CHINESE JOURNAL OF MECHANICAL ENGINEERING
Vol. 28, No. 3, 2015

DOI: 10.3901/CJME.2015.0313.030, available online at www.springerlink.com; www.cjmenet.com; www.cjme.com.cn

Effect of Cone Angle on the Hydraulic Characteristics of Globe Control Valve

LIN Zhe[1,*], WANG Huijie[1], SHANG Zhaohui[1], CUI Baoling[1], ZHU Chongxi[2], and ZHU Zuchao[1]

1 *Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Zhejiang Sci-Tech University,*
Hangzhou 310018, China
2 *Zhejiang Windey Co., Ltd., Hangzhou 310012, China*

Received July 3, 2014; revised January 8, 2015; accepted March 13, 2015

Abstract: Globe control valve is widely used in chemical, petroleum and hydraulic industries, and its throttling feature is achieved by the adopting of valve plug. However, very limited information is available in literature regarding the influence of valve plug on the internal and external features in globe control valves. Thus the effect of valve plug is studied by CFD and experiment in this paper. It is obtained from external features that the pressure drop between upstream and downstream pressure-sampling position increases exponentially with flow rate. And for small valve opening, the increment of pressure drop decreases with the increase of cone angle (β). However, with the increase of valve opening, the effect of cone angle diminishes significantly. It is also found that the cone angle has little effect on flow coefficient (C_v) when the valve opening is larger than 70%. But for the cases less than 70%, C_v curve varies from an arc to a straight line. The variation of valve performance is caused by the change of internal flow. The results of internal flow show that cone angle has negligible effect on flow properties for the cases of valve opening larger than 70%. However, when valve opening is smaller than 70%, the pressure drop of orifice decreases with the increase of β, making the reduction in value and scope of the high speed zone around the conical surface of valve plug, and then results in a decreasing intensity of adjacent downstream vortex. Meanwhile, it is concluded from the results that the increase of cone angle will be beneficial for the anti-cavitation and anti-erosion of globe control valve. This paper focuses on the internal and external features of globe control valve that caused by the variation of cone angle, arriving at some results beneficial for the design and usage of globe control valve.

Keywords: globe control valve, cone angle, numerical simulation, experiment, internal and external features

1 Introduction

Globe control valve is a special kind of globe valve, having the function of regulating flow, as shown in Fig. 1. It is widely used in piping systems applied to the chemical, petroleum and hydraulic industries. The throttling feature of Globe control valve is achieved by the adopting of valve plug. It is thus important to study the influences of valve plug on the performance of valve, and then to conduct valve design.

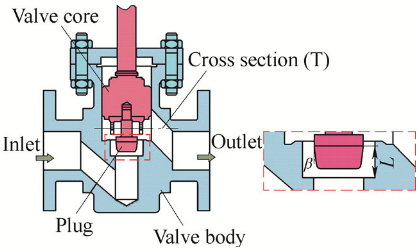

Fig. 1. Structure of globe control valve

The internal and external features of valves have been analyzed in many studies[1–10]. MOUJAES, et al[1], used CFD to investigate the effects of Reynolds number on the loss coefficient and flow coefficient of a ball valve. AN, et al[2], predicted three-dimensional incompressible flow characteristics of high pressure drop control valves by a CFD-ACE code. MORRIS, et al[3], carried out experimental studies to obtain the compressible flow field of a butterfly valve, and the roles of valve disc shape, valve disc angle and operating pressure ratio were investigated. LEUTWYLER, et al[4–6], have done many two-dimensional and three-dimensional simulations for butterfly valves, and the influences of compressible flow on flow field, resultant force and aerodynamic torque were indicated. CHERN, et al[7], investigated the flow patterns and cavitation phenomena in a ball valve by flow visualization experiments. PALAU, et al[8], analyzed the effect of geometry on the performance of a 3D control valve by numerical simulations and experiments. FERRARI, et al[9], presented the relationship between valve pressure drops and flow-induced forces of a globe valve. CHO, et al[10], analyzed the force balance relationship of an unbalanced globe valve with respect to the opening of the valve. However, only a few studies are focused on globe control

---

* Corresponding author. E-mail: linzhe0122@zstu.edu.cn

Supported by National Natural Science Foundation of China (Grant Nos. 51406184, 21276241), and Science Foundation of Zhejiang Sci-Tech University of China (Grant No. 14022005-Y)

© Chinese Mechanical Engineering Society and Springer-Verlag Berlin Heidelberg 2015

valves. JAMES, et al[11–12], used CFD and experimental methods to study the external performance and the flow fields of globe control valves, and the role of seat radius was investigated. Thus, very limited information is available in literature regarding the influence of valve plug on the internal and external features of globe control valves.

Owing to the development of computer technology, numerical simulation have been an efficient way for the investigation of fluid machineries[1–5, 13–31], including valves. Therefore, in present study, globe control valves are investigated by combining CFD and experimental methods. The influence of cone angle of plug on flow properties (velocity distribution, pressure distribution and vortex distribution) and throttling performance (pressure-flow rate curve and flow coefficients) are discussed, under different valve openings.

2 Problem Description and Setting

Since most of control valves use conical plug to obtain better throttling performance, conical plug is investigated in present study (Fig. 1). The cone angles (β) of 7.5°, 10°, 12.5°, 15°, 17.5° are analyzed, and the valve plug with β of 7.5° is tested experimentally. The maximum distance (L) between valve plug and valve body is 27.5 mm, which is used to get a non-dimensional valve opening (l). So L=0 represents the valve is closed, L=27.5 mm represents for the full open. The valve diameter is 50 mm and flow rate varies from 0 to 26 m^3/h, the working fluid is water with a density of 1.01×10^3 $\mathrm{kg/m}^3$, and the working temperature is 25 °C.

