IOP Conf. Series: Materials Science and Engineering **129** (2016) 012040 doi:10.1088/1757-899X/129/1/012040

Numerical simulation analysis and optimum design for combined type pressure reducing valves

D M Gou[1], P C Guo[1], X B Zheng[1], X Q Luo[1] and L G Sun[1]

[1] State Key Laboratory Base of Eco-hydraulic Engineering in Arid Area, Xi'an University of Technology, Xi'an 710048, Shaanxi Province, China

Email:guoyicheng@126.com

Abstract.Pressure reducing valve is an extremely significant equipment of energy dissipation for the water supply by gravity with pressure reducing technology in hydropower stations, and which has a pronounced effect on the normal technical water supply even safety operation for the hydropower units. A three-dimensional numerical calculation of flow field and cavitation characteristics towards a combined type pressure reducing valves was carried out based on the system of technical water supply in this paper. The numerical results show that the investigated valve could meet the requirements of technological supply water pressure and great pressure loss was caused when the water flow was accelerated by narrow overflowing section between throttling cone and valve seat. At working operation, obvious cavitation phenomenon was observed on the surface of throttling cone, and the maximum volume fraction of vapor reached 0.537%. Based on above researches, this paper introduces an optimization model for profile line design of throttling cone. The optimal results show that the cavitation performance is effectively improved with identical pressure drop compared with original results.

1. Introduction

The supplied water of the water supply in hydropower stations is for various equipment which is working, including air cooler and bearing cooler for hydroelectric generators, water cooling transformer cooler and air compressor cooler for hydraulic turbines and so forth. The main function and role of technical water supply is applied to the cooling and lubrication of all kinds of facilities consumed water, which have some requirements for water, such as water quantity, water temperature, water pressure and water quality. The application of water supply by gravity with pressure reducing technology to hydraulic station with high head is becoming a new trend because of its advantages of simple equipment, no rotating parts, stable operation, convenient maintenance and less investment, more importantly, this way shows reliable technique and better economic benefit, thus is increasingly enjoyed by engineering designers in high-head hydro plants[1].

For the water supply by gravity with pressure reducing technology of high head stations, the water intake is generally located in both sides of the steel penstock with an inclination angle of 45 degree. Given water pressure of water intake is far higher than the technical water supply required, it is easy to see it is essential to install a reliable depressurizing facilities, that is the pressure reducing valve, to release excess pressure. The selection and application of pressure reducing valves turn out to have an even more important bearing on the water supply system of gravity flow for reducing pressure, which will provide appropriate cooling water with lower maintenance costs and labor intensity in a very long time.

At present, numerical simulation analysis technology has gradually become an important auxiliary tool for engineering design since the rapid development of computer technology. Performance characteristic of the valve can be designed and optimized by engineers with this technology, which can help avoid blind experiment and make the design more reasonable. In this way, the manufacturer could fully utilize limited development resources to develop high quality valve products and improve the competitiveness of product[2]. In the literature[3], an internal flow numerical simulation analysis was carried out towards a pressure reducing valve with high pressure difference, the results obtained show that numerical simulation technology can be used to stimulate the complex flow of the valve, furthermore, an optimization scheme was posed on the basis of numerical results, which expected to lengthen the lifespan of valve. According to the structure and working principle of the pilot diaphragm valve, a mathematical model of this valve is established, and the hydraulic transient process of the water pipeline with pressure reducing valve is calculated in literature [4], the results show that the maximum water hammer pressure of the relief valve is significantly reduced, and the pressure valve on the inlet and outlet of the valve and its change process is obviously different. Literature[5] took a three-dimensional transient flow computation in the annular orifice of pressure reducing valve applied in the rocket attitude control system and obtained some parametric drawings about the flow fields. Literature[6] applied the finite volume method of nonstructural and non staggered grid to simulate separated flows towards three kinds of common valves in the pipeline system, the results show that with the decrease of the valve opening, two vortices were formed in the back of the butterfly valve and two sides of ball valve, further experiments indicate the downstream noise induced by water flow is greater than the upstream and the noise mainly originated from the vortex sound. Based on two-phase cavitating flow control equation and RNG k-ε turbulence model, literature[7] numerically analyzed cavitation erosion and cavitation characteristics of a liquid control valve, the numerical investigation is in agreement with the experimental ones, the flow velocity increase sharply when the fluid flows through the position between the throttling core and valve seat, which lead to a sudden pressure lowering and induced cavitation phenomenon. Literature[8] qualitatively presented force situation of important parts of a butterfly valve under different opening by using the commercial software Fluent, the numerical results show that the flow pattern is better at the position of fully open and large opening, the subnormal pressure areas can be observed in the posterior of valve plate , where the velocity gradient increase evidently. In literature[9], the generation model of valve flow noise in gas transmission pipeline is established, and the steady and unsteady flow field of gas flow through the valve is simulated, the results show that the fluid turbulent fluctuations is the fundamental reason of the fluid noise, in addition, the simulation results are verified by experiments, which show that the feasibility of application of Fluent to simulate the flow field and acoustic field. The authors study the fully automatic performance and life test system with a special interest on the comparatively analysis of internal flow fields before and after retrofitting the profile line of the valve cone in literature [10], the optimization results indicate that modified curved surface type cone can achieve steady flow with a reduction in drag losses.

