Structure Optimization of Gas-Liquid Separator Based on STAR CCM

Qiufeng Liang¹,*, Changrun Xiao¹a, Lu Zhang² and Senlin He²
¹School of ship and ocean, Naval Engineering University, Hubei, China
²Chinese People's Liberation Army 92474, Sanya, China

*Corresponding author e-mail: 1581158064@qq.com, a xiaochangrun@sina.com

Abstract. At present, most of the submarine air compressors at home and abroad are directly discharged into the cabin when they blow out the condensed water and oil, resulting in environmental pollution in the cabin and corrosion of surrounding parts. In order to solve this problem, this paper studies the condensate drain device of submarine air compressor. STAR CCM + based on CFD method is adopted to analyze the flow field of two-phase gas-liquid separator. The reliability of STAR CCM + numerical simulation is verified by comparing with the existing test data [1]. On this foundation, combined with submarine cabin space and condensate discharge characteristics of submarine air compressor, the structure of gas-liquid separator is optimized. Focus on the analysis of flow field distribution and gas-liquid separation effect in the case of different baffles and the presence or absence of baffles. The results show that when the inlet dryness is 0.7, the bottom baffle of the separator can make the internal flow field more stable; the dryness of separator gas outlet can reach 0.99 with four-turn spiral deflector and the bottom mounted baffle. The result of numerical simulation can provide a valuable reference for the design of the condensate drain device of submarine air compressor.

1. Introduction
The air compressor is an important auxiliary machine for the submarine. The equipment mainly provides high-pressure air within 25 MPa for the submarine support system. It is one of the core equipment in the submarine system and an important guarantee for the reliability, safety and vitality of the submarine. When the air compressor is working, the process of oil injection and compression make the compressed air mix with water and oil. After the air compressor stops working, there is still condensate water to be discharged. At this time, because the air compressor is still under high pressure, the mixture of air, oil and water is discharged into the air in the form of atomization. The existence of oil and water discharged into the air not only affects the service life of the compressor itself, the operation and service life of subsequent working parts, but also affects the working environment inside the submarine [2]. At present, most of the submarine air compressors at home and abroad are directly discharged into the cabin when they blow out the condensate and oil, without the installation of the discharge treatment device. Therefore, oil and water must be separated from the air by a certain method, and a good gas-liquid separation system can ensure the purity of the exhaust gas. Due to the limited indoor space of the submarine cabin, the structural size of the gas-liquid separator is relatively high. In this paper, a spiral separator with high separation efficiency, compact mechanical structure and small volume is selected as the research object [3]. Based on STAR CCM +, the structure of the
spiral separator is optimized, which provides a valuable reference for the design of the submersible air compressor condensate drain device.

2. Simulation model verification

According to the data in reference [1], the reliability of STAR CCM + numerical simulation is verified. The gas dryness is converted into the relative volume fraction of gas and liquid. STAR CCM + carries out the numerical simulation, and the simulation results are compared with the test results of the literature [1].

2.1. Establishment of physical model

The geometric model of calculation is shown in Fig. 2.1, which is mainly composed of three parts: inlet pipe section, spiral deflector pipe section and outlet pipe section. Structural parameters are shown in Table 1.

| Import diameter | Gas outlet diameter | Liquid outlet diameter | Cylinder diameter | Deflector height | Diameter of the spiral deflector |
|-----------------|--------------------|-----------------------|-------------------|-----------------|-------------------------------|
| D₁=12mm        | D₂=10mm            | D₃=12mm               | D=55mm            | H₂=112mm        | D₄=54mm                       |

The mixture of condensation water discharge of submarine air compressor flows into the gas-liquid separator from the horizontal inlet, and the two-phase flow rotates downward along the spiral deflector. Under the action of centrifugal force, the liquid phase with higher density is thrown to the inner wall of the cylinder and flows to the liquid outlet along the wall, while the gas with lower density rotates along the spiral deflector and finally flows to the gas outlet at the top.