3 Experimental Method

Experiments play an important role in the study of fluid machinery, and verify simulation procedures. Hence, in this study, experiments are carried out for the globe control valve.

Fig. 2 is the schematic of the experimental system and the test section. This experimental system is built in accordance with the international standard (EN 1267: 1999), and it consists of a reservoir, two control valves, a centrifugal pump (S80×65, Q=50 m^3/h, H=32 m), a flow meter (Optiflux2100, 0–60 m^3/h), a thermometer (PT100, 0–100 °C), two pressure sensors (S-10, 0–0.4 MPa), a test valve and several pipes. The pipes upstream and downstream the test valve have the same internal diameter (d) with the test valve. The pressure ports are located at 5d upstream the test valve and 10d downstream, respectively. The center lines of pressure ports are horizontal and coincide with those of pipes.

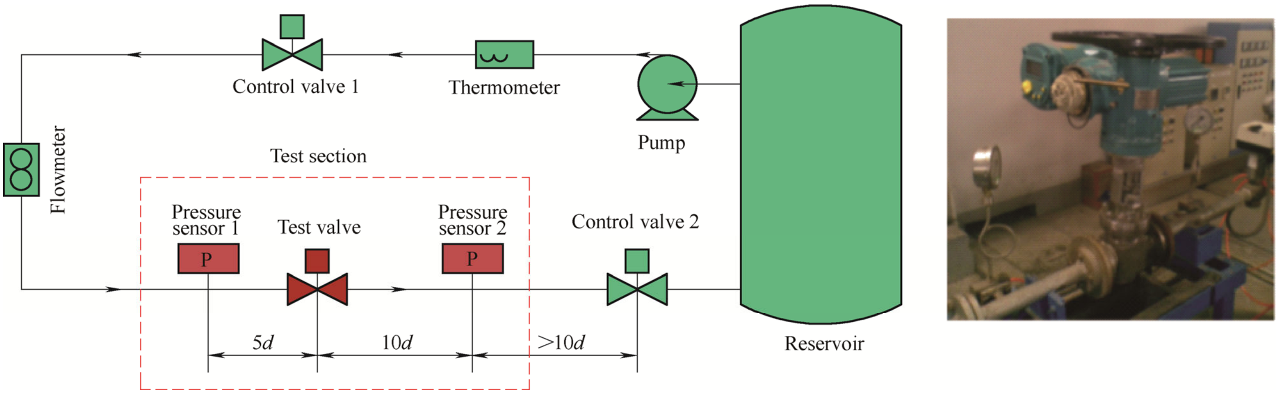

Fig. 2. Flow schematic of the experimental system

Experiments are carried out at 25 °C. The pressures upstream and downstream the test valve are adjusted through the variation of valve opening. The experimental data are recorded only when stable flow is reached. The uncertainties of meters are estimated as follows: the thermometer ±0.3 °C, the pressure sensor ±0.25% and the flow meter ±0.5%. Each experimental process is repeated 5 times to reduce random error.

4 Simulation Modeling

In this study, the commercial software FLUENT (Ansys, Inc., Canonsburg, PA, USA) is employed for numerical calculation. The dynamics of incompressible flow is governed by the time-varying conservation of mass and momentum, namely the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations. The standard k–ε turbulence model is adopted as the Reynolds number has a magnitude of 10^4. The conservation equations for the continuous phase are given by

$$\frac{\partial}{\partial x_i}(\rho u_i)=0, \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\rho\overline{u_i' u_j'}\right], \tag{2}$$

$$\rho u_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k-\rho\varepsilon, \tag{3}$$

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}, \quad (4)$$

where G_k represents the generation of turbulence kinetic energy, μ_t is the turbulent viscosity, σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are equation coefficients.

5 Simulation Procedure and Validation

Following to the international standard of valve test, the computational domain of the present calculation extends 10*d* upstream and 10*d* downstream of the test valve, as shown in Fig. 3. And likewise, the pressure drop between 5*d* upstream the valve and 10*d* downstream are extracted for analysis.

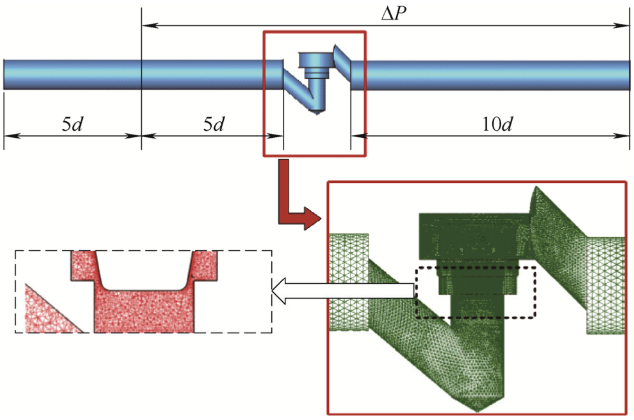

Fig. 3. Computational domain and grid of globe control valve

According to the 3D structure of globe control valve, unstructured mesh technique is adopted for the flow passage. In order to simulate small opening condition accurately, local mesh refinement approach is applied for throttling regions (Fig. 3). In the simulation procedure, an uniform velocity is set for the entry of computation region, and an uniform pressure of 1.01×10^5 Pa (working pressure) is inducted for outlet. The turbulence intensity of 3% and the hydraulic diameter of 50mm are used for both boundaries to describe turbulence. No-slip boundary condition is applied for wall boundaries. The SIMPLEC algorithm of DOORMAAL, et al[32] is used to solve the equations. The Green-Gauss cell-based method is used for the evaluation of gradients and derivatives. The standard pressure interpolation scheme and the second order upwind scheme are used for pressure gradient, convection terms and divergence terms. The convergence of this procedure is obtained when the residual of 10^{-4} is obtained, at which the monitored flow parameters are almost invariably between two consecutive iterations.

