Numerical investigation on the swirler parameters for an axial liquid-liquid hydrocyclone

Min Zhan¹, Xinping Cheng, Wanyou Yang, Fenghui Zhang, Peiwen Guo and Qiong Shen

Engineering Company, Energy Development Limited Company of CNOOC, Tianjin 300452 China

¹Corresponding author’s e-mail: zhanmin@cnooc.com.cn

Abstract. The exploitation of most oil fields has entered the middle and late stages and the proportion of water in the mixture extracted from the oil fields at this stage is very high, even reaching 80%-90%. In order to lower the cost of exploitation and make the oil field more sustainable, oil and water separation needs to be carried out downhole. The downhole space is small, and the separation device should be small and efficient. Therefore, this paper studies the axial flow oil-water separator that can be used for downhole oil-water separation. Study the influence of swirler structural parameters on the velocity field in the separator, and find structural parameters that are more beneficial to oil-water separation. The results show that increasing the outlet angle of the guide vane, the twisted diameter of the blade and the number of blades can all increase the tangential velocity of the flow field in the separation section, and generate centrifugal force to promote the aggregation of light fluid towards the center.

1. Introduction

Hydrocyclones are widely used in the field of petrochemical industry [1]. In order to reduce the proportion of water in the produced fluid, reduce the burden of process and the treatment link, it is selected to install hydrocyclones in the underground, through which the pre-separation of oil and water and the purification of oily water can be completed in the underground [2]. Thew et al [3-4] first applied cyclone separators in the field of oil-water separation in the 1980s and were favored by the petroleum industry. In the 1990s, our country introduced foreign advanced hydrocyclones and gradually realized independent research and development [5-6]. A novel axial centrifugal separator with multiple separation stage was proposed to separate oil-water liquids [7]. Daqing Oilfield Design Institute [5-6] designed single-cone and double-cone hydrocyclones. Through experimental comparison, it was found that the separation efficiency of double-cone hydrocyclone is better than that of single-cone hydrocyclone. Standard liquid-liquid hydrocyclone. Wang Zhenbo et al used Fluent to simulate the effect of guide vane structure on oil droplet breaking, and found that the main reason for oil droplet breaking is flow shear force and turbulent flow energy, and the drop probability of oil droplet breaking of guide vane hydrocyclone is lower. Ogawa [8] innovatively designed an axial flow cyclone separator. The results show that the separation efficiency of the axial flow cyclone separator is lower than that of the shear flow type at the same inlet flow rate, but its pressure loss is smaller than that of the tangential hydrocyclone. Wang Shuai et al [9] simulated the effect of different guide vane structures on the separation efficiency of axial flow liquid-liquid cyclone separator. The results show that the 45° straight plate guide vane structure has the best comprehensive efficiency and pressure
drop performance. And the experiment verifies the correctness of the simulation results. Shi Shiying [10] took the axial flow guide vane type liquid-liquid cyclone separator as the research object, and proposed a new type of cylinder structure. The main structure of the separator is a section of straight pipe with steady flow and a section of rectangular tangential slit in the cone-shaped dewatering pipe section, the guide vane of the separator is a straight guide vane. The results show that the multi-blade model obtains the best separation effect. Kataoka [11] studied the effect of different overflow pipe structures on the pressure drop through experiments, and found that the gap between the overflow pipe and the inner wall of the separator is an important factor affecting the pressure drop. Nieuwstadt [12] believes that reducing the diameter of the cylinder after the guide swirler can improve the separation efficiency. The flow field method was used to calculate the velocity field and the pressure field near the inner wall of the separator. By this method, the optimal structure of the separator was found to improve the separation efficiency.

The swirler is an important component in the separator, and it is also a component that plays a role to generate vortex in the separator. After the fluid passes through the swirler, it changes from axial motion to swirling motion, which directly affects the flow field distribution and separation efficiency in the separator. However, most of the current research focuses on changing the structure of the cylinder behind the guide swirler to improve the separation efficiency. There are few studies on the structure parameters of the guide swirler, so it is necessary to further study it. Numerical simulation method was used and single-phase fluid was used to study the effect of different swirler structure parameters on the velocity field in the separator.