2.2. Calculation area and grid division

The whole of fluid flow area in the separator is chosen as the calculation domain. Because the model studied in this paper has complex topological structure and complex boundary conditions, and there are certain forms of connection between the intakes pipe, the outlet pipe and the central pipe and the outer cylinder, so it is not suitable to use structured grid. It is suitable to use tetrahedral unstructured mesh with fast calculation, strong adaptability and accurate solution. The grid size of the main structure part is about 7.5cm, and the total number of grids is about 900000. The quality of the grid after division is more than 0.4. Cut off the finished grid division Figure 2.
2.3. Mathematical model and boundary conditions

2.3.1. Mathematical model

A large number of studies show that in practical applications, the accuracy of numerical simulation of two-phase separated flow mainly depends on the turbulence model, while the traditional linear turbulence model has limited ability to simulate separated flow. For a long time, many authors have proposed different nonlinear models to simulate separated flows to overcome the shortcomings of traditional linear turbulence models [4]. These studies lay the foundation for better numerical simulation of separated flow, but the problems of universality and computational cost seriously restrict its application to engineering. The SST k-ω Turbulence model proposed by mender [5] combines the advantages of k-ε model and k-ω model. The SST k-ω turbulence model avoids the assumption of isotropic eddy viscosity, and is suitable for the simulation of complex three-dimensional motion such as strong eddy current.

The gas-liquid separator studied in this paper belongs to the two-phase flow of liquid dilute phase flow. Therefore, the Euler multiphase flow is selected, the liquid is regarded as a discrete phase, and the gas is regarded as a continuous phase. In the case of high pressure and non-steady state, the flow field simulation is performed using the SST k-ω turbulence model, which considers the turbulent shear stress and does not over predict the eddy viscosity. Moreover, the SST k-ω turbulence model can automatically judge the convergence mode near the wall, which is convenient for the convergence of the calculation results. The continuous equation and momentum equation are solved by SIMPLE method, and the convection term is discretized by the first-order upwind scheme.

2.3.2. Calculation boundary conditions

1) Inlet boundary condition: The gas-liquid mixture inlet adopts the velocity inlet boundary condition, and it is considered that the gas-liquid mixture is a flow under ideal conditions, which can be regarded as an infinitely uniform flow. According to the actual conditions, the two-phase flow velocity and the volume fraction of each phase on the inlet boundary are given [7].

2) Outlet boundary conditions: the liquid phase and gas phase outlets use pressure outlet boundary conditions and consider as escape boundaries.

3) Boundary of spiral deflector: the boundary of the spiral deflector is defined as a slip free boundary, because there is no heat exchange problem in the fluid assumption of the spiral structure, the thermal boundary condition is not considered here.

4) Wall fixing conditions: the outer wall of the spiral deflector is a solid wall without slip and mass addition. The turbulent parameters near the wall are determined by the wall function method.
2.4. Result analysis

Figure 3. Comparison of experimental results and numerical simulation results

Figure 3 shows the comparison between the results of STAR CCM+ numerical simulation and the experimental data in reference [1] under the condition of different intake air and liquid volume ratio. It can be seen from the figure 3 that with the increase of the volume fraction of the inlet gas, the separation efficiency is better. When the volume fraction of the inlet gas is 0.33, the simulation results are closest to the experimental results. Overall, the trend of the simulation results and the experimental results are the same, and within the error range, which verifies the reliability of STAR CCM+ numerical simulation.

3. Structure optimization of gas-liquid separator

According to the analysis of the above model, the result of STAR CCM+ numerical simulation is reliable. Next, according to the requirements of submarine cabin space, this paper will establish a suitable model (structural parameters are shown in Table 2). Carry out the numerical analysis of whether the gas-liquid separator has bottom baffle and the number of different spiral guide vanes, optimize the structure of the gas-liquid separator, predict the performance of the gas-liquid separator, and provide valuable reference for the design of the condensate drain device of submarine air compressor. In order to simplify the calculation, oil drop is not considered in this paper. Water is regarded as liquid phase and gas as gas phase. Main parameters are shown in Table 3.