In order to validate our computational procedure, a set of experimental data for different valve openings are extracted. As the flow coefficient, denoting as $C_{\rm v}$, is an important parameter to express the performance of valve, it is used for the comparison. $C_{\rm v}$ is determined as

$$C_{\rm v} = Q\sqrt{\frac{\gamma}{\Delta P}}, \quad (5)$$

where Q is the flow rate ($\rm m^3/h$), γ is the specific gravity of the carrier fluid relative to water and ΔP is the pressure drop (Pa) between upstream and downstream pressure measuring point.

A grid independence test was also performed by using the grid numbers of 140 000, 330 000, 470 000, 820 000, 1 300 000, 1 490 000 and 1 700 000, as shown in Fig. 4. The valve opening is 40%, the cone angle is 10° and the flow rate is 7.672 $\rm m^3/h$. Calculation results indicate that the variations in the flow coefficients are less than 1% when the grid number is larger than 820 000. So further mesh refinement leads to negligible changes in flow coefficients. A grid number of 1 300 000 is used for subsequent calculations.

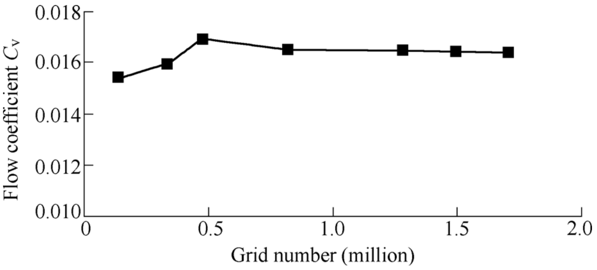

Fig. 4. Comparison of flow coefficients for various grid numbers

Fig. 5 compares the flow coefficients obtained from simulation and experiments. Reasonable agreements are found between simulation and experiments, indicating the computational procedure is suitable.

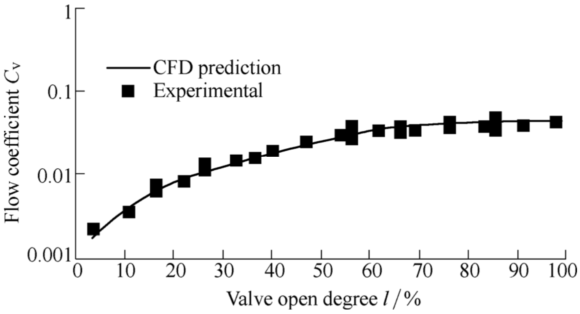

Fig. 5. Comparison of the flow coefficients obtained from simulation and experiments

6 Results and Discussions

In this study, CFD simulations and experiments are used to investigate the globe control valve. We discuss the effect of cone angles on the throttling performance and the internal flow properties of the valve.

6.1 Throttling performance of valve

The pressure-flow rate curves for the opening of 22%, 40%, and 76% are obtained, as shown in Figs. 6–8. It is found that for the cone angles concerned, pressure drop increases almost exponentially with flow rate. This phenomenon is actually due to the conservation of mechanical energy (Bernoulli equation). The figures also show that while valve open is small, pressure drop decreases with the increase of cone angle. This means that the increase of cone angle would leads to the decrease of flow losses, and makes a higher flow efficiency. However, with the increase of valve opening, the effect of cone angle on pressure-flow rate curve decreases. And once valve opening exceeds 76%, the pressure-flow rate curve becomes independent of cone angle.

