inlet flow rate increases.

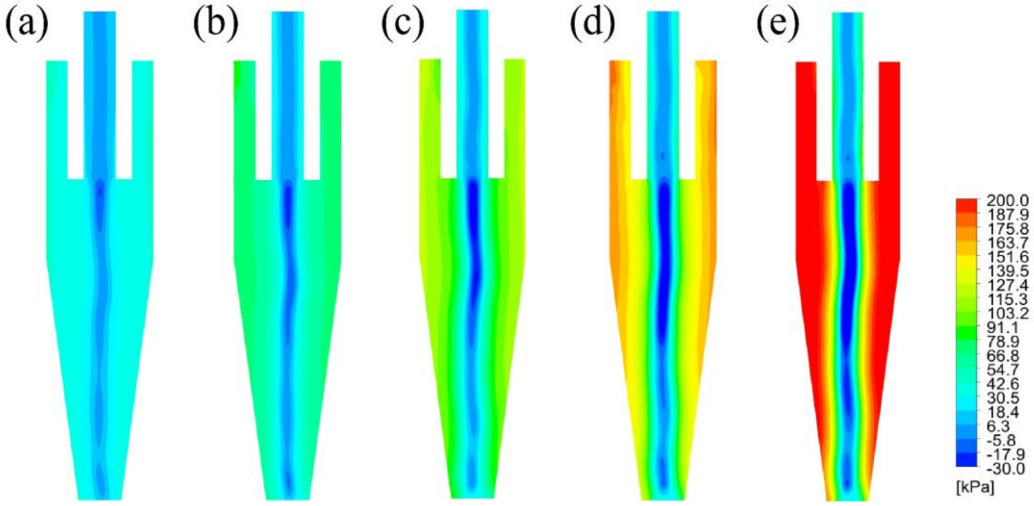

Fig.3 Pressure contours at different inlet flow rate

3.2. Effect of inlet flow on axial velocity

In the cyclone separation process of the cyclone, since the overflow and underflow ports of the cyclone are located in the axial direction, the axial velocity generally affects the time of solid-liquid phase separation in the cyclone. In the case of the rest of the conditions are consistent, generally speaking, the greater the axial velocity, the shorter the time required for the separation of materials in the cyclone. The axial velocity inside the micro-cyclone at the same inlet flow rate is shown in Fig. 4, and the inlet flow rate from a to e is gradually increasing. As can be seen from the cloud diagram, with the increase of the inlet flow rate, there will be relatively alternating red positive high-speed intervals near the inner wall of the microcyclone, and with the increasing inlet flow rate, the area of the red high-speed intervals, as well as the number of the increase in the area of the red high-speed intervals. This is because the inner wall of the microcyclone belongs to the external cyclone region, so in the case of increasing inlet flow, the media inside the microcyclone speed is also increasing, which makes the axial velocity of the external cyclone increasing, and because the external cyclone movement for the spiral downward movement, so the red high-speed intervals in the vicinity of the inner wall for the relative alternation of the presentation. In the axial region, with the increase of the inlet flow, the blue negative high-speed intervals gradually appear, and with the gradual increase of the inlet flow, the blue negative high-speed intervals will gradually occupy the entire axial region.