Table 2. Main parameters of the model structure

| Model | Cylinder diameter | Import diameter | Gas outlet diameter | Liquid outlet diameter | Number of spiral deflectors | Baffle |
|-------|------------------|----------------|---------------------|-----------------------|----------------------------|--------|
| a     | 140mm            | 50mm           | 50mm                | 15mm                  | 3                          | Yes    |
| b     | 140mm            | 50mm           | 50mm                | 15mm                  | 4                          | Yes    |
| c     | 140mm            | 50mm           | 50mm                | 15mm                  | 5                          | Yes    |
| d     | 140mm            | 50mm           | 50mm                | 15mm                  | 4                          | No     |

Table 3. Main parameters of each phase

| Phase | Inlet speed | Phase volume fraction | Phase density | Absolute pressure |
|-------|-------------|-----------------------|---------------|-------------------|
| gas   | 10m/s       | 0.7                   | 81.621 k/m³   | 2.5Mpa            |
| liquid| 10m/s       | 0.3                   | 1018.1 k/m³   | 2.5MPa            |
3.1. The effect of the number of spiral guide vanes on separation efficiency

The purpose of the baffle at the bottom of the separator (sometimes called the vortex elimination plate or the vortex stabilizing plate) is to provide a contact surface on which the end of the vortex rotates. The purpose is not to destroy and disturb the vortex, but to avoid the direct contact between the vortex and the outlet of the liquid and to entrain the liquid drop twice. We selected model b and model d for numerical simulation of unsteady state, and analyzed the effect of bottom baffle on separation efficiency. The time step is 0.001 s. The trajectory flow chart after 15 s is as follows:

![Figure 4. streamline diagram of model b](image)
![Figure 5. streamline diagram of model d](image)

![Figure 6. Gas outlet gas volume fraction](image)
![Figure 7. Liquid outlet liquid volume fraction](image)

It can be seen from the trace streamline diagram that the two-phases flow enter the separator at the same speed, and the liquid phase with high density directly collides with the wall directly opposite the inlet pipe due to the inertial effect, forming the first gas-liquid separation. It can be seen from Fig. 3.4 that most of the liquid in the first 7 seconds adhered to the wall and did not flow to the liquid outlet. After 7 seconds, the liquid flows along the wall to the bottom liquid outlet, and the liquid volume fraction increases and tends to be stable. In model b, because of existence of the baffle, the end of the rotating airflow directly hits the baffle, and the airflow changes direction to the top outlet. There is no baffle in the model d. under the action of inertia; the end of the rotating airflow continues to flow downward. One part of the airflow hits the bottom sidewall and the liquid drop attached to the bottom sidewall will flow to the top outlet for secondary clamping, and the other part of the gas will flow directly to the liquid outlet. By combining the streamline diagram and volume fraction diagram, it can be concluded that the internal flow field of gas-liquid separator with baffle at the bottom is more orderly and the separation effect is better than that without baffle at the bottom.

3.2. The effect of the number of Spiral deflector on separation efficiency

The spiral guide vane changes the gas-liquid mixture in the incoming flow into a rotary motion, which is conducive to the downward inclined spiral motion of the fluid [7]. Therefore, the interference between the two adjacent spirals is avoided, and the gas with small density flows inside the spiral, while the small liquid particles with large density are thrown to the cylinder wall due to centrifugal
force, which accelerates the gas-liquid separation. This is a simultaneous axial and tangential motion, a special flow of vortices and streamlines that is directly related to the number of spiral deflector.

Therefore, during the comparative experiment, other conditions were controlled, and only the number of gas-liquid separator deflectors was changed to perform the non-steady-state numerical simulation. The simulation results are shown in the figure.

