Flow Field Analysis of Hydraulic Multi-way Valve Based on CFD

Chunlei Luo\(^{1,a}\), Jinyang Li\(^{1,b}\), Danjie Liu and Tao Wang

School of Central South University, Changsha 410000, China

\(^a\)1595583961@qq.com, \(^b\)214388492@qq.com

Abstract. Referring to the structure and parameters of the actual multi-way valve, the CAD software SOLIDWORKS was used to establish the three-dimensional geometric model of the internal flow passage of the hydraulic multi-way valve, and the model was calculated and analyzed by CFD software Star CCM+. By studying the flow field distribution in the valve chamber, the pressure distribution at each monitoring point in the chamber and the change of the overall flow rate were obtained, and the steady-state hydrodynamic force of the valve core was calculated. The effects of flow rate and the opening of the valve core on the pressure distribution and steady-state hydrodynamic force in the valve chamber were analyzed under various working conditions. The results show that the steady-state hydraulic force on the valve core increases with the increase of flow rate and decreases with the increase of valve opening. The pressure loss at the inlet and outlet of the slide valve is mainly caused by the throttling characteristics of the oil at the throttle port. The smaller the opening, the greater the inlet pressure and the greater the pressure at all points in the valve chamber.

1. Introduction

Hydraulic multiple valves are widely used in hydraulic systems as hydraulic control valves in fluid transmission and hydraulic control. Its main function is to control the movement state of the actuator. The main performance of the hydraulic multi-way valve is stable commutation, small impact and fast response. Its performance plays a vital role in the stability of the whole hydraulic system. At present, more and more domestic and foreign scholars use the computational fluid dynamics method (CFD) to visually analyze the flow field inside the hydraulic valve, which mainly studies the velocity distribution, pressure change, energy change of the fluid in the valve. Cavitation at the valve port [1, 2, 3]. The application of CFD software overcomes the shortcomings of traditional theoretical calculation, can well guide the experimental research, and can also simulate the ideal conditions which can only be approached but cannot be achieved.

In this paper, the CFD analysis of the flow field inside the valve cavity of a non-full-opening hydraulic multi-channel valve is conducted to study the flow field distribution of the valve cavity, obtain the pressure distribution of each monitoring point in the cavity and the change of the entire flow velocity, and calculate the steady-state hydraulic force suffered by the valve core through calculation. The effects of flow rate and core opening on pressure distribution and steady-state hydrodynamic force in the valve chamber were analyzed under various working conditions. It provides important theoretical significance for the design of high efficiency, low energy consumption and low noise hydraulic multi-way valve.
2. Modeling and meshing

2.1. Modeling with SOLIDWORKS
Some simplifications of the model before modeling: the multi-way valve is an ideal multi-way valve, that is, the valve core and the valve body do not have radial clearance, the fitting is accurate, and there is no manufacturing error. Firstly, the fluid area without spool (only the surface of oil passage directly contacted with hydraulic oil is considered, and the remaining surface is deleted.) and the entity of spool are established respectively, and then the assembly is carried out. The assembly diagram is shown in Fig. 1.

![Figure 1. Simplified assembly](image1)

2.2. Using STAR CCM+ to divide grids
After introducing the spool and simplified valve body into STAR-CCM+ simulation software, the wrapping surface of the model is processed by using its unique wrapping technology. The wrapping surface can generate a closed surface mesh, which is convenient for the subsequent body mesh generation.

Due to the complexity of the internal flow passage of the valve and the influence of the orifice on the flow field, the grid is refined according to the geometric characteristics of the valve. The flow at the inlet and outlet is relatively stable, so the grid at the inlet and outlet is relatively thick. As shown in Fig. 2, the model is divided by a tetrahedral mesh, and the model is divided into 705,965 cells.

![Figure 2. Hydraulic valve grid model](image2)

3. CFD simulation calculation settings
In order to satisfy the feasibility of numerical analysis, the following basic assumptions are made for the actual model [4]:
(1) Suppose that the multi-way valve studied is an ideal valve, that is, the spool and valve body cooperate accurately without radial clearance;
(2) The fluid is incompressible and constant Newtonian fluid.
(3) Because the size of the valve body is very small, the influence of fluid gravity and fluid heat transfer in the valve chamber is neglected.
(4) Turbulence is the flow state, and standard K-E turbulence model is selected.