2. Mathematical model

2.1. Governing equation

Mass conservation equation:

$$\frac{\partial \rho}{\partial t} + \rho \nabla \cdot V = 0 \quad (1)$$

Where $\rho$ is density, $t$ is time, and $V$ is speed.

The conservation of momentum embodies the application of Newton's second law in fluid element. The momentum equation is as follows:

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \quad (2)$$

$$\frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho f_y \quad (3)$$

$$\frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho f_z \quad (4)$$

Where $\tau_{xx}, \tau_{yy}, \tau_{zz}$ represent the components in the x, y, and z directions of the viscous force generated by the viscous fluid, and $f_x$ represents the volume force component of the fluid element.

The energy equation is as follows:

$$\frac{\partial (\rho T)}{\partial t} + \nabla \cdot (\rho u T) = \nabla \cdot \left( \frac{k}{c_p} \nabla T \right) + S_T \quad (5)$$

$$LL_t(dh'iu'j) + LL_kx(dU' u'iu'j) = Dij + pij + Cij - X$$

Where $c_p$ represents the fluid specific heat capacity, $T$ represents the fluid temperature, and $S_T$ represents the portion of the fluid element that is converted by the heat source and the fluid mechanical energy into thermal energy due to the viscous effect, referred to as the viscous dissipation term.
2.2. Turbulence model

Turbulence is a highly complex three-dimensional non-steady state, an irregular motion with rotation, a non-linear flow that is irregular in space and disorderly in time. In the calculation of turbulence, it is necessary to not only conform to the laws of conservation of mass, laws of conservation of momentum, and laws of conservation of energy, but also to comply with the turbulent transport equation. Therefore, the turbulence equation is used to better simulate the turbulent flow of the fluid. Dyakowski [13] coupled the turbulence model and the Reynolds stress equation to obtain a relatively perfect Reynolds stress model. Lu Yaojun et al [14] used three models: standard k-ε model, RNGk-ε model and Reynolds stress model to simulate the turbulent process. The results show that the simulation results of the Reynolds stress model are most consistent with the measured results. Dai Guangqing [15], Yang [16] and others also used the Reynolds stress model to verify the flow field distribution in the swirl field, so this paper also uses the Reynolds stress model as the turbulence model.

3. Numerical simulation

3.1. Physical model and meshing

This paper mainly studies the influence of swirler structure parameters on the flow field distribution, and uses the 3D modeling software to establish the geometric model as shown in Figure 1. The guide vane is fixed on the stationary hub, and at the same time, the blade itself does not rotate, and the flow channel generated by its torsion angle guides the fluid. The model is mainly composed of swirler, secondary phase outlet, barrel and main fluid inlet and outlet. The fluid flows into the cylinder from the inlet side to form a swirling field in the cylinder. The pressure in the pipe is higher than the outside atmospheric pressure. The fluid collected in the center of the pipe flows out from the secondary phase outlet, and the rest flows out from the main flow outlet.

In this paper, ICEM CFD software is used to mesh the computing domain, which is divided into hexahedral meshes, as shown in Figure 2. The grid quality is shown in Table 1. The grid density has a great influence on the results of numerical calculations. It only makes sense when the grid density reaches a certain level and the calculation results do not change much. Three kinds of grids with different densities and densities were used to verify grid independence. The number of divided grids was 470736, 604692, and 724221 respectively. Using numerical simulation software, the calculation is performed under the same calculation conditions, and the axial velocity distribution result is obtained on a diameter at the position of Z=3D after the guide vane exit as shown in Figure 3 below. According to the axial velocity distribution curve, it can be seen that only the axial velocity calculated by the high-density grid near the wall surface is slightly higher than that of the low-density grid. Therefore, it can be considered that the number of 470736 grids can control the error within a reasonable range.
Table 1. Grid quality distribution.

|                  | 0.4-0.6 | 0.6-0.8 | 0.8-1.0 |
|------------------|---------|---------|---------|
| Determinant 2×2×2| 0%      | 13.7%   | 86.3%   |
| Equiangle skewness| 6.9%    | 10.7%   | 82.4%   |

(a) Swirler grid  
(b) Section grid

Figure 2. Schematic diagram of axial flow guide vane cyclone structure.

