Internal Flow Field Analysis of Three-way Regulating Valve Based on CFD Technology

Hou Yingzhe ¹, Wu Hao¹ and Li Zhong²

¹ Wuhan Secondary Ship Design and Research Institute, Wuhan, Hubei, 430205, China
² Hefei General Machinery Research Institute Co., Ltd., Hefei, Anhui, 230031, China

Abstract. According to the requirement of low energy consumption and high efficiency of a three-way regulating valve used in a system, based on the CFD software of fluid mechanics, the internal flow performance of the three-way regulating valve was simulated and studied in three dimensions, and the flow field information such as internal pressure and flow velocity of the valve were obtained. The results show that with the increase of the opening degree, the pressure gradient and velocity gradient at the upper valve seat gradually tend to be obvious, the pressure drop at the valve seat and the maximum flow velocity at the sealing surface gradually tend to be extreme, while the change rule at the lower valve seat is opposite. When the valve works under extremely small and extremely large opening conditions for a long time, it will be beneficial to protect the valve.

1. Introduction

Three-way control valve, also called three-way control valve, is divided into two structural forms of confluence and diversion. It can replace two single-seat and double-seat control valves which are mutually switched to realize automatic adjustment and control of fluid medium in process pipelines. It has the characteristics of high adjustment precision, compact structure, stable and reliable action, etc. It is widely used in industrial systems which can accurately control the process parameters of gas, liquid, steam and other media to maintain a given value.

Using computational fluid dynamics software, this paper conducts a three-dimensional simulation study on the internal flow performance of the three-way regulating valve, a key regulating device in a certain system. The flow performance of the area near the transition cone of the valve core of the three-way regulating valve under typical working conditions is mainly analyzed, and the influence law of the existence of the transition cone of the valve core on the internal turbulence kinetic energy and turbulent dissipation rate of the three-way regulating valve is discussed, which provides a certain reference for the internal flow law research and structural optimization design of similar valves.

2. Structure Principle and Performance Parameters

2.1. Structure Principle

The three-way regulating valve is mainly composed of valve body, valve core, valve seat, valve stem, support and other components (Figure 1). Its working principle is that the actuating mechanism drives the valve stem to move up and down to realize up and down sealing of the valve core sealing surface and valve seat sealing surface in the stroke of the three-way valve. Through the regulating valve core at
different positions, the flow rate of the main inlet and the side inlet can be controlled to realize that the outlet flow rate meets the functional requirements of the system.

Figure 1. Structure of three-way regulating valve.

2.2. Performance Parameters

Table 1. Performance Parameters of Three-way Regulating Valve

| Name                      | Value     |
|---------------------------|-----------|
| Nominal diameter          | DN100     |
| Working pressure (MPa)    | 1.0       |
| Temperature range (°C)    | -10 ~ 45  |
| Valve core stroke (mm)    | 50        |
| Medium                    | water, oil|

3. Numerical Simulation

3.1. Governing equation

In this paper, a relatively mature standard model is used for numerical simulation [1-2]. In standard turbulence k-ε model, turbulent kinetic energy k and turbulent kinetic energy dissipation rate transport equation ε can be expressed as,

- Continuity equation
Momentum conservation equation (N-S equation) formula

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}
\]

- The turbulent kinetic energy \(k\) and turbulent kinetic energy dissipation rate transport equation \(\varepsilon\) of the standard turbulence model can be expressed as,

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial u_i}{\partial x_i} + \frac{\partial \mu}{\partial x_i} \right) + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} \right) + \frac{k}{\varepsilon} (C_{\varepsilon} P + \rho C_{\varepsilon,\varepsilon}) \tag{3}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \mu + \frac{\mu_t}{\sigma} \right) \frac{\partial \varepsilon}{\partial x_i} + \frac{\varepsilon}{k} \left( C_{\varepsilon} P + \rho C_{\varepsilon,\varepsilon} \right) \tag{4}
\]

In the formula, \(u_i\) is the velocity component in direction \(i\), \(i=1, 2, 3\), \(\mu\) is kinematic viscosity coefficient of medium, \(\mu_t\) is vortex viscosity coefficient, \(\delta_k\) is Prandtl number Corresponding to Turbulent Kinetic Energy \(K\), \(\delta_\varepsilon\) is Prandtl number Corresponding to Turbulent Kinetic Energy Dissipation Rate \(\varepsilon\), \(P\) is turbulent kinetic energy generation term.

\[
P = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \tag{5}
\]

Among them,

\[
\mu_t = C_{\mu} \rho \frac{k^2}{\varepsilon} \tag{6}
\]

Table 2. Coefficient in k-\(\varepsilon\) Model

| \(C_{\varepsilon}1\) | \(C_{\varepsilon}2\) | \(C_\mu\) | \(\delta_k\) | \(\delta_\varepsilon\) |
|---|---|---|---|---|
| 1.44 | 1.92 | 0.09 | 1.0 | 1.3 |

3.2. Flow Channel Model Establishment and Grid Division

Three-way regulating valve with nominal diameter DN=100mm diameter is selected as simulation experiment object. According to specific structural parameters, Solidworks is used to establish three-dimensional model of three-way regulating valve, and fluid mechanics software is imported for reverse modelling to generate internal flow channel model of three-way valve (Figure 2).
The flow channel model is imported into ANSYS ICEM CFD for grid division (Figure 2). Considering the complexity of the flow channel inside the three-way valve, the numerical calculation grid is divided by tetrahedron/mixed grid, as shown in reference [3], local encryption treatment is carried out at the corner of the flow channel to make the calculation result more accurate, and grid independence test is carried out by taking the mass flow rate in the fully open state as the measurement standard [3].

