ISSN 2616-5767 Vol.4, Issue 7: 37-43, DOI: 10.25236/AJETS.2021.040707

Design and Simulation of Two-Stage Valve Core of Pure Water Medium Safety Valve

Xinjian Zhou[a], Zhouda Hou[b,*], Hongbing Zhang[c], Linhai Ding

Xi'an University of Science and Technology, Yanta Middle Road, Xi'an, China
[a]893393465@qq.com, [b]houzhouda@163.com, [c]178299628@qq.com
**Zhouda Hou*

Abstract: *Aiming at the phenomenon that the pure water medium safety valve of coal mine support is easy to produce cavitation in high pressure operation. A kind of secondary throttle valve structure is designed and optimized to achieve the effect of restraining the formation of water vapor bubbles. Firstly, the mathematical model of vapor bubble is obtained through theoretical analysis, and the valve core model is optimized in three ways. Their respective flow field models are established and imported into Fluent for simulation calculation, and the performance of the three anti cavitation is analyzed. The results show that: by adding a high-pressure orifice channel to the valve seat, the pressure compensation of the orifice channel can effectively reduce the original serious cavitation area. Therefore, adding a high-pressure orifice in the appropriate position of the valve core or valve seat can effectively improve the pressure distribution and inhibit the occurrence of cavitation.*

Keywords: *Water medium safety valve,CFD simulation,design of valve core*

1. Introduction

Hydraulic transmission has many advantages. It uses transmission medium to achieve energy transmission, and it has a very wide range of applications in coal mines. However, if the hydraulic system is used at high frequency for a long time, leakage will inevitably occur.Hydraulic oil leakage will have an irreversible impact on the underground ecological environment, so the research on pure water hydraulic transmission is carried out, but the serious cavitation phenomenon caused by the replacement of transmission medium with pure water. It is one of the main problems of hydraulic components in water medium[1-2].In this paper, the cavitation phenomenon of pure water hydraulic valve core is theoretically analyzed and simulated, and the performance of two-stage heteromorphic valve core structure in cavitation resistance is designed and discussed.

2. Cavitation Phenomenon Analysis of Hydraulic Valve Core in Pure Water Medium

2.1. Formation of Cavitation

Cavitation is one of the main factors that endanger the hydraulic components of pure water medium. When the transmission medium flows through the vicinity of the throttle orifice of the hydraulic valve, the sudden change of the flow passage area will cause the sudden change of the flow velocity and pressure of the transmission medium. When the liquid pressure is lower than the saturated vapor pressure at the temperature of the liquid, the liquid will boil, resulting in a large number of bubbles.The generated bubbles will be carried by the transmission medium along the pipeline flow. When the bubbles flow to the high pressure area, the bubbles will collapse under the high pressure around them. At the moment of the bubble collapse, the vacuum will be formed in the area occupied by the bubbles themselves, and then the high-pressure water will influx from around to fill the vacuum formed by the bubble collapse, thus forming a severe local shock and high temperature and high pressure in a moment[3].

2.2. Mathematical Model of Cavitation

The liquid inside the hydraulic valve is reduced when it passes through the region near the throttle orifice. When the pressure is reduced to saturated vapor pressure and continues for a period of time,

ISSN 2616-5767 Vol.4, Issue 7: 37-43, DOI: 10.25236/AJETS.2021.040707

bubbles begin to form and develop.The saturated vapor pressure p_v is related to the temperature of the transmission medium.The literature shows that the working temperature of the hydraulic components of the water medium is 5~50 ℃ (more than 50 ℃ is prone to water dirt), and the p_v is 0.0125 MPa at 50 ℃[4].

The gas phase volume ratio equation is as follows:

$$\frac{\partial}{\partial t}(\alpha_a)+\frac{\partial}{\partial x_i}(\alpha_a \mu_i , a_v)=\frac{3\rho_a \alpha_a}{\rho_a R}\sqrt{\frac{2(p_v - p)}{3\rho_l}} \tag{1}$$

$$R=(\frac{3\alpha_a}{4\pi m})^{\frac{1}{3}} \tag{2}$$

In the formula, a is the bubble phase; l is the liquid phase, α_a is the bubble volume fraction, ρ_a is the gas phase density, ρ_l is the liquid density, R is the bubble radius, p_v is the vaporization pressure of water.