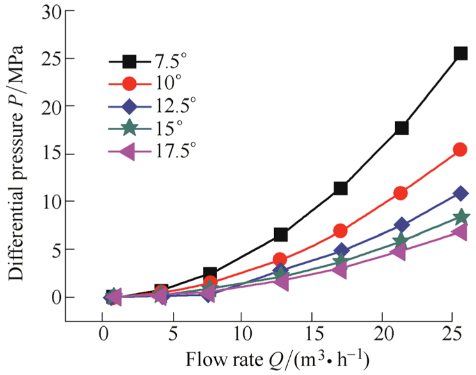

Fig. 6. Pressure-flowrate curves with valve opening of 22%

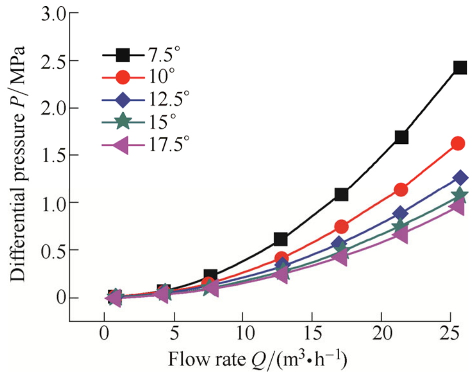

Fig. 7. Pressure-flow rate curves with valve opening of 40%

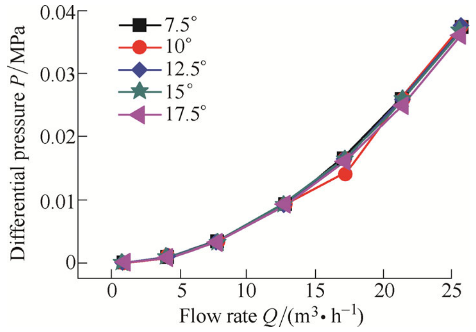

Fig. 8. Pressure-flowrate curves with valve opening of 76%

Since the throttling performance of control valve is mainly reflected by flow coefficient, C_v curves for the examined cone angles are calculated and drawn, as shown in Fig. 9. The curves show that the flow coefficient increases with the opening of valve until the opening reaches 70%, then it remains almost steady. This means the throttling performance of the globe control valve is dominated by the valve opening less than 70%, and the rest 30% barely have any contributions. For the cases of valve opening over 70%, flow coefficients coincide. That is, the cone angle has little effect on valve performance in these conditions. For the cases of valve opening less than 70%, the C_v curve varies from an arc to a straight line, and the change becomes more and more insignificant. Since valve designers may not be familiar with CFD method, these results offer a reference for the design of globe control valves.

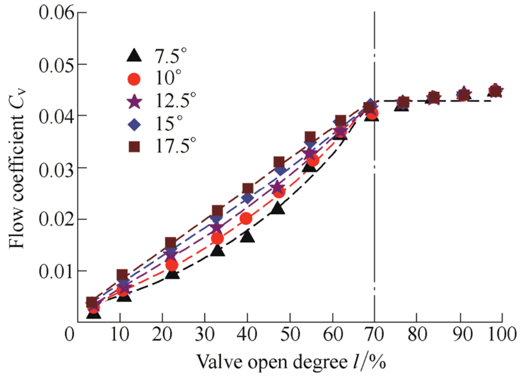

Fig. 9. C_v curves for the examined cone angles

6.2 Internal flow properties

Since valve's throttling performance is caused actually by internal flow, the influence of cone angle on internal flow properties are discussed. By large amount of data processing, it is found that cone angle has negligible effect on flow properties for the valve opening larger than 70%. This leads to the above mentioned results that the effect of cone angle on valve performance is non-significant at these large openings. Thus present part is focused on the cases of valve opening less than 70%. It is also obtained from data processing that for various discussed valve openings and flow rates, cone angle has a similar effect on internal flow. Therefore the valve opening of 40% and the flow rate of 7.672 $\mathrm{m^3/h}$ are adopted for exhibition.

For the studied cone angles, there exists a similar pressure distribution along flow passage. The pressure distribution for the cone angle of 10° is shown in Fig. 10. It clearly shows in the figure that there exists a high-pressure zone upstream the valve. The pressure decreases in the valve region, and reaches the minimum at the outlet of the valve. Then it recovers slightly while the fluid flowing downstream. From the pressure distribution, it is concluded that the globe control valve has an excellent pressure regulation.

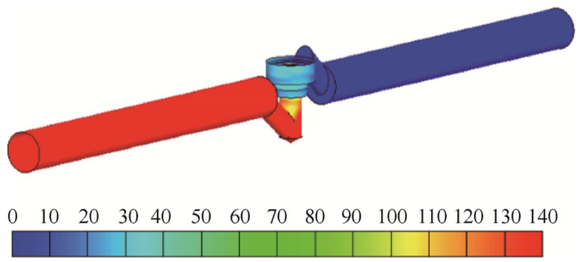

Fig. 10. Pressure distribution along the flow passage for the cone angle of 10° (kPa)

To investigate the effect of cone angle on pressure distribution in the valve area, two typical sections (middle plane of the valve and the cross section T of valve core) are extracted and contrasted, as shown in Fig. 11 and Fig. 12. Fig. 11 shows that the pressure drop of orifice (between valve body and valve plug) is obviously for smaller cone angle, and its value decreases with the increase of cone angle. As the inlet pressure of valve is fixed in the practical usages, the valve plug with smaller cone angle is thus easier to make the pressure downstream the orifice lower than the saturated vapor pressure, and then causes the production of bubbles. Meanwhile, the quick recovery of pressure in the subsequent region indicates a high risk of bubble collapse once the cavitation bubbles exist. Thus, it is concluded that the increase of cone angle is beneficial for the anti-cavitation performance of the globe control valve. Fig. 12 points out that the pressure distributions of cross section T for the discussed cone angles are equal to each other. This is due to the same outlet condition and same flow rate used in present simulation procedure. This figure further illustrates that the performance of the valve is dominated by the throttling effect of valve orifice.