In this paper, a three-dimensional numerical calculation of the pressure reducing valve of the technical water supply system is carried out, and the internal flow characteristics of the valve are analyzed. Based on the steady calculation results, the cavitation performance is investigated by FLUENT software. According to the analysis results of cavitation performance of the valve, the throttle cone profile that vitally influence the valve performance is modified by profile line optimizing. Furthermore, a comparative analysis of the flow field characteristics and cavitation characteristics between the original results and optimized numerical results are performed.

2. Calculation model and grid generation

The combined pressure reducing valve are highly thought by users owing to their high ratio of reduced pressure, large flow, strong adaptability to inlet pressure , low noise and long service life. This type valves are comprised of main valve and its corresponding feedback and realize automatically regulation to keep the outlet pressure always the formerly adjusted low-pressure valve.

IOP Conf. Series: Materials Science and Engineering **129** (2016) 012040 doi:10.1088/1757-899X/129/1/012040

Figure 1 shows the overall assembly of the combined type pressure reducing valve, which consists of body, valve seat, spindle and throttling cone, and whose basic parameters are listed in table 1. Three-dimensional modeling software is utilized to generate calculation model by means of surface rotation, stretching, fillet and so on. Figure 2 presents the three-dimensional sectional drawing of the investigated valve and the assembly relationship of throttle cone and valve seat. The working principle of the valve also shows in figure 2, the flow rate and outlet pressure were regulated by changing the openings of throttle cone, which would reduce the outlet pressure with decreasing area of passage. The valve would realize automatically adjustment of rated pressure and flow rate by the energy of the flow medium itself when variation of pressure or flow rate in outlet were detected, so as to ensure the outlet pressure an original preset value.