According to the actual operating conditions of the three-way regulating valve, the internal flow model under typical operating conditions is established. The closed equations are formed by Reynolds time-averaged equations of three-dimensional incompressible flow and k-ε turbulence model to solve the internal flow field of the three-way regulating valve. The fluid medium is set as water, and the temperature change is not considered in the solving process. Both the main inlet and the side inlet of the three-way valve are set as velocity inlet boundary conditions, the velocity is 1.5m/s, the outlet is pressure outlet boundary conditions, the wall surface of the three-way valve is set as adiabatic non-slip boundary conditions, and three-dimensional flow field numerical simulation analysis is carried out [4-5].

4. Research on Internal Flow Performance

4.1. Pressure Field Distribution under Different Openings
Due to the particularity of the valve core structure, the internal flow field at the upper and lower valve seats of the three-way regulating valve shows the opposite change rule during the whole opening and closing process. Therefore, the evolution rule of the internal flow field is analyzed under four typical opening states of valve core stroke of 20%, 40%, 60% and 80%.

Figure 3 is a pressure distribution cloud chart at the upper and lower valve seats of the lower valve core with four typical opening degrees. It can be seen from Figure 3 that with the increase of opening degree, the pressure distribution law at the upper and lower valve seats shows an opposite change trend, the pressure distribution at the lower valve seat gradually tends to be uniform, and the pressure gradient change at the upper valve seat gradually becomes obvious.

The area at the upper and lower valve seats of the three-way regulating valve is the area where the pressure changes greatly and the flow velocity is concentrated during the opening and closing process of the three-way valve. Monitoring points are set at the upper and lower valve seat areas for further research, as shown in reference [4] to monitor the pressure drop at the upper and lower valve seat positions at different opening degrees. Fitting the curve to obtain the pressure drop characteristic curve of the three-way valve and quantitatively analyze its internal evolution law. Figure 4 is a pressure drop characteristic curve at the upper and lower valve seats at different opening degrees. It can be seen from
the figure that the pressure drop at the position of the upper valve seat fluctuates gently within 0.5% of the opening. After the opening is greater than 50%, due to the special structure of the structure, the throttling area at the position of the upper valve seat gradually decreases, and the pressure drop increases sharply with the increase of the opening, resulting in certain energy loss. The pressure drop at the lower valve seat position is opposite to its change, which decreases sharply at first and then fluctuates gently with the increase of the opening.

4.2. Velocity Field Distribution under Different Openings
Similarly, the velocity field distribution in the transition cone area of the valve core of the three-way regulating valve was studied, and the velocity distribution nephogram in the transition cone area under different opening degrees was obtained, as shown in Figure 5. It can be seen from Figure 5 that with the increase of opening degree, the velocity distribution law at the upper and lower valve seats shows an opposite change trend, the velocity distribution at the lower valve seat tends to be stable, the pressure gradient at the upper valve seat changes obviously, and the local maximum velocity occurs at the maximum sealing surface, resulting in certain scouring to the sealing surface of the upper valve seat.

The monitoring data obtained the maximum velocity characteristic curve of the transition cone area, as shown in Figure 6. It can be seen from the figure that within 0~50% opening, the throttling area at the sealing surface of the upper valve seat is larger, the flow inside the valve is more stable, the maximum
flow velocity at the sealing surface of the upper valve seat basically tends to be stable, after more than 50% opening, the movement of the valve core makes the throttling area at the upper valve seat gradually decrease, the maximum flow velocity at the sealing surface greatly increases, and a certain amount of scouring occurs to the sealing surface; The maximum flow velocity at the sealing surface of the lower valve seat changes in the opposite direction. With the increase of the opening degree, the maximum flow velocity decreases sharply at first and then fluctuates gently. When the opening degree is small, the sealing surface of the lower valve seat will be scoured to a certain extent.

![Figure 6. Characteristic curve of maximum velocity under different opening degrees.](image)

5. Conclusion
The flow evolution law at the valve seat of the three-way regulating valve shows a certain change trend during the whole opening and closing process: with the increase of opening degree, the change law of pressure gradient and velocity gradient at the upper valve seat gradually tends to be obvious. The pressure drop and maximum flow rate fluctuate smoothly within 0~50% opening degree, and increase sharply after being larger than 50% opening degree, while the change law at the lower valve seat is opposite.

References
[1] Song X.G., Cui L., PARK Y.C. Three-dimensional CFD analysis of spring-loaded pressure safety valve from opening to re-closure. Proceedings of the ASME 2010 Pressure Vessels and Piping Division/K-PVP Conference, Washington, ASME, 2010 (5): 295–303.
[2] Chen J., Yuan Z.H., Guo Q., etc. Dynamic flow field analysis of the prestage of jet pipe servo valve. Chinese Journal of Scientific Instrument, 2017, 38(07):1731-1737.
[3] Zhang Y.K., Liu P.K., Yang X.H., etc. Numerical Simulation of Turbulent Flow Field of Mud Purification Hydrocyclone Based On CFD. Fluid Machinery, 2014, 42(06):26-32.
[4] Li S.X., Li Z., Zhou A.M., etc. Optimization and test on throttle plate opening profile line of 3-way control ball valve. Journal of Huazhong University of Science and Technology (Nature Science Edition), 2017, 45(02):61-66.
[5] Srikanth C, Bhasher C. Flow analysis in valve with moving grids through CFD techniques. Advances in Engineering Sofware.2009,40(3):193-201.