CHANG CHEN
QIXIN ZHANG
SHIQIANG SONG
SHIFU ZHANG
JIKUN GUO
and
DONGYU YANG

Numerical simulation of internal flow field of liquid level control valve during closing process

Liquid level control value is widely used in the process industry to control internal liquid height as pipe flow control component. During the operation of the liquid level control valve, the force condition of the value core directly affects the stability of the valve works. Therefore, it becomes the important factors in the design of the spring components of the liquid level control valve. In this paper, the internal flow field of the automatic control valve during closing process is simulated by using Fluent CFD software. The numerical simulation results show that in the process of closing the automatic control value, the water hammer phenomenon will be occurred at the beginning stage. With the continuation of the valve closing process, the pressure in upper chamber of the value core and internal chamber are rapidly reduced and tend to be gentle, while the pressure difference increases gradually. What is more, the flow of throttle gradually increases, and the inner chamber of the valve core appears small disturbance. These conclusions provide a theoretical basis for the technical design and improvement of the control valve.

Keywords: *Numerical simulation, internal flow field, level control valve, closing process.*

1. Introduction

In recent years, with the rapid development of computer technology and computational fluid dynamics, numerical simulation has been widely used to study the complex flow inside the valve. Through numerical simulation, designers can understand the complex flow in it, so as to improve its internal structure to reduce the flow loss, improve valve vibration, and also save manpower and resources (Xu, et al., 2003). The basic principle of the liquid level control valve is to rely on the liquid pressure difference in different parts of the valve to control the valve opening and closing. Subtle changes in the internal structure of the control valve can lead to changes in the internal flow field, and even the formation of vortex and the disturbance, which will affect the working performance of the control valve. So it is necessary to study the internal flow field of liquid level control valve which provides the basis for the design and optimization of the structure of liquid level control valve.

There are many scholars who have performed a certain studies on this aspect. Eatwell, W. D illustrated an effective way of controlling and protecting the well formations by installing a liquid level control subsurface value (Eatwell, 1988). In order to minimize the movement of the motor value and maintain the tank level at the specified position, Khoei, A proposed a controller for achieving that purpose (Khoei, et al., 2005). C Hirsch introduced the fundamentals of computational fluid dynamics for numerical computation of internal and external flows (C, 2007). Wang, Zhan Yong put forward the function, working principle and characteristic of the oil control valve test system (Wang, et al., 2008). The influences of fluid-structure interaction on fluid velocity and eddy formation had been analyzed by Cao, Fang by building the fluid-structure interaction system model of control valve (Cao, et al., 2011). Zhang, Zhongzhen emphatically analyzed causes of the failure for the level control valve in the ammonia purification unit, so as to optimize the valve internal structure accordantly (Zhang, et al., 2011). Buchtel, Michael E introduced the liquid level control valve and its method in detail (Buchtel, 2012). Liu, Changfeng had introduced a new liquid level sensor with TDR technology and a robust motor valve which will work together to make the refrigeration system running in a safe and energy-saving way with easier service (Liu, 2013). Wang, Yaping conducted a simulation analysis of internal flow field in water storage tank which will make level control valve of water storage tank better to meet the work requirements (Wang, et al., 2014). D Wu aims to elaborate on specific computational fluid dynamics (CFD) simulation methods for fitting the flow-pressure curve of a pressure control valve, which is spring-load valve widely used in the automotive fuel supply system. A direct CFD