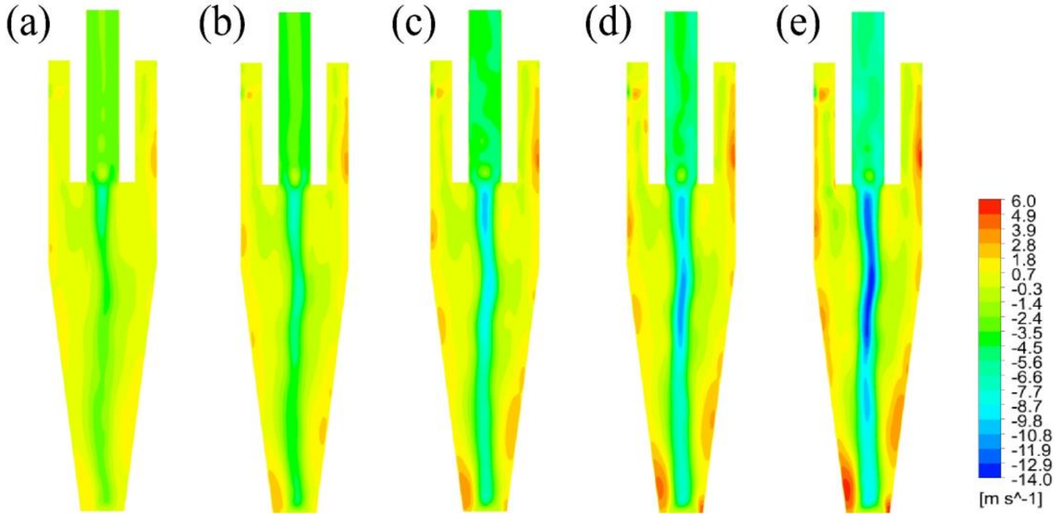

Fig.4 Axial velocity contours at different inlet flow rate

3.3. Effect of inlet flow on tangential velocity

The role of tangential speed is to drive the fluid inside the cyclone to do high-speed rotary movement, and through the centrifugal force will be the density of the heavier phase thrown to the cyclone wall, to achieve the purpose of light and heavy two-phase separation. Generally speaking, increasing the tangential velocity will make the separation efficiency relatively increase, but because the space inside the cyclone is usually small, so if the tangential velocity is large, it will cause internal flow disorder, but make the separation efficiency decrease. The radial velocity cloud diagrams inside the microcyclone under different inlet flow rates are shown in Figure 5, and the inlet flow rate from a to e is gradually increasing. As can be seen from the cloud diagram, there is a blue high-speed interval at the inner wall of the overflow port, and with the increase of the inlet flow rate, this blue interval in the axial and radial direction gradually to the two sides of the diffusion, when the inlet flow rate of 0.2 m3/h, the blue high-speed intervals in the axial direction diffuse to the outlet of the overflow port and the bottom of the mouth near the mouth, and in the radial direction, the diffusion to the inner wall of the microcyclone. At the same time, with the increase of the inlet flow, the area of the high-velocity zone at the microcyclone inlet is also increasing. However, the variation of tangential velocity in the axial region of the microcyclone is not significant with the inlet flow rate. In addition, for the area between the inner wall of the microcyclone and the outer wall of the overflow port, the tangential velocity gradually decreases in the radial direction in the cylindrical section inserted in the overflow port, i.e., the medium has a larger tangential velocity the closer it is to the inner wall of the microcyclone, and a smaller one the closer it is to the outer wall of the overflow port, while in the rest of the cylindrical section as well as in the conical section, the tangential velocity gradually increases in the radial direction.