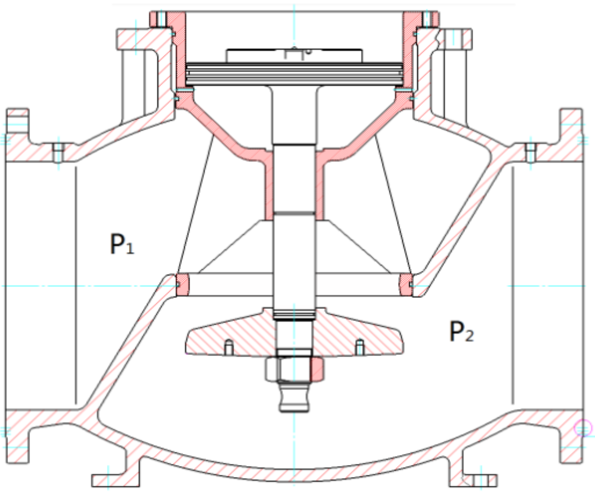


Figure 1. The assembly view of the model

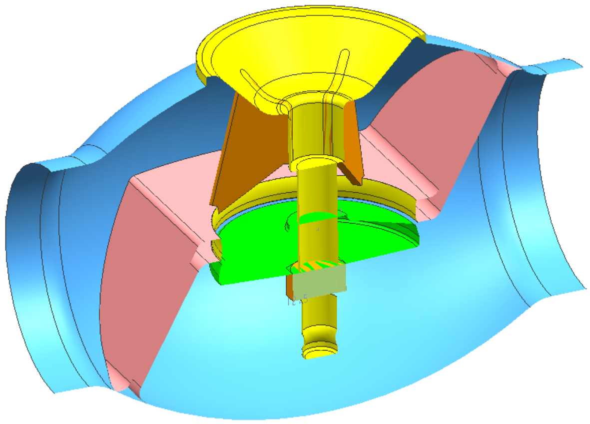
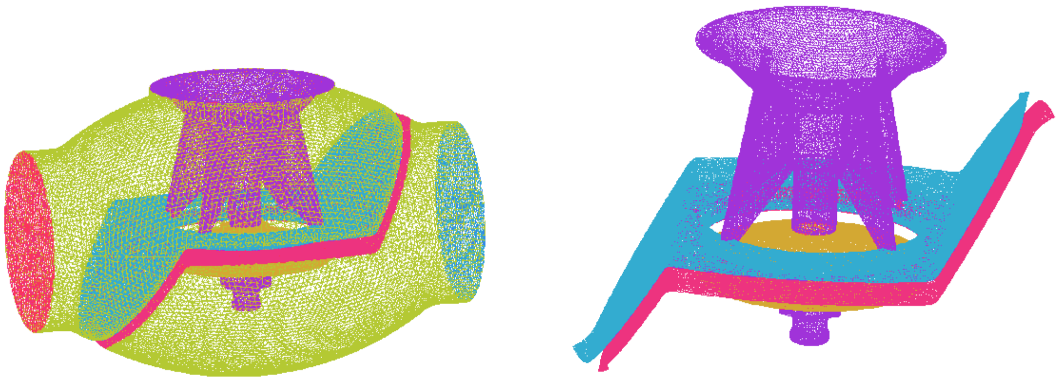


Figure 2. The 3-d sectional drawing

Table 1. The basic parameters of pressure reducing valve

Parameters	Remark	Parameters	Remark
Nominal diameter(DN)	450mm	Nominal pressure(PN)	2.5MPa
Inlet pressure(P_1)	1.56-2Mpa	Outlet pressure(P_2)	0.6MPa
Flow rate(Q)	1600m^3/h	Opening(H)	28mm

The generation of computational grid is the most crucial and time-consuming tasks, which accounts for the majority of the workload for the numerical calculation. The quality of the grid has direct affect on the numerical calculation accuracy and the reliability of solution results. Also, the accuracy and convergence rate of numerical calculation have a good advance by using a set of high quality grid. For the computational model of the pressure reducing valve, the grid generation tool ICEM-CFD is used to create the unstructured grids for the calculation domains because of good adaptability to complex model. In the addition, the method of local mesh refinement of critical region has been introduced in this paper for more credible flow calculation results, for example the end surface of the throttle cone, which seriously influence the valve flow pattern. Figure 3 shows the entire schematic diagram of the computational grid and the internal grid. The total number of grid nodes is 314,000, and the tetrahedral mesh is 1,800,000.



(a) Entire gird (b) Internal grid

Figure 3. Grid generation of the investigated pressure reducing valve

3. Numerical computational conditions

In order to simulate the water current movement more realistically, it is particularly essential to keep the flow pattern in inlet and outlet of valve as much as steady, therefore the valve were equipped with two sufficiently long pipeline on inlet and outlet. As a general rule, the length of the pipelines are setup as 5~10 times of the DN(nominal diameter=450mm) on the basis of the engineering experiment we have learned, in this paper the length of the pipes is 5 meters respectively for the investigated domain. The numerical model of ANSYS CFX is shown in figure 4.