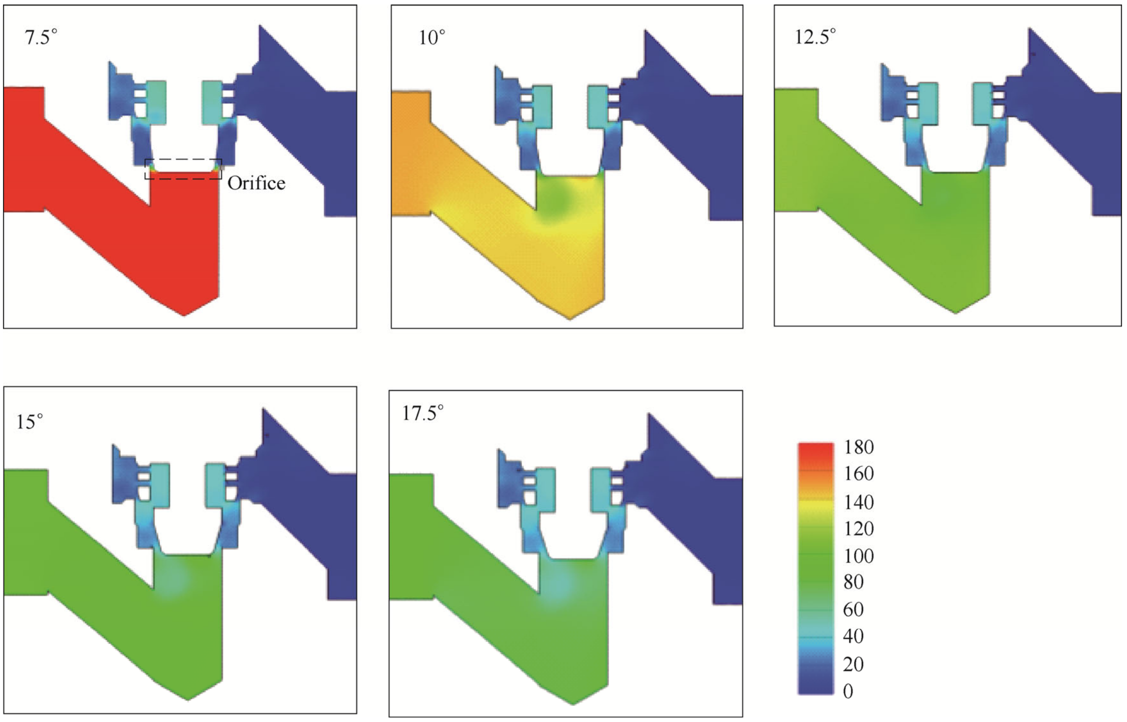

Fig. 11. Pressure distribution in the middle plane of valve for different cone angles (kPa)

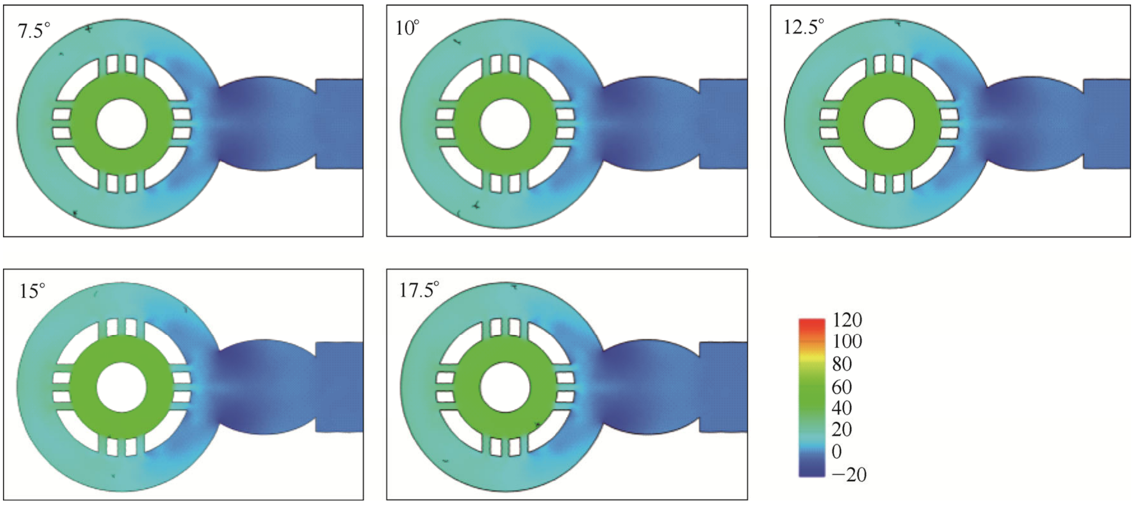

Fig. 12. Pressure distribution in the cross section T of valve core for different cone angles (kPa)

Fig. 13 shows the distribution of flow velocity in the middle plane, while Fig. 14 shows the velocity vector in this plane. Due to the complexity of the valve structure, the distribution is extremely inhomogeneous. Under the effect of pressure drop between the front and back of the orifice, it is obviously that there is a high-speed zone around the conical surface of the valve plug. Meanwhile the value and the scope of this high speed zone decreases with the increase of cone angle, leading to a decreasing intensity of adjacent downstream vortex, and thus causing variations of valve performance. Since the valve is used for flow controlling, local high speed is inevitable. Thus when there are impurities (like particles) contained in the working fluid, the valve plug will face a great risk of particle erosion. Moreover, as ZHANG, et al[33] mentioned that the erosion is an exponential function of particle velocity, the increase of cone angle will be benefit for the anti-erosion of globe control valves.