3. Calculation and CFD Simulation of Geometric Parameters of Safety Valve Core

3.1. Geometric Parameter Design

Safety valve is an important safety guarantee for mining hydraulic support, which ensures the smooth operation of the hydraulic support under the regulated pressure. When the pure water safety valve is opened, a local vacuum is formed at the valve core, resulting in cavitation damage[5].

The main dimensions that determine the anti-corrosion effect of water medium hydraulic valve are: inlet diameter, valve core diameter, valve seat aperture. Hydraulic valve design parameters: working pressure 31.5 MPa, nominal flow rate 100L/min.

$$v_s=\frac{4Q_s}{\pi D_0{}^2}\le[v_s] \tag{3}$$

In the formula, v_s is the hydraulic valve inlet velocity (m/s), $[v_s]$ is the permitted import speed, Q_s is the hydraulic valve nominal flow (L/min).

Since the liquid flow state in engineering is turbulent, the flow rate ratio of water medium to oil medium under the same conditions is as follows[6]:

$$\frac{Q_{\text{water}}}{Q_{\text{oil}}}=[(\frac{\mu_{\text{water}}}{\mu_{\text{oil}}})^{0.75}\times(\frac{\rho_{\text{water}}}{\rho_{\text{oil}}})^{0.75}]^{\frac{1}{1.75}}\approx 1.55 \tag{4}$$

In the formula, μ_{water} is the motion viscosity coefficient of water. Because the allowable inlet velocity of oil medium is 2 ~ 6 m/s, according to formula (4), $[v_{\text{water}}]$=3.1~9.3 m/s ,the allowable inlet velocity of water medium hydraulic valve is 9 m/s.

The inlet diameter D_0 is calculated as follows:

$$D_0\ge 1.13\sqrt{\frac{Q_s}{[v_s]}} \tag{5}$$

It is known that D_0 more than 14.5 mm, inlet diameter 15 mm. In this design, the valve core is a two-stage throttling structure, and the valve seat aperture D_1 is 24 mm. The valve core diameter D_2 is calculated as follows:

$$D_2=D_1\times\sqrt{\frac{A_2}{A_1}} \tag{6}$$

Where A_1 and A_2 are the orifice area of valve seat and the left section area of valve core, respectively. According to the data, A_1 / A_2= 0.95 ~ 0.98, and D_2=24.24 ~ 24.62 mm, which is taken as 24.5 mm. The design schematic is shown in Figure.1.

ISSN 2616-5767 Vol.4, Issue 7: 37-43, DOI: 10.25236/AJETS.2021.040707

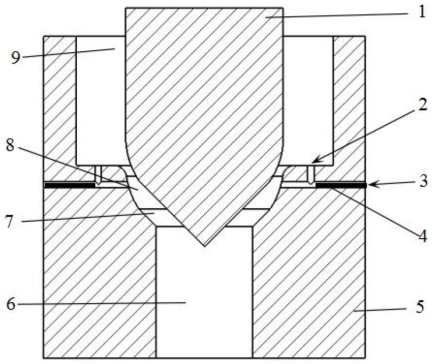
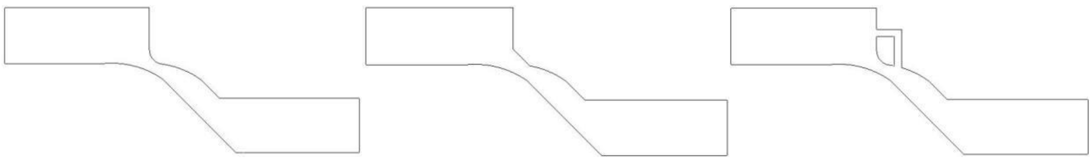


Figure 1: Valve core and seat coordination diagram.

As shown in Figure.1, part 1 is the valve core, position 2 is the valve seat hole A, position 3 is the valve seat hole B, part 4 is the plug, part 5 is the valve seat, position 6 is the water inlet, position 7 is the throttle 1, position 8 is the throttle 2, position 9 is the water outlet. The lower end throttle 1 is formed by conical core and seat, the upper end throttle 2 is formed by ball valve structure and seat. According to the literature, the sealing form of the ball valve and the cone valve is very suitable for the hydraulic system of water medium.