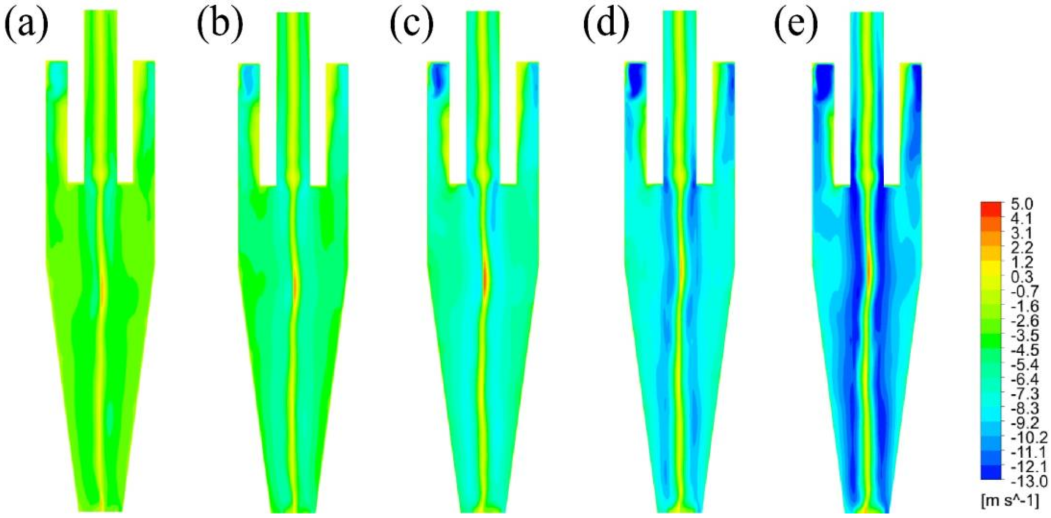

Fig.5 Tangential velocity contours at different inlet flow rate

3.4. Effect of inlet flow on turbulent kinetic energy

Turbulent kinetic energy reflects the turbulence intensity and size of the fluid. In general, the larger the turbulent kinetic energy, the greater the turbulence intensity and turbulence scale of the fluid. The distribution of turbulent kinetic energy inside the microcyclone at different inlet flow rates is shown in Figure 6, and the inlet flow rate from a to e is gradually increasing. From the cloud diagram, it can be seen that with the increase of the inlet flow rate, higher turbulent kinetic energy occurs at the bottom of the overflow port and the bottom flow port of the microcyclone, and the maximum value of turbulent kinetic energy occurs at the bottom flow port. This is due to the bottom of the overflow port region of the fluid flow is more turbulent, and the larger the inlet flow rate, the stronger the degree of turbulence, so the turbulence energy is relatively large; while the bottom of the inlet flow rate increases, the bottom of the inlet flow rate of the discharge velocity is smaller than the entry velocity of the fluid medium, so that the bottom of the inlet turbulence energy is larger. In addition, in the case of the inlet flow rate of 0.1~0.15m3/h, the turbulent kinetic energy in the axial region of the micro-selector are maintained at a low level; when the inlet flow rate of 0.175m3/h, in the axial region began to appear a certain degree of high turbulence kinetic energy region and radial diffusion; when the inlet flow rate is further increased to 0.2m3/h, in addition to the inner wall of the junction of the column cone, the entire micro-cyclone is basically covered by high turbulence kinetic energy region and spreads radially outward. When the inlet flow rate is further increased to 0.2 m3/h, except for the inner wall of the column-cone junction, the entire interior of the microcyclone is basically covered by the high turbulent kinetic energy region. In general, although the inlet flow did not change the distribution law of the turbulent kinetic energy, with the increase of the inlet flow, the turbulent kinetic energy inside the microcyclone gradually increased, and the fluid movement was more intense.

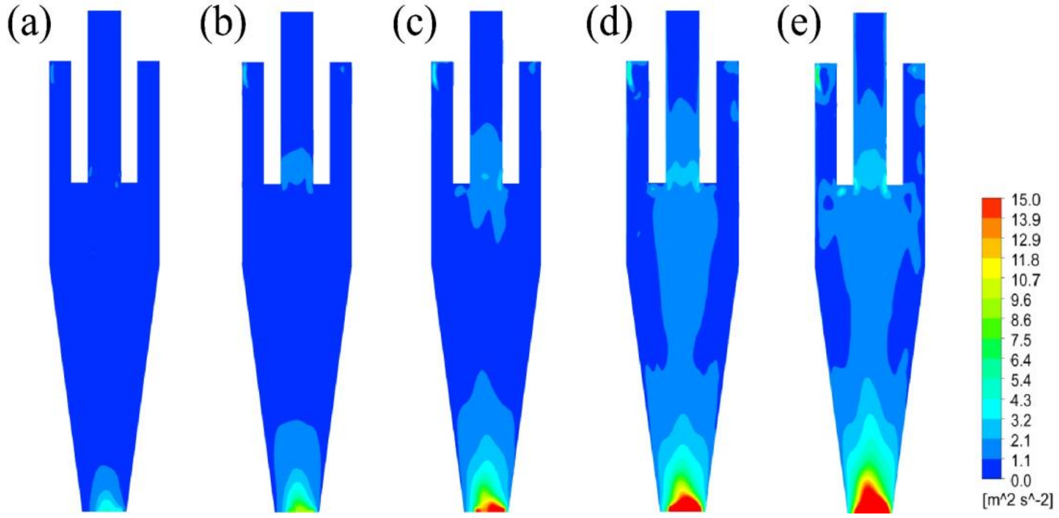

Fig.6 Turbulent kinetic energy contours at different inlet flow rate

3.5. Effect of inlet flow rate on separation efficiency

The effect of inlet flow rate on the total separation efficiency is shown in Fig. 7. From the figure, it can be seen that with the increase of inlet flow rate, the total separation efficiency presents a trend of first increasing and then slightly decreasing, when the inlet flow rate increases from 0.1m3/h to 0.15m^3/h, the increase in the total separation efficiency is 16.12%, while when the inlet inlet flow rate continues to increase the value of 0.2m^3/h, the total separation efficiency decreases, the decrease is 10.11%.