![Figure 8. Streamline diagram of different number of deflectors](image)

![Figure 9. Gas volume fraction at gas outlet](image)

![Figure 10. Liquid volume fraction at liquid outlet](image)

By analyzing the numerical simulation results of gas-liquid separators with different numbers of baffles, it is found that the number of baffles has a great influence on the separation effect and internal flow field. It can be seen from Figure 8 that within the scope of the study, as the number of baffles continues to increase, the gas-liquid separation efficiency continues to increase. The flow field is most ordered when the deflector is increased to 4 turns, and the separation effect begins to decline when the number of guide deflectors increases to 5 turns, which means that the number of baffles is not as good as possible. At the same time, the volume fractions of Figure 9 and Figure 10 are analyzed. The separation of model c is best in the first 7 seconds. This is because model c has 5 turns of spiral deflector. The spiral path of gas-liquid mixture is longer, and the gas-liquid separation is more thorough. However, too many spiral turns affect the stability of the flow field, causing the gas to entrain the droplets on the inner wall surface twice. After the flow field is stable, the separation effect of model B is the best, and the volume fraction of outlet gas can reach 0.99, which shows that the structure of model B is the best in the research scope, and this structure can provide valuable reference for the design of the condensate drain device of submarine air compressor.
4. Conclusions and Prospect

4.1. Numerical simulation conclusion of gas-liquid separator

a) CFD software is used to simulate the flow field of different structures of spiral deflector gas-liquid separator. The theoretical data and simulation results are analyzed, and the following conclusions are obtained.

b) Compared the data of literature [1] with the numerical simulation results, it is proved that the use of CFD software to simulate the separator has certain reliability, which can reduce the difficulty in the mechanical design process and provide reference for engineering design.

c) In the process of optimizing the gas-liquid separator, it is found that the baffle at the bottom of the separator has better separation effect than the other, and the flow field is more stable; the increase of the number of spiral turns will make the separation effect better, but it's not that the more laps the better.

d) When the intake speed is 10m/s, the number of spiral turns is 4 turns, and the bottom has a baffle, the separation effect is the best and the internal flow field is the most stable. Condensate drain device of submarine air compressor can be designed and optimized with reference to this conclusion.

4.2. Prospect of structure optimization of gas-liquid separator

In this paper, aiming at the actual problem of condensate drainage of submarine air compressor, a gas-liquid separator is added. STAR CCM + software simulate referring to the existing test data in reference [1], the prototype of the gas-liquid separator, which verifies the reliability of the software. Based on the CFD technology, the structural optimization of the gas-liquid separator is carried out in the same way to find an excellent and suitable structural form. After that, we can use the same method to study the condensate drain device of submarine air compressor more detailed (such as changing the diameter of the separator inlet, changing the position of the inlet and outlet, etc.), so as to get the gas-liquid separator with more optimized structural characteristics.

5. Acknowledgments

This work received financial and technical support from the “Research on Submarine Air Compressor Condensate Release Device” project.

References

[1] Lü Jiaming, Ye Qizhen, Chen Jiangping. Optimal Design of Cyclone Separator Based on Computational Fluid Dynamics Model [J]. Journal of Refrigeration, 2010(3):11-15.
[2] Liu Xiaozhou. Research on Compressor Oil and Gas Cyclone Separator [D]. Nanjing University of Science and Technology, 2005.
[3] Jiang Minghu, Xu Baorui, Zhao Lixin. Application and comparison of turbulence model in spiral flow field simulation of circular tube [J]. Chemical Engineering, 2016, 44(9).
[4] Abid R, Rumsey C, Gatski T. Prediction of nonequilibrium turbulent flows with explicit algebraic stress models [J]. Aiaa Journal, 1995, 33(11): 2026-2031.
[5] [Menter, F. R. Two-equation eddy-viscosity turbulence models for engineering applications [J]. AIAA Journal, 1994, 32(8): 1598-1605.
[6] Meng Lingli. Simulation and experimental study of flow field in spiral gas-liquid separator [D]. Daqing Petroleum Institute, 2010.
[7] Liu Wei, Happiness Hall. Structural Optimization of New High Pressure Oil-Water Separator [J]. Mining Research and Development, 2015(1).