3.2. Analysis of Simulation Scheme

As shown in Fig. 1, the lower end of the safety valve is the inlet, and the upper end is the outlet. The fluid part used for CFD analysis is plotted as shown in Figure.2.



(a) (b) (c)

Figure 2: Flow field region of three valve cores.

The opening of the valve port is set to 3.5 mm, and the cone angle of the first-order cone valve is 45°. As shown in Figure. 1, when the valve core is opened, the street throttling port 2 formed by the valve core and the valve seat has a trend of narrowing. Therefore, the valve seat is treated as follows:

- As shown in Figure.2(a) the fillet treatment of the upper end of the valve seat at the throttle orifice and the fillet radius is 2 mm.
- The chamfering distance is also 2 mm, as shown in Figure.2(b).
- As shown in Figure.2(c), on the basis of adopting the fillet on the edge of the valve seat, the high pressure throttle orifice is drilled on the valve seat, and the diameter is 1 mm. The drilling location on valve seat is shown in Fig. 1 position 2 and position 3. After drilling at position B, the plug is used to block from the outside of the valve seat to avoid the leakage of liquid after the valve core is opened. The length of plug length is 5 mm.

Since the liquid flow field region at the valve core is a two-dimensional axisymmetric structure, half of the model is taken for analysis, so that the accuracy of the three-dimensional model can be achieved under the premise of saving computer resources.In this paper, the Fluent module in ANSYS19.0 is used for flow field simulation. After the establishment of the two-dimensional model, using grid module partition, and the grid processor is set to a fluid mode. The size function adopts the adaptive function, and the mesh size is set to 0.1 mm. The final generated mesh is imported into Fluent for inspection, showing that the mesh without negative volume indicates that the mesh is successfully divided.

By analyzing the flow field at the valve core, it can be seen that the water medium generates a large number of vapor bubbles because the pressure is reduced to below the saturated vapor pressure through the throttle orifice. Therefore, from water to vapor bubbles is a phase change rather than a simple external gas mixed into the water medium.Therefore, the Mixture calculation model in Fluent multiphase flow is selected, and the Mixture model is suitable for simulating the mutual transformation

ISSN 2616-5767 Vol.4, Issue 7: 37-43, DOI: 10.25236/AJETS.2021.040707

between phases (water phase and vapor phase). And activation of phase transition options cavitation, using the default cavitation model. Realizable K-ε turbulence model is used in turbulence model, which has better convergence and is suitable for coupling solution[7-8].

Pressure inlet and pressure outlet are used at the inlet and outlet respectively. The inlet pressure is set to 31.5 MPa, the outlet pressure is atmospheric pressure, and the wall is non-slip wall. The solver uses the Coupled algorithm to calculate and solve 5000 times.

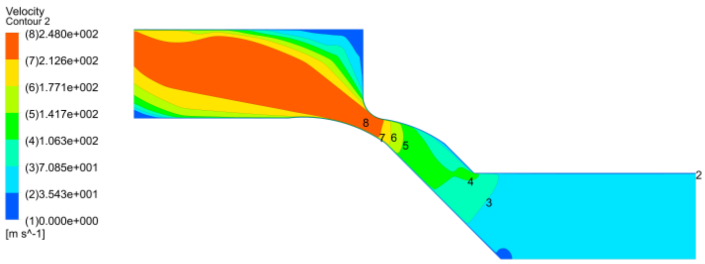
4. Analysis of CFD Simulation Results

The destination file is imported into the CFD-POST after Fluent19.0 calculation for reprocessing analysis. The three models of this analysis, the inlet diameter, valve core cone angle, valve opening are all the same.Analysis of the valve internal velocity field, pressure field distribution, vapor volume distribution can show the valve cavitation resistance performance[9-10].