Figure 3. Axial velocity distribution at different grid numbers of the same model at Z = 3D.

3.2. Boundary conditions and solution settings
In this paper, water is selected as the single-phase medium, the inlet boundary condition is the mass flow inlet, and the outlet boundary condition is the pressure outlet. The pressure set at the outlet is greater than the secondary phase outlet pressure to prevent backflow. The algorithm uses the separation algorithm SIMPLEC. The pressure discrete format uses PRESTO!, the momentum discrete
format uses the second-order upwind style, and the remaining difference formats use the QUICK format.

4. Analysis of simulation results
The tangential velocity of fluid is the key factor of two-phase separation. In the same case, the larger the tangential velocity is, the greater the centrifugal force can be generated, and the stronger the ability of light fluid to collect to the center, so as to promote two-phase separation. Therefore, the effects of the outlet angle, the twist angle and the number of blades on the velocity field in the separation chamber are studied by using the model mentioned above.

4.1. Influence of guide vane outlet angle
The guide vane outlet angle is one of the important factors affecting the separation efficiency, as shown in Figure 4 is the schematic diagram of the blade outlet angle, which is the angle formed by the tangent direction of the blade end and the axis in the same plane. Two guide vane outlet angles of 55° and 40° are selected to calculate its influence on velocity field distribution. Figure 5 shows the tangential velocity contrast of fluid at same cross section in swirl chamber under different guide vane outlet angles. It can be seen from the figure that the tangential velocity of the flow at the outlet angle of 55° guide vane is 26.7% (2.17m/s-2.75m/s, the same as the maximum tangential velocity). At the same time, the outlet angle of 55° guide vane with tangential velocity within 10 mm of the pipe radius is larger than that of 40° guide vane. This is because when the tangential velocity increases, a larger centrifugal force will be generated, which makes the fluid flow to the outside, and a larger vacuum area will be generated in the center of the pipe. The velocity attenuation of the fluid in the guide vane will reduce the tangential velocity. Therefore, increasing the outlet angle of the guide vane can obviously increase the tangential velocity of the fluid, generate more centrifugal force to make the light phase medium converge to the center, and promote the two-phase separation.

Figure 4. Schematic diagram of outlet angle.
4.2. The influence of guide vane twist angle

Analyze the swirler with 90° and 180° torsion angles respectively, as shown in Figure 6. The torsion angle is the angle that the blade rotates on the hub from the inlet to the outlet, and (a) is the blade with a twist angle of 180°. Figure 7 shows the distribution of the tangential velocity of the fluid at same cross section in swirl chamber under different guide vane torsion angles. It can be seen from the figure that the tangential velocity of fluid at 180° torsion angle is 15.5% (1.87m/s-2.16m/s, select the maximum tangential velocity) higher than that at 90° torsion angle. From the inlet to the outlet, the flow section decreases gradually, and reaches the minimum flow section at the outlet. Comparing the two swirlers, it can be seen that the minimum flow section at the outlet is smaller at the 180° torsion angle. At the same time, the 180° twist angle has a longer flow channel at the swirler, which has a better effect on the change of fluid velocity direction. Increasing the twist angle of the guide vane will reduce the flow cross section between the vanes, accelerate the flow, increase the tangential velocity, and generate more centrifugal force, so that the light phase fluid converges to the center, which is beneficial to separation.
4.3. The influence of the number of guide vane

When analyzing the influence of the number of guide vanes on the distribution of separation space flow field, the situation of 5 blades and 8 blades are simulated respectively, and the velocity distribution is shown in Figure 8. It can be seen from the figure that the tangential speed of 8 blades is 15.3% higher than that of 5 blades (2.74m/s-3.16m/s, select the maximum tangential speed). As the number of blades increases, the closer the distance between adjacent blades is, the narrower each channel is divided, the closer the fluid in the channel is to the solid wall, the closer the flow field distribution near the upstream side and the downstream side of adjacent blades in the same channel is, and the more uniform the flow distribution from each channel is. The centrifugal force produced by the increase of tangential velocity makes it easier for the light phase medium to aggregate to the center, thus promoting the separation of the two phases.