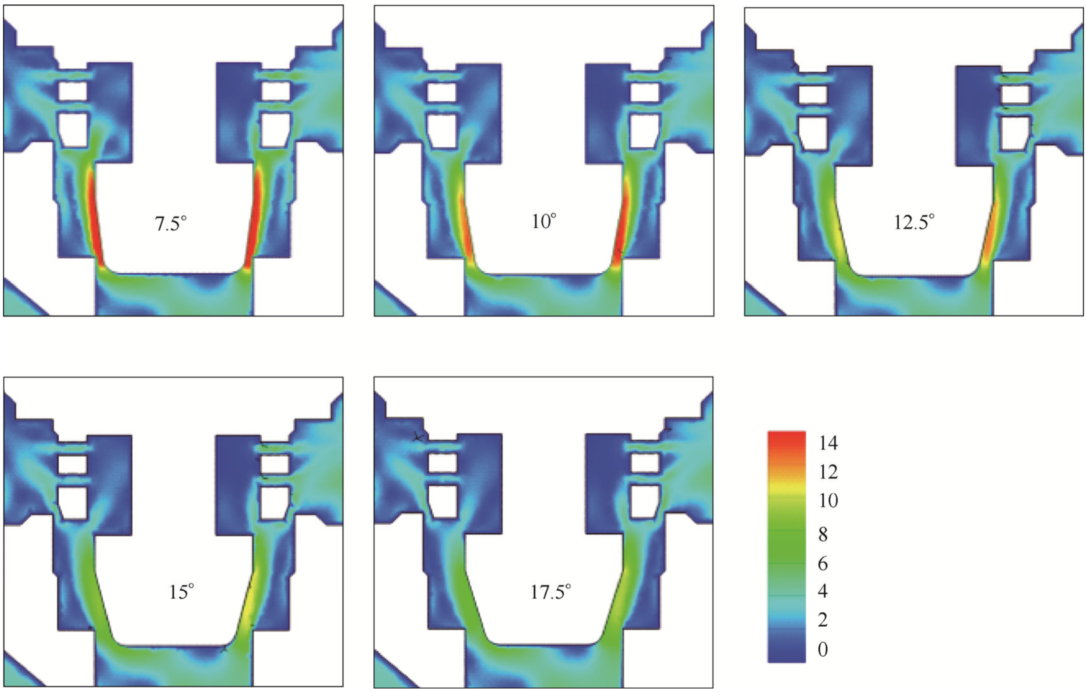

Fig. 13. Velocity distribution in the middle plane of valve for different cone angles (m/s)

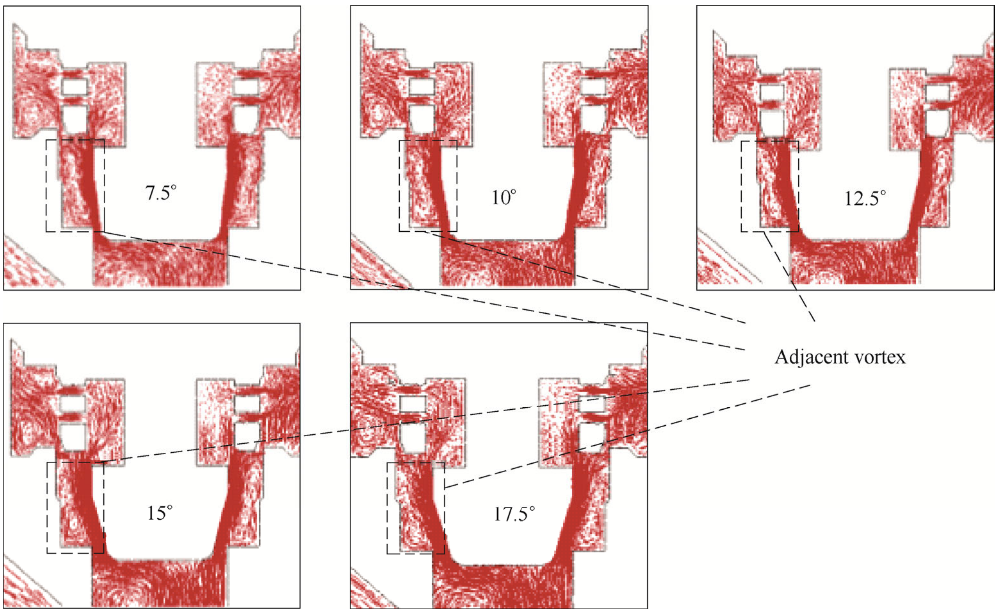

Fig. 14. Velocity vector in the middle plane of valve for different cone angles

Since there are little differences of pressure distribution in the cross section T for different cone angles, velocity distributions in this surface are similar. Hence, the velocity distribution of section T for the cone angle of 10° is analyzed (Fig. 15). Due to the effect of valve structure, the velocity distribution is symmetrical, and the flow velocities around the valve outlet are relatively larger than anywhere else.

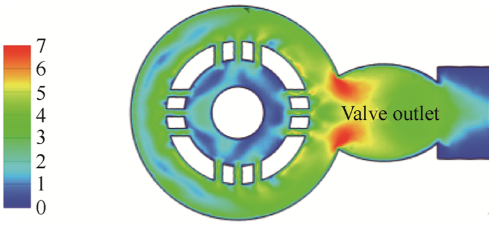

Fig. 15. Velocity distribution in the cross section T of valve core for the cone angle of 10° (m/s)

Similarly, the distributions of vortexes for different cone angles are almost the same. The streamlines in two typical sections for the cone angle of 10° are drawn in Fig. 16 and Fig. 17. There are large vortexes formed in the inlet and outlet regions of valve, and they are the main reasons of hydraulic loss. Meanwhile, as there exist many cavities in the valve flow passage, large quantities of small vortexes appeared. These small vortexes not only lead to unsteady flow parts like secondary flow and back flow inside the valve, but also make the hydraulic loss even larger. Thus, it is appropriate for us to carry out optimization designs of globe control valve, to improve the flow situation of valve and to reduce vortexes inside valve.