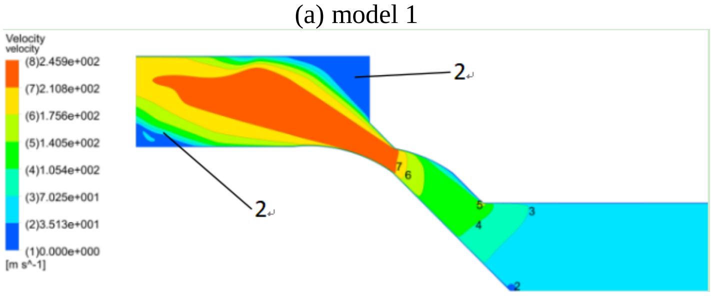
4.1. Velocity Field Analysis of Valve Core Area

Three kinds of valve core velocity distribution contour lines as shown in Figure.3, the greater the number on the contour line (1~8) represents the higher the velocity value. Compared with the three models, it can be seen that the maximum velocity at the outlet is not much different but the velocity distribution near the throttle mouth in the valve core is significantly different. It can be seen from Figure.3(a) and Figure.3(b) that there is no high-pressure throttling port on the valve seats . Therefore, the velocity from the inlet to the outlet show an increasing trend. This is why the water medium passes through the two throttling ports, and the sudden narrowing of the valve core flow channel leads to a sharp increase in the flow velocity of liquid pressure.

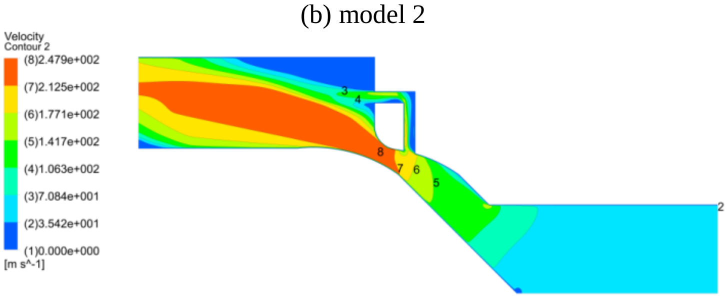
The analysis Figure.3(b) shows that due to the chamfering treatment of the valve seat, the water flows along the direction of 45° chamfering when passing, resulting in local vacuum at area 2. Compared with the same position in Figure.3(a), since the valve seat is filleted, the local vacuum area generated by the flow passing is significantly less than that generated by the chamfering.



(a) model 1



(b) model 2



(c) model 3

Figure 3: Velocity distribution of three models.

ISSN 2616-5767 Vol.4, Issue 7: 37-43, DOI: 10.25236/AJETS.2021.040707

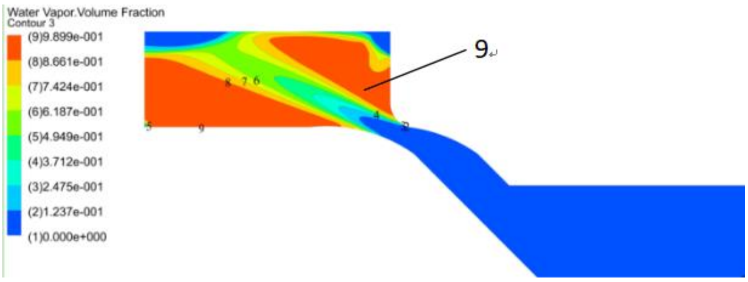
It can be seen from Figure.3 (c) that by drilling a 1 mm throttling channel on the valve seat, the fluid between the valve core and the valve seat can be drawn into a branch, so as to fill the vacuum area generated by the flow out of the chamfer of the valve seat. Due to the small aperture of throttling channel, the water velocity in the channel is only half of the maximum velocity.

By comparing the velocity flow fields in the three models, it can be seen that the valve seat treated by chamfering is obviously worse than that of the other two models, and the vacuum area generated in the flow field inside the valve core model with high pressure throttling channel is the least.