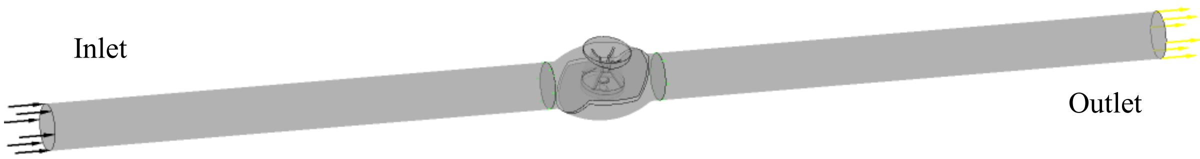


Figure 4. Numerical model of the investigated valve

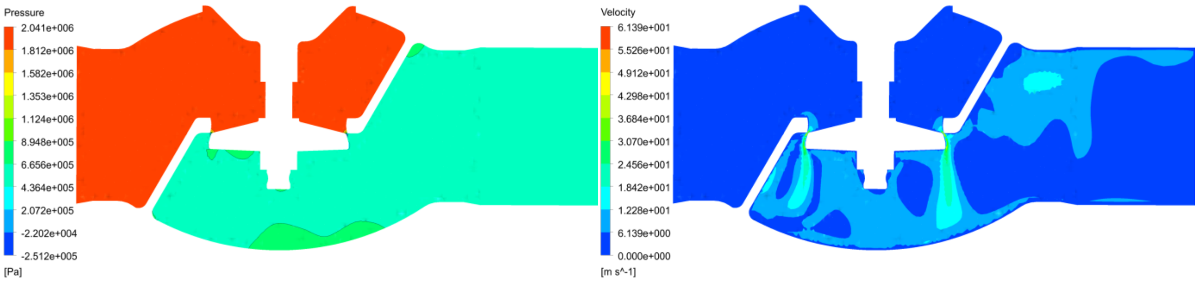
The flow field and cavitation numerical computation are carried out using ANSYS Fluent, given an inlet total pressure and a mass flow rate at outlet, the average Reynolds equation is closed with the standard k-ε turbulence model. In the near wall region automatic near wall treatments were employed, at the same time, no-slip and smooth conditions for solid boundary, and GGI(General Grid Interface) connections for the interfaces between the valve and pipes.

On the other hand, the “Schnerr-Sauer” cavitation model based on homogeneous muti-phase transport equation was employed for cavitation simulation with vaporization pressure 3540Pa, and bubble number density 1e+13, as well as zero water vapor volume fraction of inlet and outlet.

3. Numerical results analysis

3.1. Flow field characteristic analysis

Figure 5 shows the valve pressure distribution and velocity distribution on the symmetry plane at the working condition of opening 28mm, as is shown in figure, inlet pressure of the valve uniformly distributes and outlet pressure is set around 0.6Mpa. The pressure drop of pressure reducing valve mainly concentrate in the narrow flow passage between the throttle cone and valve seat, where observes obvious pressure gradient, which shows that the flow pattern of the region is relatively not stable, and presents significant pressure decrement.