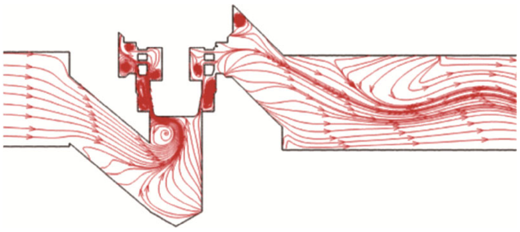

Fig. 16. Flow streamlines in the middle plane of valve for the cone angle of 10°

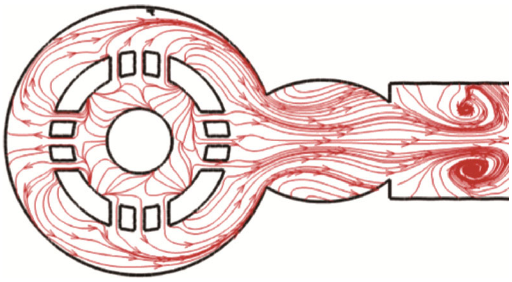

Fig. 17. Flow streamlines in the cross section T of valve core for the cone angle of 10°

7 Conclusions

In this study, the globe control valves are studied by CFD and experimental methods. The influence of cone angles of plug on flow properties and throttling performance are discussed, with valve opening varied. The calculation procedure is validated by comparing CFD results with experimental data.

(1) For the discussed cone angles, pressure drop across the valve increases exponentially with flowrate. While valve opening is small, pressure drop decreases with the increase of cone angle. With the increasing of valve opening, the effect of cone angle on pressure-flowrate curve decreases.

(2) The cone angles have little effect on C_v when valve opening exceeds 70%. For the cases less than 70%, the C_v curve varies from an arc to a straight line, and the change becomes more and more insignificant;

(3) The pressure drop of orifice is more obvious for smaller cone angle, and its value decreases with the enlarging of cone angle. That is the increase of cone angle is beneficial for the anti-cavitation performance of the globe control valve.

(4) It is obvious that there exists a high-speed zone around the conical surface of the valve plug. The value and the scope of this high speed zone decreases with the increase of cone angle, making an decreasing intensity of the adjacent downstream vortex, and thus leading to the variation of valve performances. Meanwhile the increase of cone angle is also beneficial for the anti-erosion performance of the globe control valve.

(5) There are little differences of pressure distribution in the cross section T for different cone angles, as well as velocity distributions. Meanwhile, the distribution of vortexes in the middle plane and cross section T for different cone angles are almost the same.

Based on present study, optimization design of gable control valves will be conducted in the subsequent investigation, to obtain a better throttling performance and flow properties.

References

[1] MOUJAES S, JAGAN R. 3D CFD predictions and experimental comparisons of pressure drop in a ball valve at different partial openings in turbulent flow[J]. *Journal of Energy Engineering*, 2008, 134(1): 24–28.

[2] AN Y J, KIM B J, SHIN B R. Numerical analysis of 3-D flow through LNG marine control valves for their advanced design[J]. *Journal of Mechanical Science and Technology*, 2008, 22(10): 1998–2005.

[3] MORRIS M, DUTTON J. Compressible flowfield characteristics of butterfly valves[J]. *Journal of Fluids Engineering*, 1989, 111(4): 400–407.

[4] LEUTWYLER Z, DALTON C. A CFD study to analyze the aerodynamic torque, lift, and drag forces for a butterfly valve in the mid-stroke position[C]//*Proceedings of the 2004 ASME HTFED Summer Conference*, Charlotte, North Carolina, 2004. American Society of Mechanical Engineers: New York, 2004, HT-FED2004-56016: 41–49.

[5] LEUTWYLER Z, DALTON C. A computational study of torque and forces due to compressible flow on a butterfly valve disk in mid-stroke position[J]. *Journal of Fluids Engineering*, 2006, 128(5): 1074–1082.

[6] LEUTWYLER Z, DALTON C. A CFD study of the flow field, resultant force, and aerodynamic torque on a symmetric disk butterfly valve in a compressible fluid[J]. *Journal of Pressure Vessel Technology*, 2008, 130(2): 021302.

[7] CHERN M J, WANG C C, MA C H. Performance test and flow visualization of ball valve[J]. *Experimental Thermal and Fluid Science*, 2007, 31(6): 505–512.

[8] PALAU S G, GONZALEZ A P, ARVIZA V J. Three-dimensional modeling and geometrical influence on the hydraulic performance of a control valve[J]. *Journal of Fluids Engineering*, 2008, 130(1):

011102.1–9.

[9] FERRARI J, LEUTWYLER Z. Fluid flow force measurement under various cavitation state on a globe valve model[C]//*Proceedings of the 2008 ASME Pressure Vessels and Piping Conference*, 2008. American Society of Mechanical Engineers: New York, 2008, PVP2008-61238: 157–165

[10] CHO T D, YANG S M, LEE H Y, et al. A study on the force balance of an unbalanced globe valve[J]. *Journal of Mechanical Science and Technology*, 2007, 21: 814–820.

[11] JAMES A D, MIKE S. Predicting globe control valve performance-Part I: CFD modeling[J]. *Journal of Fluids Engineering*, 2002, 124: 772–777.

[12] JAMES A D, MIKE S. Predicting globe control valve performance-Part II: Experimental verification[J]. *Journal of Fluids Engineering*, 2002, 124: 778–783.

[13] WANG Y, ZHU Y, SHEN X R, et al. Numerical simulation and experimental research of a new butterfly valve[J]. *Applied Mechanics and Materials*, 2012, 212: 1255–1260.

[14] SONG X G, WANGL, PARK Y. Analysis and optimization of a butterfly valve disc[J]. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 2009, 223: 81–89.

[15] WANG L, SONG X G, PARK Y C. Dynamic analysis of three-dimensional flow in the opening process of a single-disc butterfly valve[J]. *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, 2010, 224(2): 329–336.

[16] LIU Y H, JI X W. Simulation of cavitation in rotary valve of hydraulic power steering gear[J]. *Science in China Series E: Technological Sciences*, 2009, 52(11): 3142–3148.