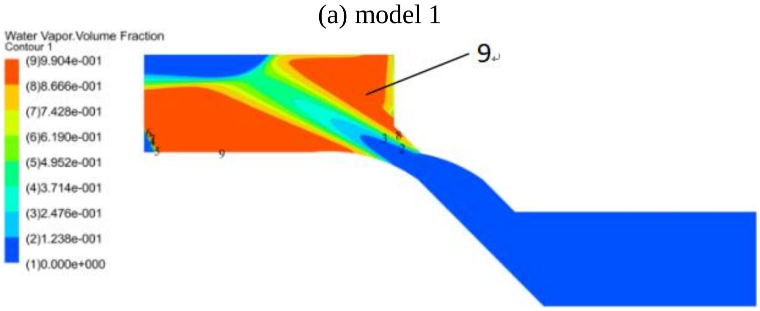
4.2. Analysis of Pressure and Vapor Distribution in Valve Core

The volume distribution of water vapor in the flow field of the valve core is shown in Figure.4. The larger the equivalent line number, the higher the volume fraction of water vapor.From the post-treatment results, the volume fraction of water vapor is zero from the inlet of the valve to the throttle of the valve core, so the analysis is focused on the throttle formed by the valve core and the seat and near its outlet.

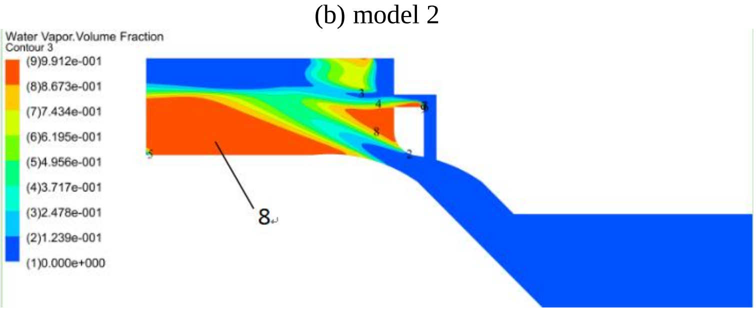
As shown in Figure.4(a) and (b), the water pressure decreases when the flow passes through the throttle, when flowing out of the orifice the area of the flow passage suddenly increases, so it is easy to form a negative pressure area on both sides of the flow ejected from the valve core. As shown in area 9 above the two figures, this area has the largest volume fraction of water vapor. Area 9 is the upper part of the valve core and the seat port respectively. The upper part of the core does not participate in the sealing of the valve, but if the seat is eroded by bubbles for a long time and high strength, the safety valve will gradually fail. In addition, the flow pressure will decrease gradually in the process of gradually away from the throttle, so the volume fraction of water vapor is also increasing in this part of the flow. There is an area where the volume fraction of water vapor is 0 in the upper left corner of the three models. Comparing the area of the three upper left corner positions, it is found that the inner area of model 3 is the largest and the inner area of model 1 is the smallest. It can be seen that the pressure variation gradient inside the model 3 is smaller than the other two.



(a) model 1



(b) model 2



(c) model 3

Figure 4: Three kinds of valve core water vapor distribution.

Analysis Figure.4 (c) shows that a small part of liquid is drained from the valve core and seat to compensate the pressure in the area where the cavitation is serious. It can be seen that the largest volume fraction of water vapor is in area 8. The area 8 is divided into left and right parts on the valve core and high-pressure section outlet. Comparing the same position of model 1 and 2, it is found that the water vapor on the right side of model 3 is significantly less than that of the other two models. The area of area 9 on the right side of model 1 and 2 is almost the same, and the volume of water vapor through the high-pressure throttling flow channel is reduced to 1/3. By comparing the left part of area 8 in model 3, it can be found that the area of water vapor also decreases significantly, but this part belongs to the upper end of the valve core, so cavitation damage will not affect the overall performance of the valve.

The pressure changes of the three models from the inlet to the outlet are shown in Figure.5. In the initial stage, there is no significant difference in the internal pressure changes of the three models, but there is a significant difference in the pressure changes through the valve core. The comparison between model 1 and model 2 shows that the pressure decline trend is basically the same, but the negative pressure appears at the outlet and the pressure is lower than the saturated vapor pressure value of water. The model 1 outlet pressure is -6.775 MPa, and the model 2 outlet pressure is -2.955 MPa. However, in model 3, there is an orifice inside the valve core and valve seat to share part of the water flow, and the water flow velocity at the valve core decreases. The final outlet pressure is 0.5195 MPa higher than the saturated vapor pressure of water.