Messrs. Chang Chen, Qixin Zhang, Associate Professor, Oil Department, Shifu Zhang, Professor, National Engineering Research Center for Disaster & Emergency Relief Equipment, Jikun Guo, Professor, Service Command Department and Dongyu Yang, Postgraduate Student, Oil Department, Army Logistics University of PLA, Chongqing 401 331 and Shiqiang Song, Master Degree, Logistics Training Center of PLA Rocket Force, Hebei 075 000, China. *Corresponding author Mr. Chang Chen, e-mail: 408430734@qq.com

method is introduced to solve this problem. Two evaluation criteria are used to determine whether the internal flow is physically real. An experiment is conducted to verify the simulation results, and the accuracy of this CFD method is proved (Wu, et al., 2015). Nenniger, John invented an inflow control value for controlling the flow of fluids into a generally horizontal production wall and illustrated its using method (Nenniger, 2016). Cui, Baoling reported on experiments and numerical simulations which were implemented to investigate the influence of opening and closing process on the external transient performance and the internal flow characteristics of ball valve under different opening and closing time. The results show that external transient performance and flow field of ball valve have obvious difference between opening and closing process as the fluid lags behind the change of relative opening. With the opening or closing time increase, the differences gradually become small and they are gradually close to that of steady condition (Cui, et al., 2017).

It must also be mentioned that, little is known, about the numerical simulation of internal flow field of liquid control value during its closing process. And we demonstrate through an extensive literature review that the existing models are not capable of handling the specifics of problem in this study. Based on the basic theory of computational dynamic fluid and the basic model of turbulence, this paper uses the computational fluid dynamics Fluent CFD software to simulate the internal flow field of the liquid level control valve during closing process, which is believed that the research result has great practical significance for advancing research level of liquid level control valve.

2. Basic theory and algorithm

2.1 Computational fluid mechanics

Computational fluid dynamics (CFD) is a numerical simulation system for analyzing physical problems such as fluid flow and heat conduction (Wen, et al., 2009). It is based on the three equations of mass conservation, momentum conservation, and energy conservation, and synthetically uses the numerical calculation and image display of compute function. Through the calculation and analysis of computational fluid dynamics, the basic physical quantity (such as velocity, pressure, temperature, concentration, etc.) of the complex problem in the flow field and the changes over time can be expressed in the form of image or animation. The method has the advantages of strong universality, fast operation speed and high precision. The solution process is shown in Fig.1 (Yu, 2008).