46# hydraulic oil is selected in this paper. Assuming that the lubricating oil is an incompressible Newtonian fluid, the oil density is 870.0 kg/m³, and its dynamic viscosity is 0.0261 pa-s when the operating temperature is 60°C.

Combined with the actual working conditions of the experiment and the multi-way valve, the inlet is stagnation of the inlet boundary, the mass flow is defined according to the test requirements; the outlet is at the pressure outlet boundary, and the pressure is also defined according to the test requirements. The boundary between the oil and the wall surface is a refined non-slip wall surface, and the boundary type is Wall.

4. Analysis of CFD simulation results
In order to understand the characteristics of the flow field in the valve chamber and the stress of the valve core under different opening degrees of the multi-way valve, the simulation calculation was carried out for the valve core under different opening degrees and different flow rates [5, 6]. The flow field simulation results were obtained when the valve opening degree was 5mm, 7mm, 8.5mm, 9.5mm, 10.5mm and 12mm, and the flow rate was 30 L/min, 62L/min, 180 L/min, 226.5l/min and 240 L/min, respectively. Due to the space limitation, this article does not enumerate one by one. Only the simulation results of valve orifice opening of 8.5mm, 9.5mm and 10.5mm and flow rate of 240 L/min are analyzed to provide theoretical basis for multi-channel valve optimization through the analysis of pressure, speed and hydraulic force.

4.1. Pressure distribution
According to the distribution of the static pressure cloud chart of Fig. 3, the high-pressure oil enters the multi-channel valve from the inlet, and when the oil flows through the main throttle of the multi-channel valve, the oil pressure drops rapidly, and the pressure change occurs mainly here, mainly due to the smaller flow area, resulting in the increase of the flow velocity, thus causing the pressure to decrease, and forming a partial low pressure area at the corner of the throttle orifice. Through the other throttles, the oil pressure decreases slowly and gradually becomes stable, and finally flows out through the outlet.

Observing 2a, 2b and 2c, it was found that the inlet pressure at the opening of 8.5 mm was higher than that at the opening of 9.5 mm and 10.5 mm. Compared with the simulation results of the opening of other valves, it is found that the smaller the opening, the greater the inlet pressure, the greater the pressure of the points in the valve cavity. By comparing the local low pressure zones of each valve opening, it is found that when the inlet flow rate is the same, the pressure in the local low pressure zone decreases with the decrease of the valve opening. With the same opening and different inlet flow rate, the smaller the flow rate, the greater the pressure in the local low pressure zone at the corner of the valve core.

![Figure 3. Static pressure cloud of cross section](image)
4.2. Velocity Distribution

From the distribution of the streamline cloud chart of Fig. 4 and the cross section velocity cloud chart of Fig. 5, it is known that the flow of the hydraulic oil in the valve cavity is basically the same, and the inlet velocity is kept steady. When the nozzle reaches the throttle orifice, the flow rate shrinks rapidly and the velocity increases rapidly due to the smaller flow area. The pressure is drastically reduced by the equation, which is consistent with the sudden decrease of pressure at the throttle in Fig. 3. From Fig. 4, we can also see that the inflexion of the valve chamber and seat produces jet flow at the throttle. At the same time, because the size of the flow passage suddenly decreases, there will be a whirlpool between the flow beam and the wall. Observing 5a, 5b and 5c, it was found that the flow velocity at the throttle opening was greater than that at the throttle opening of 9.5 mm and 10.5 mm when the valve opening was 8.5 mm. Comparing the velocity cloud chart of the same flow rate and different valve opening, it is found that the smaller the valve opening, the greater the flow velocity at the throttle opening. When the valve opening is small, the pressure in the valve chamber will increase, and the eddy current in the valve chamber will also decrease.

![Figure 4. Streamlines cloud chart](image-url)
4.3. Hydrodynamic Distribution

The hydraulic power can be divided into steady state hydraulic power and transient liquid power according to the different generation mechanism. Transient hydraulic power is small in the general configuration of multi-way valves and is usually negligible. Steady-state hydraulic power means that when the valve opening is constant and the flow rate is kept constant, the hydraulic oil acts on the valve core due to the change in flow rate and direction when passing through the valve port. The pressure on the axial wall of the spool is not uniform. In theory, the integral can be used to calculate the hydraulic power of the spool.