[17] LEE W G, REITZ R D. A numerical investigation of transient flow and cavitation within minisac and valve-covered orifice diesel injector nozzles[J]. *Journal of Engineering for Gas Turbines and Power*, 2010, 132: 052802.1–8.

[18] LIN Z, RUAN X D, CUI B L, et al. 3-D Euler-Lagrange CFD simulation of particle impact on carrier fluid through gate valve[C]//*ASME 2012 Fluids Engineering Division Summer Meeting*, Puerto Rico, 2012. American Society of Mechanical Engineers: New York, 2012, FEDSM2012-72202: 1–5.

[19] LIN Z, RUAN X D, ZHU Z C, et al. Three-dimensional numerical investigation of solid particle erosion in gate valves[J]. *Journal of Mechanical Engineering Science*, 2014, 228(10): 1670–1679.

[20] LIN Z, RUAN X D, ZHU Z C, et al. Numerical study of solid particle erosion in a cavity of different wall heights[J]. *Powder Technology*, 2014, 254: 150–259.

[21] ZHU Z C, GUO X M, CUI B L, et al. External characteristics and internal flow features of a centrifugal pump during rapid startup[J]. *Chinese Journal of Mechanical Engineering*, 2011, 24(5): 798–804.

[22] TAN M G, YUAN S Q, LIU H L, et al. Numerical research on performance prediction for centrifugal pumps[J]. *Chinese Journal of Mechanical Engineering*, 2010, 23(1): 21–26.

[23] SHI W D, ZHANG D S, GUAN X F, et al. Numerical and experimental investigation of high-efficiency axial-flow pump[J]. *Chinese Journal of Mechanical Engineering*, 2010, 23(1): 38–44.

[24] SHIGEMITSU T, FUKUTOMI J. Performance analysis of mini centrifugal pump with splitter blades[J]. *Journal of Thermal Science*, 2013, 22(6): 573–579.

[25] ZHANG Y L, ZHUZC, DOUH S, et al. Experimental and theoretical study of a prototype centrifugal pump during startup period[J]. *International Journal of Turbo & Jet Engines*, 2013, 30(2): 173–177.

[26] ZHANG D. Unsteady flow analysis and experiment investigation of axial-flow pump[J]. *Journal of Hydronamics*, 2010, 22(1): 35–43.

[27] KHALIFA A E. Study of pressure fluctuations and induced vibration at blade-passing frequencies of a double volute pump[J]. *Arab J. Sci. Eng.*, 2011, 36: 1333–1345.

[28] GOURDAIN N, WLASSOW F, OTTAVY X. Effect of tip clearance dimensions and control of unsteady flows in a multi-stage high-pressure compressor[J]. *Journal of Turbomachinery*, 2012, 134(5): 1–30.

[29] WONG C Y, SOLNORDAL C, SWALLOW A, et al. Experimental and computational modelling of solid particle erosion in a pipe annular cavity[J]. *Wear*, 2013, 303(1–2): 109–129.

[30] MATJAZ E, BRANE S, MARKO H, et al. Numerical and experimental investigation of axial fan with trailing edge self-induced blowing[J]. *Journal of Fluids Engineering*, 2009, 131(11): 129–138.

[31] LIN P F, WU D C, ZHU Z F. Nanoparticle transportation and brownian diffusion in planar jet flow via large eddy simulation[J]. *Open Journal of Fluid Dynamics*, 2012, 2(4A): 354–358.

[32] DOORMAAL J V, RAITHBY G. Enhancements of the SIMPLE method for predicting incompressible fluid flows[J]. *Numer., Heat Transfer*, 1984, 7: 147–163.

[33] ZHANG Y, MCLAURY B S, SHIRAZI S A. Improvements of particle near-wall velocity and erosion predictions using a commercial CFD code[J]. *Journal of Fluids Engineering*, 2009, 131(3): 031303-12.

Biographical notes

LIN Zhe, born in 1987, is currently a lecturer at *Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Zhejiang Sci-Tech University, China*. He received his PhD degree on mechatronics from *Zhejiang University, China*, in 2013. His research interests include fluid engineering and mechatronic engineering.
Tel: +86-571-86843348; E-mail: linzhe0122@gmail.com

WANG Huijie, born in 1991, is currently a ME candidate at *Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Zhejiang Sci-Tech University, China*. His research interests include fluid engineering and valves.
Tel: +86-571-86843348; E-mail: wanghuijie0107@126.com

SHANG Zhaohui, born in 1986, is currently a junior engineer. He received his ME degree on mechatronics from *Zhejiang Sci-Tech University, China*, in 2013. His research interests include fluid engineering and valves.
Tel: +86-571-86843348; E-mail: shangtianbiao@163.com

CUI Baoling, born in 1975, is currently a professor at *Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Zhejiang Sci-Tech University, China*. She received her PhD degree from *Zhejiang University, China*, in 2006. Her research interests include fluid engineering and mechatronic engineering.
Tel: +86-571-86843348; E-mail: blcui@zstu.edu.cn

ZHU Chongxi, born in 1984, is currently a technical support engineer at *Zhejiang Windey Co., Ltd., China*. He received his bachelor degree on automation from *Zhejiang Sci-Tech University, China*, in 2007. His research interests include fluid engineering.
Tel: +86-571-87397666; E-mail: zhucx@chinawindey.com

ZHU Zuchao, born in 1966, is currently a professor at *Zhejiang Provincial Key Laboratory of Fluid Transmission Technology, Zhejiang Sci-Tech University, China*. He received his PhD degree from *Zhejiang University, China*, in 1997. His research interests include fluid engineering and mechatronic engineering.
Tel: +86-571-86843348; E-mail: zuczhu@163.com