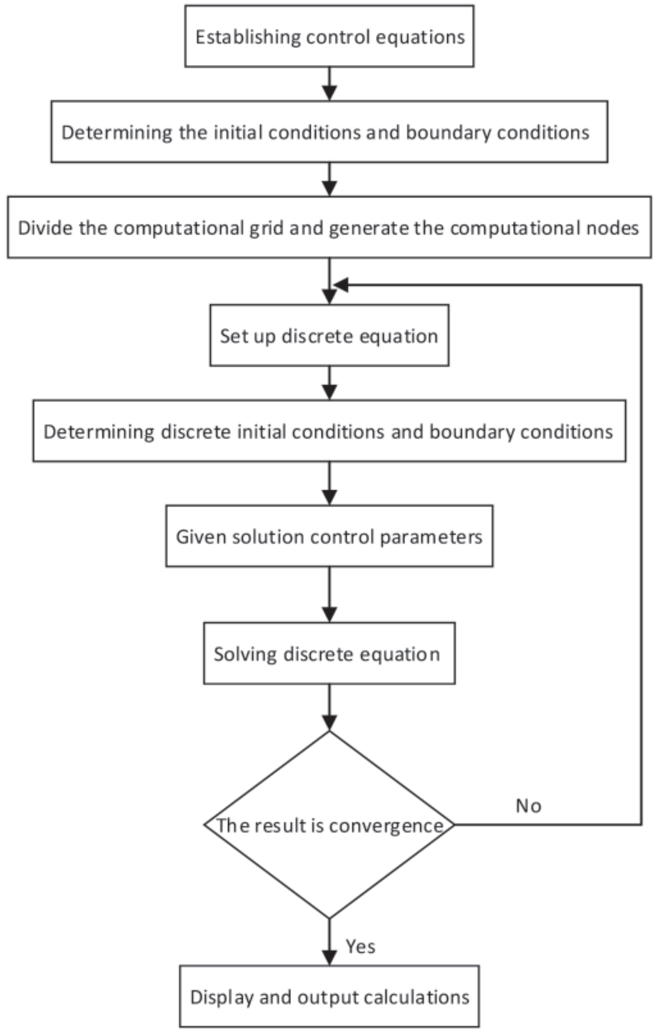

Fig.1 Work flow diagram of CFD

2.2 Numerical simulation of turbulence

Turbulence is a highly complex and irregular unsteady flow formed by the vorticity of randomly distributed vanes in the size and direction of rotation. During the turbulent flow, the parameters such as pressure, velocity, and drag coefficient of the fluid change with time and space. The characteristics of turbulence include randomness, diffusivity, vorticity and dissipation (Liu and Liao, 2000), which contain both random and deterministic ordered components. Therefore, it is difficult to solve the turbulence problem by means of deterministic methods. At present, the turbulence numerical simulation method mainly includes direct simulation method, large eddy simulation method and Renault time-averaged method, in which direct simulation method and large eddy simulation method cannot be widely used in engineering calculation, mainly because the current computer memory and computing speed cannot meet its calculation requirements, so the Renault time-averaged method has become the most widely used turbulence numerical simulation method (Rollet-Miet, et al., 1999).

It is generally believed that the unsteady continuity equation and the N-S (Navier-Stokes) equation can describe any complex turbulent instantaneous motion in nature, but the computational complexity is very large, so it cannot be achieved in engineering practice. The Renault time-averaged method is to simplify the calculation of the N-S equation with respect to the time-dependent variable application statistics (H.K.Versteeg and W.Malalasekera, 2000).

For incompressible fluids, the continuity equation is:

$$\frac{\partial u_i}{\partial x_i}=0 \quad ... \quad (1)$$

In formula (1), u_i represents the instantaneous speed in the direction i.

The N-S equation is:

$$\rho\frac{\partial u_i}{\partial t}+\rho u_j\frac{\partial u_i}{\partial x_j}=\rho F_i-\frac{\partial p}{\partial x_i}+\frac{\partial^2 u_i}{\partial x_j \partial x_j} \quad ... \quad (2)$$

The Reynolds averaging method is introduced to define the time average value of any variable ϕ as:

$$\overline{\varphi}=\frac{1}{\Delta t}\int_t^{tt+\Delta}\varphi(t)\,dt \quad ... \quad (3)$$

The instantaneous value of any physical quantity can be expressed as:

$$\varphi=\overline{\varphi}+\varphi' \quad ... \quad (4)$$

In formula (4), the superscript "-" represents an average value relative to time and " ' " indicates the ripple value. The turbulent flow variables are replaced by the sum of the average value and the ripple value:

$$\begin{cases} u_i=\overline{u}_i+u_i^{'} \\ p_i=\overline{p}_i+p_i^{'} \end{cases} \quad ... \quad (5)$$

Substituting formula (5) into continuity equation (1) and N-S equation (2), and then taking the time average for the equation:

$$\frac{\partial \overline{u}_i}{\partial x_i}=0 \quad ... \quad (6)$$

$$\rho\frac{\partial \overline{u}_i}{\partial t}+\rho\overline{u}_j\frac{\partial \overline{u}_i}{\partial x_j}=\rho\overline{F}_i-\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}(\mu\frac{\partial \overline{u}_i}{\partial x_j}-\rho\overline{u}_i\overline{u}_j) \quad ... \quad (7)$$

Equation (7) is called Reynolds-Averaged N-S equation (RANS), often called Reynolds equation, and among it, the $-\frac{\partial}{\partial x_j}\rho\overline{u}_i\overline{u}_j$ is called Reynolds stress. The Reynolds equation simplifies the turbulence variable to the average value and the ripple value, but also introduces a new unknown-the Reynolds stress, which is related to the turbulence ripple value. The introduction of new unknowns makes the original equations no longer closed and cannot be solved. Therefore, new turbulence models must be introduced. In the new turbulence model, the Reynolds stress associated with the turbulence ripple value is reassembled to correlate the pulsating value of the turbulence with the average value.