The formula is as follows [1]:

$$ F_2 = \sum \int A_i \cdot \left( \vec{n} \times dA \right) $$

Type: $p$ is the wall pressure, $\vec{n}$ is the positive direction of the spool axis, $A_i$ is the force wall surface of the spool, $dA$ and is the area element.
Fig. 6 shows the comparison curves of steady-state hydrodynamic simulation results and calculation results of spool valves with different openings at the inlet flow rates of 62 L/min, 226.5 L/min and 240 L/min, respectively. It can be seen that the simulation results of CFD are consistent with the theoretical results, and the numerical values are close to each other, thus verifying the feasibility of CFD simulation for steady-state hydrodynamic calculation of spool valves.

As can be seen from Fig. 6, when the opening of the valve is the same, the larger the flow rate, the greater the steady hydrodynamic force the valve core is subjected to. At the same flow rate, the larger the opening of the valve, the smaller the steady hydrodynamic force on the valve core. When the opening of the valve is 7 mm to 10.5 mm, the change of steady-state hydrodynamic force is remarkable. When the opening of the valve is 10.5 mm to 12 mm, the change rate of steady-state hydrodynamic force tends to be stable. This is due to the small opening of the valve, the remarkable Throttling Characteristics of the throttle and the sharp increase of the flow velocity. Therefore, in the work of sliding valves, attention should be paid to the control of steady-state hydraulic power with large workflow and small opening of valve mouth. Under different flow rates, the change of steady-state hydrodynamic force is similar. The direction of hydrodynamic force tends to close the valve orifice, and increases with the increase of flow rate and decreases with the increase of opening.

5. Conclusion
Simulation analysis of multi-way valve pressure, speed and hydraulic power, and the corresponding cloud map, data and curve, comparison simulation and test results, can get the following conclusions:

1) When the oil passes through the main orifice of the multi-way valve, the flow rate is sharply contracted due to the smaller flow area, the speed increases rapidly, and the pressure is drastically reduced;

2) When the flow rate is the same, the smaller the opening degree, the larger the inlet pressure, the greater the pressure at each point in the valve chamber; the larger the flow rate at the orifice. And the steady-state hydraulic power acting on the valve core increases with the increase of the flow rate, and decreases with the increase of the opening degree;
3) The theoretical calculation results of steady-state hydraulic power are close to those obtained by CFD simulation. When the valve opening is the same, the larger the flow rate, the greater the steady-state hydraulic power that the spool receives. When the flow rate is constant, the steady state hydraulic power decreases as the valve opening degree increases.

References

[1] Amirante R, Del Vescovo G, Lippolis A. Flow forces analysis of an open center hydraulic directional control valve sliding spool [J]. Energy Conversion and Management, 2006, 47: 114 - 131.

[2] Wang Fu-jun. Computational Fluid Dynamics Analysis: Principle and Application of CFD Software [M]. Beijing: Tsinghua University Press, 2004.

[3] Tao G, Chen H Y, J Y Y. Optimal design of the magnetic field of a high-speed response solenoid valve [J]. Journal of Materials Processing Technology, 200, 129 (13): 555 - 558.

[4] Zheng Shu-juan, Quan long, Chen Qing. CFD calculation and analysis of flow field of hydraulic cone valve during spool movement [J]. Chinese journal of agricultural machinery, 2007, 38 (1): 168 - 172

[5] Zhang Hai-ping. Correct some misconceptions about steady-state hydraulic power [J]. Hydraulics pneumatics & seals, 2010, (9): 10 - 15.

[6] Han Zhan-zhong, Wang Jing, LAN Xiao-ping. FLUENT: examples and applications of fluid engineering simulation calculation [M]. Beijing: Beijing institute of technology press, 2004.

[7] Guillermo Palau-Salvador. Three-dimensional modeling and geometrical influence on the hydraulic performance of a control valve [J]. Journal of Fluids Engineering, 2008, 130: 11102.