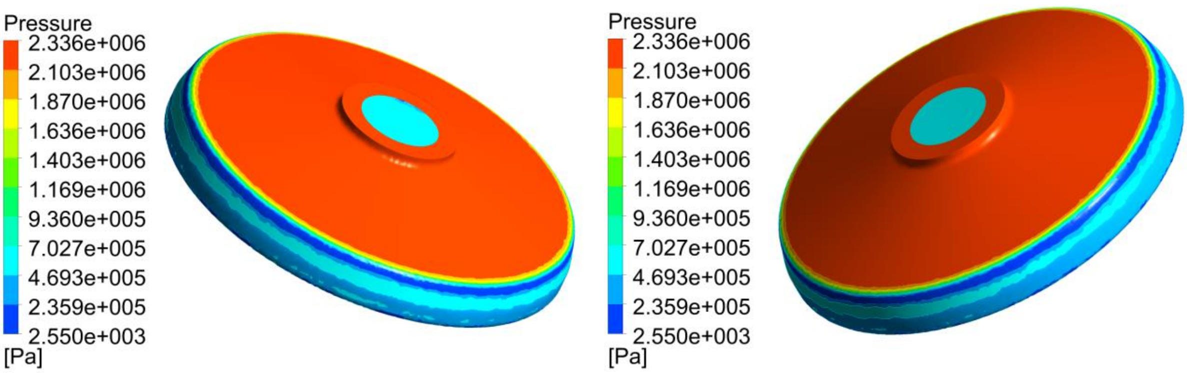


(a) Pressure distribution (b) Velocity distribution

Figure 5. Pressure and velocity distributions on the symmetry plane at H=28mm

For the velocity distribution, the influence of incoming originated form upstream on the inlet velocity seems to be neglected, water flow velocity increased gradually with decreasing area of passage, and it would reach its extreme points finally. The spindle serves as the boundary for the velocity distribution of upstream and downstream as shown in figure 5, high speed water flow directly flows into steel pipe on the right side of the spindle, while it is distributed as vortices at the upstream region. The low-pressure water flows along the superior wall of pipe in a great extent after flow through throttle cone, which leads to a remarkable pressure reduction in this region. Poor operation stability may be caused by the unequilibrium condition of forces acting on throttle cone because of appearance of larger vortices between the region of cone and valve seat, even induces body vibration and damage to the valve parts especially operating at several worst working situations.

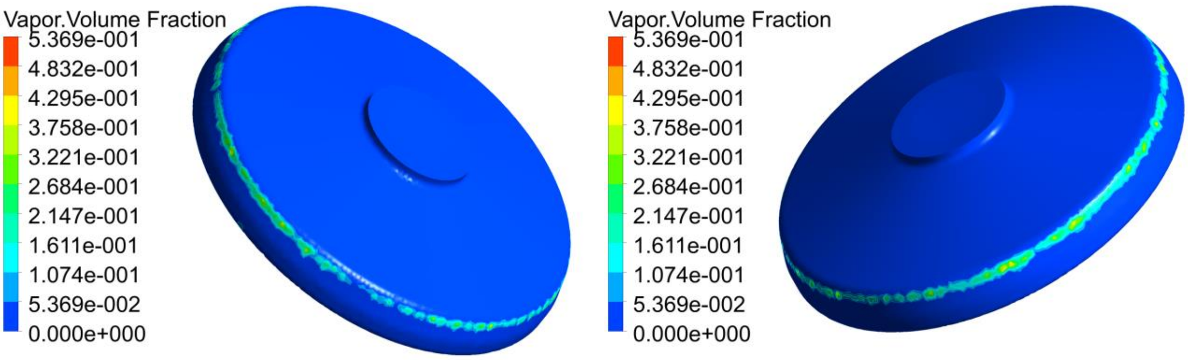
High pressure distribution can be observed on the upper surface of cone owing to the impact action form upstream water, as reported in figure 6 form different views, and the extreme low pressure lies in end face of the cone. The minimum value of pressure is as low as 2550Pa, lower than the vaporization pressure, as a consequence, cavitation phenomenon may occur in the low-pressure region.



(a) Inlet direction (b) Outlet direction

Figure 6. Pressure distribution of the cone from different views

3.2. Cavitation characteristic analysis
Cavitation is the formation of vapor or gas cavities within a given liquid due to pressure drop, whose inception and development may be the origin of several negative effects, such as noise, vibrations, performance alterations, erosion and structural damages. Cavitation generally cause material breakage because of the impact action of collapsed bubble. The cavitation phenomenon of vapor volume fraction can be extracted form numerical results, see figure 7.

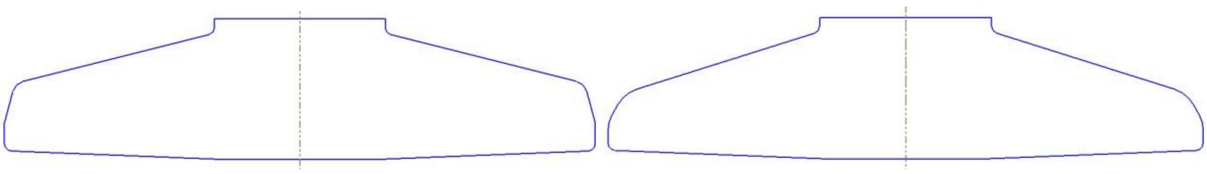


(a) Inlet direction (b) Outlet direction

Figure 7. The vapor volume fraction on the surface of cone from different views

The figure 7 implies the almost all vapors is distributed on the end surface of the cone, and the vapor volume fraction from outlet direction is larger than the inlet direction largely due to greater pressure drop, which has resulted from the high-speed water flow discharging into from upstream to downstream, all of cavitation results present good agreement with that of flow characteristics results. It is apparent that increasingly pressure sink induces more serious cavitation phenomenon. In addition, it’s a failure to observe vapor in the rest of the computational domain.