2.3 Wall-function method

The RNG k-ε model is only effective for the fully developed turbulence, but in the wall area, the flow characteristics are very different from those in the turbulent region, especially in the viscous bottom layer. Since the effect of turbulence stress is almost negligible in this area, the flow of the viscous bottom layer almost exhibits as the laminar flow. In order to solve this problem, the usual practice is to introduce the wall function method in the wall area. Wall-function method is actually a set of semi-empirical formulas that relate the physical quantity on the wall directly to the unknown quantity sought in the core of the turbulence (Zhang, et al., 2005). The basic idea is to use the RNG k-ε model to solve the flow of the turbulent core region. Instead of solving the solution in the wall area, it used the set of semi empirical formulas to directly analyze the physical quantities with the turbulence core region. It needs to place the first node in the troposphere, and the value of the first node is determined by the formula, so that the node variable value of the adjacent control volume of the wall can be obtained directly. As shown in Fig.2(b), the number of meshes required for the wall function method is small compared to the grid of the low Reynolds k-ε model (Fig.2(a)), which saved the memory and time required for the calculation.

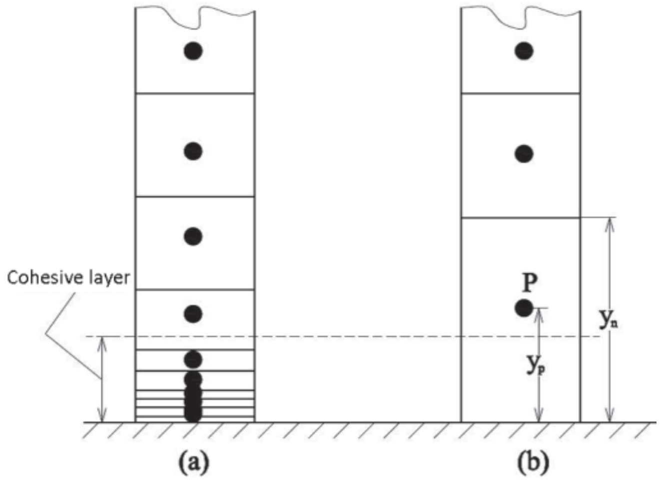

Fig.2 Processing method for region near the wall

In the wall function method, the velocity u^+, distance y^+, wall shear stress τ_w, turbulent kinetic energy generation item G_k and dissipation rate ε on the control volume node adjacent to the wall are calculated according to the following formula:

$$u^+=\frac{1}{\kappa}\ln\left(Ey^-\right) \quad ... \quad (8)$$

$$y=\frac{\Delta y_p\left(C_\mu^{1/4}k_p^{1/2}\right)}{\mu} \quad ... \quad (9)$$

$$\tau_w=\rho C_\mu^{1/4}k_p^{1/2}u_p/u^+ \quad ... \quad (10)$$

$$G_k=\tau_w\frac{\tau_w}{\kappa\rho C_\mu^{1/4}k_p^{1/4}u_p\Delta y_p} \quad ... \quad (11)$$

$$\varepsilon=\frac{C_\mu^{3/4}k_p^{3/2}}{\kappa\nabla y_p} \quad ... \quad (12)$$

In the above formula, u_p represents time averaged velocity, k_p represents turbulence energy, Δy_p represents distance between node p and wall, μ represents hydrodynamic viscous coefficient.

The internal fluid of the liquid level control valve is affected by the internal structure and the wall of the valve. The Reynolds number near the wall area is low, and the influence of the molecular viscosity cannot be neglected. In this paper, the wall function method has been applied to research the near wall area of the liquid level control valve.

3. Establishment of geometric model

After the float valve is closed, all the fluid flows out through the outer chamber of the automatic control valve. At this time, the automatic control valve is a longitudinal symmetrical structure. The flow characteristics of the fluid in the longitudinal midline section can basically reflect the whole situation of the fluid in the valve. In order to simplify the calculation, this paper establishes a two-dimensional approximation model to simulate the closing process of the automatic control valve.