5. Conclusions

In this paper, the numerical simulation method is used to study the influence of different swirler structure parameters on the internal velocity field of an axial-flow centrifugal separator. From the
centrifugal point of view, the velocity field of various structures is analyzed to provide guidance for the optimal design of swirl structure. The following conclusions are drawn:

To a certain extent, increasing the swirler outlet angle, torsion angle and the number of blades can significantly improve the tangential velocity of the fluid in the flow field, so as to improve the centrifugal force to a certain extent, which is conducive to the polymerization of light fluid. However, with the increase of the tangential velocity in the flow field, it may increase the fragmentation of oil droplet particles, making the emulsification more serious, which is not conducive to separation.

Acknowledgements

Fund Project: CNOOC energy development key project "development of compact and efficient oil field output processing technology and equipment development" Project No. HLXMLZ-GJ2018-15.

References

[1] Jing Yuxiao 2010 Development of Axial Flow Vane-guide Cyclone Separator [D] Beijing:China University of Petroleum
[2] Sina Amini 2012 Mathematical modelling of a hydrocyclone for the down-hole oil-water separation (DOWS) [J] Chemical Engineering Research and Design 90(12) 2186-2195
[3] Svarovsky L 1984 Hydrocyclone [M] London: Rinehart and Winston Ltd, 4-12
[4] Thew M 1986 Hydrocyclone Redesign for Liquid-Liquid Separation [J] The Chemical Engineer 427 17-23
[5] Slack M D 2000 Advances in cyclone modelling using unstructured grids [J] Chemical Engineering Research and Design 78(8) 1098-1104
[6] Liu H F, Xu J Y, Wu Y X, et al. 2010 Numerical study on oil and water two-phase flow in a cylindrical cyclone[J] Journal of Hydrodynamics, Ser. B 22(5-supp-S1) 832-837
[7] Xiaobo Zeng, Le Zhao, Weiguang Zhao, Mingwen Hou, Fuqiang Zhu, Guangming Fan and Changqi Yan 2020 Industrial & Engineering Chemistry Research 59(48) 21177-21186
[8] Ogawa A, Arakawa M 2006 Control of Collection Efficiency for Axial Flow Cyclone Dust Collectors with Fixed Guide Vanes and with Funnel Shaped Exit Pipes [J] Journal of Thermal Science 15(3) 240-250
[9] Wang Shuai 2015 Phase-isolation of upward oil-water flow using centrifugal method [J] Flow Measurement and Instrumentation 46 33-43
[10] Shi Shiyi, Liu Min, Luo Changhua, et al. 2012 Preliminary Study about the Structure Optimization of a New Vane-Type of Pipe Separation for Oil-Water Separation[J] Shipbuilding of China 11(53) 271-278
[11] Kataoka H, Sou A, Hosokawa S 2009 Swirling Annular Flow in a Steam Separator[J]. Journal of Engineering for Gas Turbines and Power 131(3) 1-7
[12] Nieuwstadt F T M, Dirkzwager M A 1995 Fluid Mechanics Model for an Axial Cyclone Separator [J] Industrial and Engineering Chemistry Research 34(10) 3399-3404
[13] Dyakowski T, Willianms R A 1993 Modeling turbulent flow with-in a small-diameter hydrocyclone [J] Chemical Engineering Science 48(6) 1143-1152
[14] Lu Yaojun, Zhou lixing, Shen Xiong 2001 Different turbulence models for simulating a liquid-liquid hydrocyclone [J] Journal of Tsinghua University (Science and Technology) 41(2) 105-109
[15] Dai Guangqing, Li Jianming, Chen Wemni 1997 Numerical simulation of turbulent flow in a hydrocyclone [J] Journal of Chemical Industry and Engineering 48(1) 123-126
[16] Yang I H, et al. 2004 A three-dimensional simulation of a hydrocyclone for the sludge separation in water purifying plants and comparison with experimental data [J] Minerals Engineering 17 637-641