4. Optimization design of throttle cone profile line
Cavitation phenomenon can be observed in the region where pressure drop to below the vaporization pressure, the cavity destruction which mainly occurs on the solid wall highly cause structure alteration by erosion conversely due to the cumulative effect of impact action where travelling bubbles collapse, which may results in seriously noise and vibrations of the valve, as well as performance degradation, even can’t meet the requirement for technical water supply and increases cost of operation and maintenance. It is expected that to reduce the incidence ratio of cavitation in the premise of meeting the requirement of reducing the pressure depending on the analysis of above flow field and cavitation performance of the valve. The reason of cavitation development on the end surface of the cone consists in significant pressure drop with drastic changes of passage area, therefore, a modification scheme towards profile lines of the cone is given in figure 8 and identical numerical computation about flow field and cavitation is performed at the same level of pressure drop.



(a) original (b) modified

Figure 8. Comparison views of the original and modified cone cross-section

4.1. Comparative analysis of flow characteristics

Pressure distribution comparative contours on the symmetric plane in the region of upstream and downstream are respectively reported in figure 9 and figure 10.

(a) Original (b) Modified

Figure 9. Pressure contour at upstream

(a) Original (b) Modified

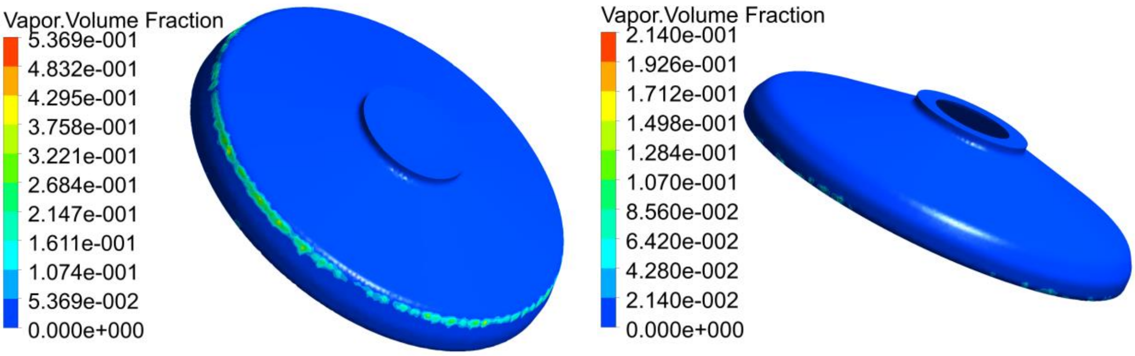
Figure 10. Pressure contour at downstream

It's evident that there is a significantly decrease in the areas of low pressure in figure 9 and figure 10 for the modified throttle cone, which is favorable to inhibit the development of cavitation. As regard the original cone, pressure drop mainly concentrated in the vertical area between the throttling cone and valve seat, where a large number of negative consequences occur such as vortices formation, uneven pressure gradient, unstable flow field and pressure falling rapidly. Contrary to the original results, the improved throttle cone shows better performance in pressure distribution and flow characteristics, and the pressure drop across the region is relatively uniform. All above analysis indicates the optimization design for the throttle cone seems to be reasonable and acceptable.

4.2 Comparative analysis of cavitation characteristics

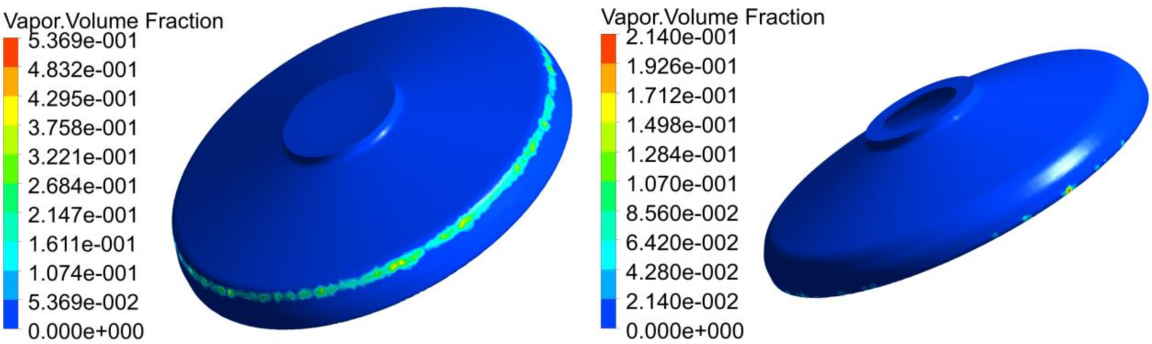
The vapor volume fraction on the surface of the throttle cone are reported in figures 11 and figure 12 for the original model and optimized one form different views, which illustrates improvement of the cavitation performance with a drop of 60% maximum vapor volume fraction from 0.537% to 0.214%.