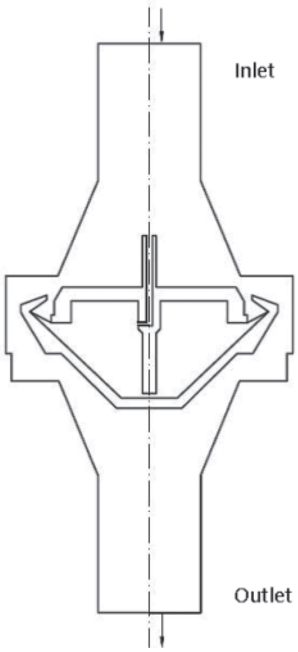

Fig.3 2D geometric model of automatic control valve

The two-dimensional geometric model of the automatic control valve in the fully open state has been established by using the AutoCAD software as shown in Fig.3. As in the simulation process, we main concern about the change of pressure, speed and other parameters in internal flow field, thus some unnecessary structures have been simplified. Taking the complexity of flow field at the inlet and outlet position of the level control valve into account, a 100 mm long pipe section is retained at the inlet and outlet positions of the automatic control valve.

4. Mesh generation

The two-dimensional geometric model established by AutoCAD is introduced into the pre-processing software Gambit to generate the grid model. Because the model is complicated, the triangular unstructured meshing is adopted. With the dynamic grid technology, the grid model for different degree of valve closure is shown in Fig.4.

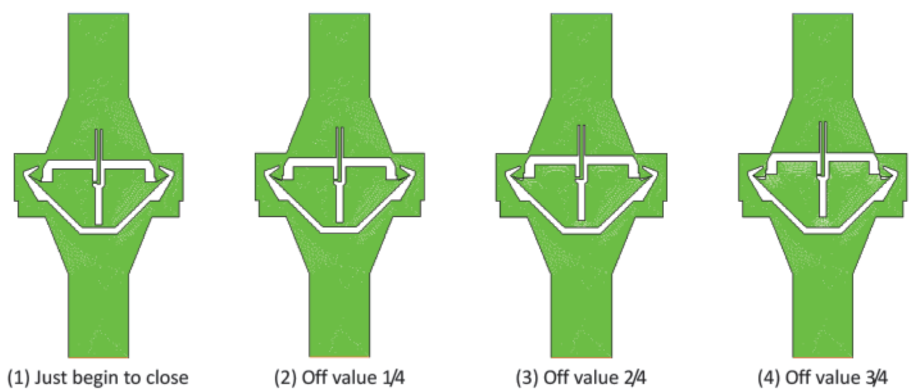

Fig.4 Mesh grids of calculation model

5. Boundary conditions and setting of calculation parameters

To start the two-dimensional single-precision version, select a pressure-based non-steady-state solver, and keep the first-order implicit format of the time term. Turbulent model select the RNG k-ε two equation model. Fluid material is set to water, and its density is modified to 1000 kg/m^3. Operating conditions remain the default settings, and the entire process ignores gravity effects. In the dynamic grid parameter setting process, smoothing, layering and remeshing are selected. The parameters of the smoothing mode are set to spring constant factor equals 1, boundary node relaxation equals 0.7, convergence tolerance equal to 0.001, number of iterations equal to 50. The parameters of the layering mode are set to split factor equals 0.4, collapse factor equals 0.04, and the parameters of the remeshing mode are set to minimum length scale equal 2e-05, maximum length scale equals 0.007, maximum cell skewness equals 0.6, size remesh interval equals 1. SIMPLE algorithm is chosen as pressure-speed coupling algorithm, and the convergence factor parameter to maintain the default settings. Since it is an unsteady flow, all residual convergence accuracy is defined as 0.001. The iteration is started after initializing the flow field.

6. Results and analysis

After the iteration calculation, the pressure, velocity distribution in the process of closing the valve can be obtained.