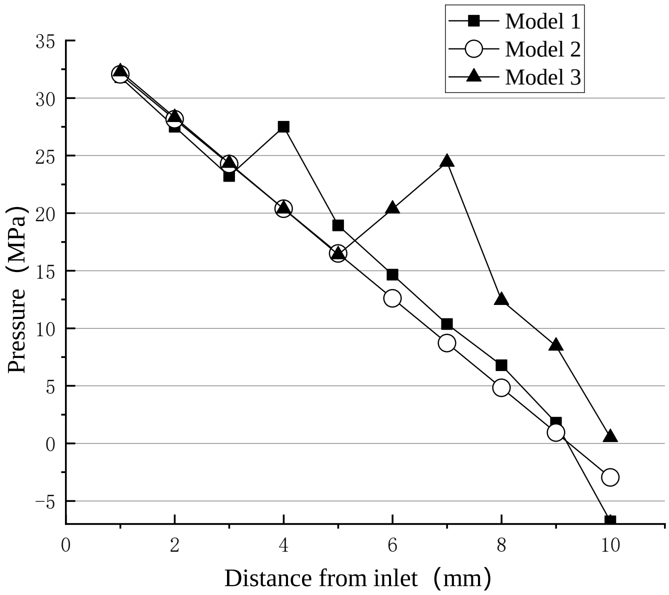


Figure 5: Pressure distribution inside three spools.

5. Conclusions

Through the CFD simulation analysis of three kinds of valve core structure, the following conclusions can be drawn.

- The secondary valve core is better than the primary valve core in terms of cavitation resistance, because the excessive pressure change is borne by two parts, but the actual processing difficulty of the secondary valve seat is relatively large. The convenience of actual use should be considered in the design.
- Comparing the three models, it is not difficult to see that the model with high pressure throttle channel performs best in uniform cavitation. Because the pressure compensation of the high pressure throttle port greatly reduces the area where the original cavitation phenomenon is serious. The second is the chamfering model of the seat, because the model is easy to process in practice, and the water vapor ratio inside the simulation model 2 is more than the model 3 but less than the model 1. Therefore, considering comprehensively, the model with high pressure throttling channel is the best, followed by the model with valve seat chamfer, and the worst is the valve seat fillet model.

- The premise of this simulation is that the outlet pressure is atmospheric pressure. If a back pressure is provided to the valve in practice, the pressure gradient inside the valve core will be reduced, thus reducing the cavitation effect.

Acknowledgments

This work was financially supported by the National Science Basic Research Project of Shanxi province of China-Joint Fund Project (2019JLM-50) fund.

References

[1] Research on pure water safety valve in Lihong hydraulic support [J].Coal mine machinery, 2008,29 (003): 123-124.
[2] Zhou Mu. Experimental Study on Flow Field Analysis and Static and Dynamic Performance of Pure Water Overflow Valve [D]. China University of Mining and Technology ,2015.
[3] Zheng Zhijian, Wang Yangding, Shen Yumei. Numerical Study on High Pressure Cavitation Flow Characteristics of Hydraulic Sliding Valve [J]. and Hydraulic and Pneumatic ,2020,(005):75-79.
[4] Shi Gaoliang. Design and experimental study of large flow hydraulic unidirectional valve [D]. China University of Mining and Technology, 2015.
[5] Han Ming-xing, Liu Yin-shui, Wu De-fa, et al. A numerical investigation in characteristics of flow force under cavitation state inside the water hydraulic poppet valves[J]. International Journal of Heat and Mass Transfer, 2017, 111: 1-16.
[6] Fang Jupeng, Li Qiang, Qupu, etc. Three-dimensional flow simulation of three-way control valve based on Fluent [J]. Coal mine machinery, 2012 (03) : 84-85.
[7] Zhang Xincheng, Meng Fanhua, Shang Huiling.Safety and reliability of hydraulic support safety valve [J].Coal mine machinery, 2010,31 (1) : 76-78.
[8] Wang Cunfei, Fang Zhongxi, Du Qi, etc.Simulation of working characteristics of pure water pump for pure water support [J].Coal mine machinery, 2019,377 (07) : 181-184.
[9] Jin Wei and Shi Junqiang. Flow field analysis of rotary valve orifice based on Fluent [J]. Hydraulic and pneumatic, 2020, (005) : 162 – 166.
[10] Li Zhi.Research on pure water medium safety valve of hydraulic support and its test bench [D]. Xi'an University of Science and Technology, 2020.