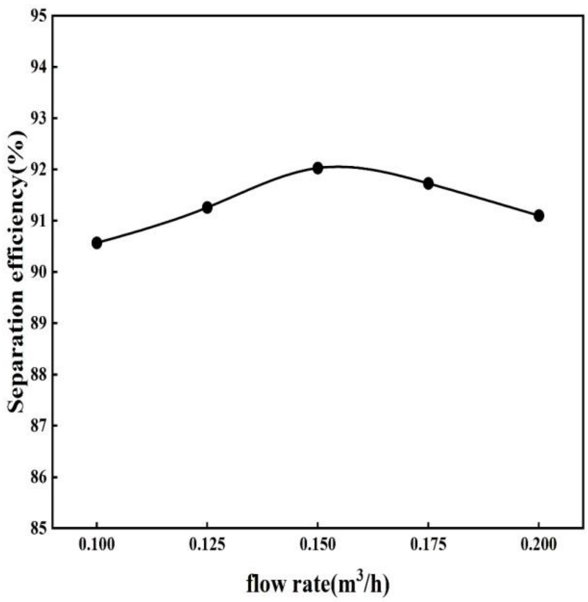

Fig.7 The influence of inlet flow rate on separation efficiency

4. SUMMARY

In this paper, the effect of inlet flow rate on the flow field and total separation efficiency of a micro-cyclone used for this purpose is investigated. The following conclusions were drawn:

(1) The change of inlet flow rate does not affect the overall "V" type pressure distribution law in the micro-cyclone, and the pressure value and pressure gradient in the micro-cyclone increase with the increase of inlet flow rate.

(2) Whether the axial velocity of the inner or outer cyclone increases with the increase of the inlet flow, and the axial velocity of the inner cyclone is larger than the axial velocity of the outer cyclone.

(3) The change of inlet flow does not affect the distribution law of tangential velocity along the radial direction which increases first and then decreases, and the tangential velocity generally increases with the increase of inlet flow.

(4) With the change of inlet flow, the turbulent kinetic energy varies greatly, and the regions with generally high turbulent kinetic energy are the bottom of the overflow opening and the vicinity of the bottom flow opening. The overall peak turbulent kinetic energy rises with increasing inlet flow.

(5) The total separation efficiency increases and then decreases slightly with the increase of inlet flow rate.

REFERENCES

[1] Growcock F, Harvey T. DRILLING FLUIDS[M]//Drilling Fluids Processing Handbook. 2003:15-68.

[2] Zijian Y, Yangsheng L. Research Progress of Water-based Drilling Solid Waste Treatment and Disposal Technology [J]. Environmental Engineering, 2021,39(10):143-149.

[3] Adekunle I M, Igbuku A O O, Shekwolo O O A P. Emerging Trend in Natural Resource Utilization for Bioremediation of Oil — Based Drilling Wastes in Nigeria[J]. Biodegradation - Engineering and Technology, 2013.

[4] Neff J M, McKelvie S, Ayers R C. Environmental Impacts Of Synthetic Based Drilling Fluids[R].2020.

[5] Zhang M, Li A, Zhu L, et al. A novel liquid-liquid cyclone reactor for ionic liquid catalyzed isobutane alkylation: Cold model investigation of the dispersed phase droplet size distribution[J]. SEPARATION AND PURIFICATION TECHNOLOGY, 2019,209:375-382.

[6] Yue T, Chen J, Song J, et al. Experimental and numerical study of Upper Swirling Liquid Film (USLF) among Gas-Liquid Cylindrical Cyclones (GLCC)[J]. CHEMICAL ENGINEERING JOURNAL, 2019,358:806-820.

[7] Fan Y, Wang J, Bai Z, et al. Experimental investigation of various inlet section angles in mini-hydrocyclones using particle imaging velocimetry[J]. Separation and Purification Technology, 2015,149:156-164.

[8] Chu K W, Wang B, Yu A B, et al. CFD-DEM modelling of multiphase flow in dense medium cyclones[J]. Powder Technology, 2009,193(3):235-247.

[9] Li F, Liu P, Yang X, et al. Effects of inlet concentration on the hydrocyclone separation performance with different inlet velocity[J]. Powder Technology, 2020,375:337-351.

[10] Yuan H, Fu S, Tan W, et al. Study on the hydrocyclonic separation of waste plastics with different density[J]. Waste Management, 2015,45:108-111.