From the pressure distribution diagram (Fig.5), it can be seen that when the automatic control valve is begin to close, the upper chamber of the valve and the internal chamber of the value core generate a high pressure, and the outer chamber of the value core produces a negative pressure. This is because at the beginning of the closure procedure, the acceleration of the core of the automatic control value is relatively large, and the value core obtained an initial closing speed in a short period of time, which results in the occurrence of water hammer phenomenon. With the continuation of the valve closing process, the pressure on the upper chamber of the value core

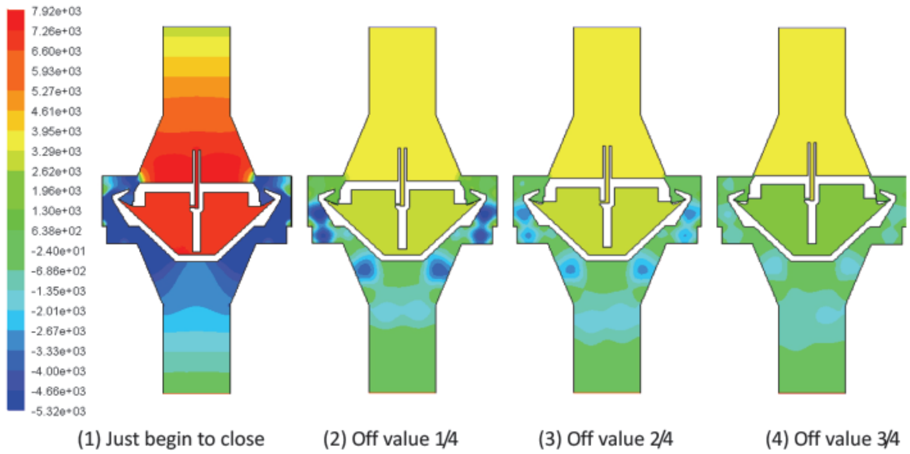

Fig.5 Pressure contours

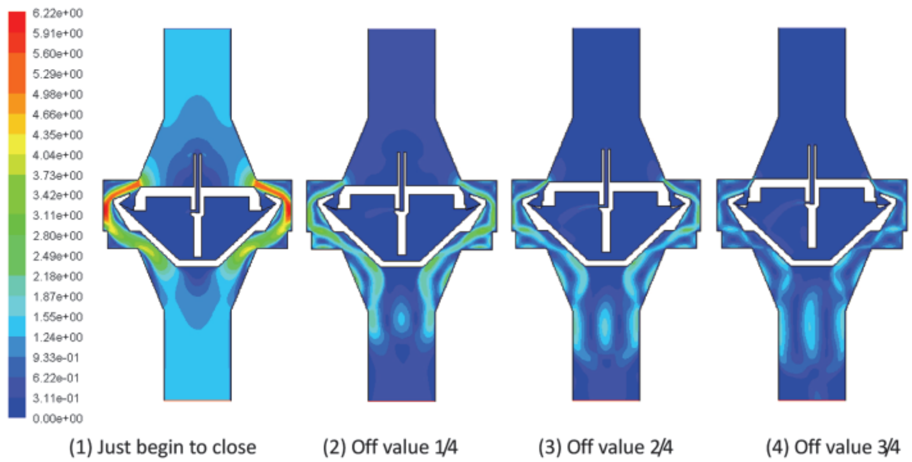

Fig.6 Velocity contours

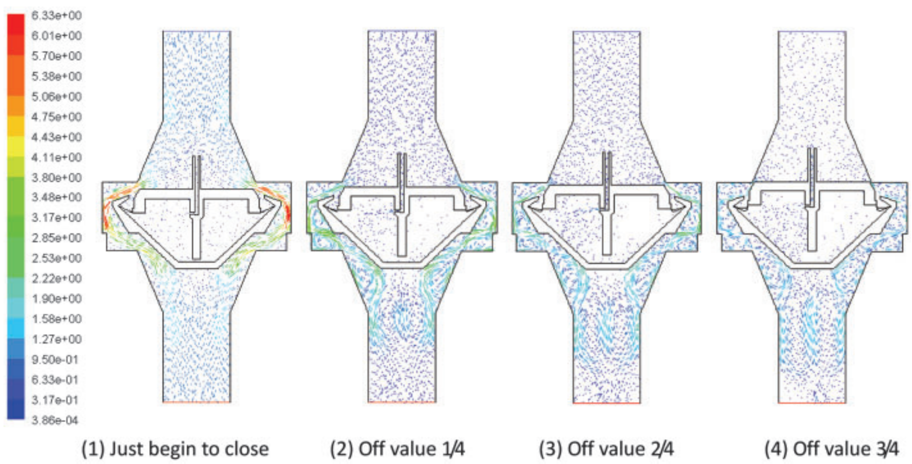

Fig.7 Velocity vectors

remained stable and the pressure in the internal chamber of the value core was gradually reduced. The pressure difference between the upper and inner chamber of the value core is gradually increased. The pressure in the outer chamber of the value core is gradually increased.

From the velocity distribution diagram (Fig.6), it can be seen that when the control valve is begin to close, the fluid velocity increases abruptly, which is caused by a sudden pressure increases at the upper chamber of value core. At the same time, the increase of fluid velocity also leads to the emergence of negative pressure in the outer chamber of the value core. As the process of closing the valve continues, the velocity at the outer chamber of the value core is gradually reduced and the velocity of the fluid at the orifice is gradually increased due to the gradual increase of the pressure difference between the upper and the internal chamber of the core of the value.

From the velocity vector diagram (Fig.7), it can be seen that there is a large eddy current and disturbance in the outer chamber of the value core and the outlet pipe during the closing process, and a small disturbance occurs in the internal chamber. Therefore, it is necessary to control the flow direction of the valve core in the form of a guide bar.

From the pressure changes diagram at the position of value core (Figs.8 and 9), it can be seen that the pressure of the upper chamber of the value core is larger than that of the internal chamber. With the continuation of the valve closing process, the pressure difference between them gradually increases, indicating that the acceleration of liquid inside the valve is gradually reduced.

7. Conclusion

In this paper, the governing equations of the internal flow field of the automatic control valve are established by using the basic theory of computational dynamic fluid and the RNG two equations turbulence model. The finite volume method is used to discretize the equations. The internal flow field of the automatic control valve during closing process is simulated by using Fluent CFD software. And the pressure diagram,