The numerical results show that it has attain the expected purpose to improve cavitation performance of the pressure reducing valve and raised reliability of technology water supply.



(a) Original (b) Modified

Figure 11. The vapor volume fraction on the surface of cone at inlet direction



(a) Original (b) Modified

Figure 12. The vapor volume fraction on the surface of cone at outlet direction

Compared with the original calculation model, inception and development of cavitation that strongly related to the operation stability of the valve significantly decreases for the modified model and no vapor on the end surface of the cone, only a slight degree of vapor in the vicinity of lower surface of the cone, which indicates that the investigated valve performance is very sensitive to the profile line of the cone and more smooth flow section effectively inhibits the occurrence and development of cavitation. It turns out that the optimization approach towards the throttle cone is successful and can be for reference.

5. Conclusion

In this paper, the three-dimensional numerical calculation of the pressure reducing valve of the technical water supply system is carried out and flow fields and cavitation characteristics are analyzed. The paper finally emphasizes the importance of the optimization approach to the throttle cone and conducts a comparative analysis between the original model and optimized one and draw some conclusions, which should include:

(1) Great pressure loss is caused when the water flow was accelerated by narrow overflowing section between throttling cone and valve seat, it forms serious vortices in the bottom of the valve. In terms of the cavitation cases, almost all vapors is distributed on the end surface of the cone, and the

vapor volume fraction from outlet direction is larger than the inlet direction.

(2) Profile optimization design and performance analysis on the improved model show inception and development of cavitation on throttle cone significantly decreases and offers valuable reference for the optimization design of this type of pressure reducing valve.

6. Reference

[1] Fan X H 1987 *Auxiliary equipment of hydraulic units* (Beijing: China WaterPower Press)

[2] Sun Q and Zheng J L 2013 Application of numerical simulation analysis technology to valve design *J GM in Electric Power.* **(9)** 67-69

[3] Zhang X W, Lan J H and He J H 2011 High-pressure difference pressure reducing valve numerical simulation *J Chemical Equipment Technology.* **32(9)** 1-3

[4] Zhang Y H, Zhou X D, Zhu M L and Wang T 2007 Numerical analysis of pressure reducing valve in the process of hydraulic transient *J Journal of Xi'an University of Technology.* **23(4)** 360-364

[5] Wang D J, Song H L, Bai S Q and Wei C 2009 Simulation and analysis of the regulator orifice flow field *J Journal of Rocket Propulsion.* **36(3)** 37-46

[6] Wu S and Zhang W P 2005 Investigated numerically on flow field of valves and experimental study of valve noise *J Valve.* **(1)** 7-10

[7] Wang L, Ou G F and Zheng Z J 2013 Numerical analysis on the cavitation and cavitation erosion of the high temperature and large pressure differential fluid control valve *J Hydraulic Pneumatics & Seals* **(6)** 40-43

[8] Huang G Q and Cao Z W 2011 The research on numerical simulation of centric type buttery valve flow field *J Machinery design & Manufacture.* **(7)** 186-188

[9] Liu C W, Li Y X, Li X J and Cao J 2012 Flow noise simulation of gas transmission pipeline based on CFD *J Oil & Gas Storage and Transportation* **31(9)** 657-662

[10] Yu X M, Mao Z M and Kong B L 2009 Computer assistant test and numerical simulation analysis for pressure reducing valve *J J. University of Shanghai for Science and Technology.* **31(2)** 183-189