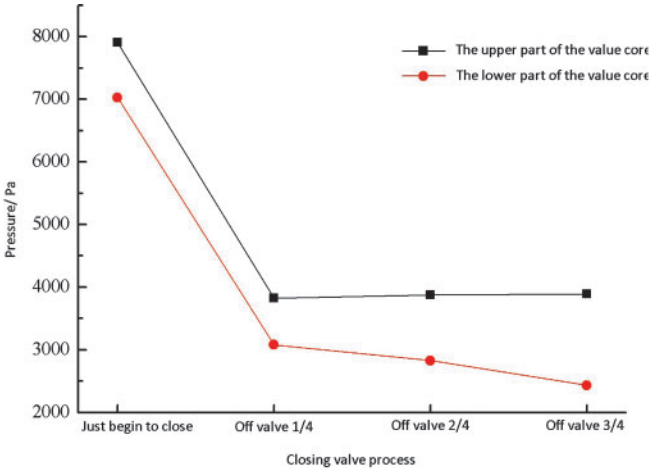

Fig.8 Average pressure at value core

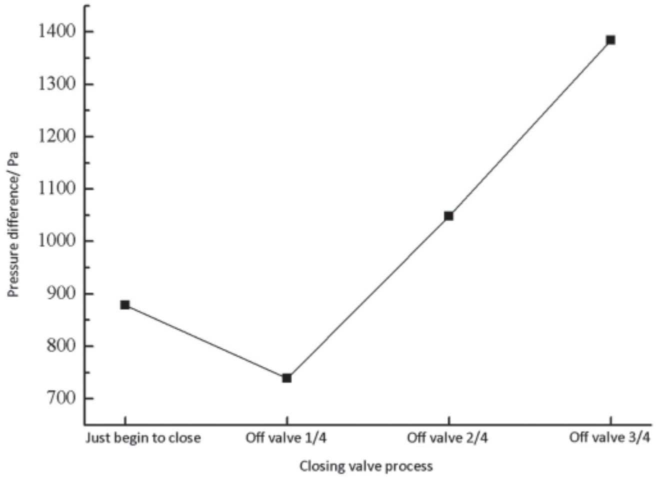

Fig.9 Pressure difference between the upper and lower part of the value core

velocity profiles diagram, velocity vector diagrams and streamlines diagram were obtained. The numerical simulation results show that in the process of closing the automatic control value, the water hammer phenomenon will be occurred at the beginning stage, resulting in the increase of pressure in upper chamber of the value core and the velocity of fluid at outer chamber of value core. With the continuation of the valve closing process, the pressure in upper chamber of the value core and internal chamber are rapidly reduced and tend to be gentle, while the pressure difference increases gradually. What is more, the flow of throttle gradually increases, and the inner chamber of the valve core appears to be small disturbance. These conclusions provide a theoretical basis for the technical design and improvement of the control valve.

Acknowledgments

This paper is supported in part by The National Key Technology R&D Program of China (2017YFC0806306), Education Science fund of the Military Science Institute of Beijing, China (No.2016JY481), and Chongqing Graduate Scientific Research Innovation Project, China (No.CYB17148).

Conflicts of interest

The authors declare that there are no conflicts of interest regarding the publication of this paper.

References

1. Buchtel, M. E. (2012): Fluid level control toggle valve device and method: US, 8,091,581 [P]. 2012-1-10.
2. C, H. (2007): Numerical computation of internal and external flows: The fundamentals of computational fluid dynamics [M]. Butterworth-Heinemann, 2007.
3. Cao, F., Wang, Y., Yantao, A. N. and Xie, Y. (2011): “Fluid-structure Interaction Analysis for Large-scale Gas Control Valve.” *Machine Tool & Hydraulics*, 455(11): 146-150.
4. Cui, B., Lin, Z., Zhu, Z., Wang, H. and Ma, G. (2017): “Influence of opening and closing process of ball valve on external performance and internal flow characteristics.” *Experimental Thermal & Fluid Science*, 80193-202.
5. Eatwell, W. D. (1988): Liquid level control subsurface valve reduces workover expense in artificial lift pump installations; proceedings of the SPE Annual Technical Conference & Exhibition, October 2, 1988 - October 5, 1988, Houston, TX, USA, F, 1988 [C]. Publ by Soc of Petroleum Engineers of AIME.
6. Versteeg, H. K. and Malalasekera, W. (2000): An Introduction to Computational Fluid Dynamics [M].World Book Inc, 2000.
7. Khoei, A., Hadidi, K., Khorasani, M. R. and Amirkhanzadeh, R. (2005):. “Fuzzy level control of a tank with optimum valve movement.” *Fuzzy Sets and Systems*, 150(3): 507-523.
8. Liu, C. (2013): “Application of new type liquid level sensor and motor valve into liquid level control.” *Refrigeration and Air-Conditioning*, 13(3): 26-29.
9. Liu, Y. and Liao, G. (2000): Advanced Fluid Mechanics [M]. Shanghai: Shanghai Jiao Tong University Press, 2000.
10. Nenniger, J. (2016): Inflow control valve for controlling the flow of fluids into a generally horizontal production well and method of using the same: U.S, 9,394,769 [P]. 2016-7-19.
11. Rollet-Miet, P., Laurence, D. and Ferziger, J. (1999): “LES and RANS of turbulent flow in tube bundles.” *International Journal of Heat & Fluid Flow*, 20(3): 241-254.
12. Wang, Y., Gao, L., Ge, J., Gao, H. and Zhang, J. (2014): “The numerical simulation analysis of flow field in level control valve of water storage tank.” *International Journal of Control and Automation*, 7(10): 45-52.

Continued on page 84