

Research on Flow and Erosion Characteristics of Supercritical Regulating Valve

Yongqiang Pan (0009-0004-4242-1177), Qiong Liu (0009-0003-7289-4517)*, Yue Tang (0009-0004-0331-9065)
Aviation Equipment Manufacturing Industry College, Chengdu Aeronautic Polytechnic University, Chengdu,
610100, China. *E-mail: tangyue_2020@163.com

The regulating valve is a key component of the hydraulic control system. The flow characteristics of the regulating valve are an important parameter for the structural design of the regulating valve. The flow coefficient of the regulating valve directly reflects its flow characteristics, so it is significant to calculate the flow coefficient of the regulating valve accurately. Generally, the flow coefficient is determined by the empirical formula method or experimental method, and the accuracy and efficiency of the calculation are relatively low. In this study, a method for calculating the flow coefficient of the regulating valve combined with numerical simulation using CFX is proposed, and the correctness of the method is verified by experiments. Taking the measured flow coefficient as reference, the maximum error is 4.3%. In addition, the cavitation numerical simulation of the regulating valve pipeline system based on CFX is carried out. Under inlet pressures of 1.96 MPa, 2.46 MPa, and 2.96 MPa, the volume fraction of liquid water decreased to 94.8%, 92.1%, and 88.1%, respectively. At an inlet pressure of 1.96 MPa, the maximum liquid flow exit velocity reached 400 m/s. A prediction method for the erosion amount of the bottom plate is proposed. The predicted erosion amount of the regulating valve reached 10.88 g after 110 hours of operation. The effectiveness of the proposed method is verified by calculating the actual erosion amount. In this study, the calculation method of flow coefficient and the prediction method of erosion amount can assist in improving the structural design method of regulating valve and its pipeline system, and shorten the design cycle of regulating valve.

Keywords: Regulating valve, Flow coefficient, Cavitation, Erosion

1 Introduction

The regulating valve is the key component of the hydraulic control system, which can adjust the pressure and flow of the pipeline system. The flow characteristics of the regulating valve directly affect the regulation of the pressure and flow of the pipeline system. The flow characteristics of the regulating valve refer to the relationship between flow and pressure under different opening degrees of the valve, which is generally described quantitatively by the flow coefficient, so it is very important to accurately determine the flow coefficient for the normal operation of the hydraulic system [1-3]. In practical engineering, due to the influence of the complex structure of the valve body, fluid properties, and working conditions, the variation trend of the flow coefficient of regulating valves under different opening degrees is nonlinear [4-5]. In order to accurately calculate the flow coefficient of the valve, empirical formulas-based methods or experimental methods are adopted. In most of these methods, the fluid is assumed to be an incompressible and stable fluid, ignoring the fluid viscosity and other non-linear factors. These methods are suitable for relatively simple working conditions, and the accuracy is not enough for complicated calculations under complex working conditions [6-8].

Most current approaches for calculating flow coefficients primarily rely on theoretical estimations or empirical formulas, which demonstrate reasonable applicability for simple fluid domains. However, these methods yield less accurate results when applied to complex fluid domains such as regulating valve, as they fail to adequately account for turbulent flow characteristics and so on. While experimental methods can provide direct measurements of flow coefficients, they require the valve to be first designed and manufactured before testing. If results are unsatisfactory, the design must be modified and retested, leading to prolonged development cycles and reduced efficiency.

At present, the working conditions of many regulating valves are relatively harsh, and many of them work in high-temperature and high-pressure pipeline systems. Under such conditions, the pipeline system is prone to cavitation. When the local pressure in the fluid is lower than the saturated vapor pressure of the flowing medium at the current temperature, cavitation occurs, followed by bubbles, bubble growth, and collapse, causing erosion to the pipeline system [9-10]. Once cavitation occurs in the pipeline system, it will not only produce noise and vibration but also reduce the service life of the regulating valve and the pipeline system, resulting in equipment failure or other safety accidents [11]. The cavitation

phenomenon involves the flow of multiphase flow, and its mechanism is complex and difficult to study [12]. Cavitation occurs in the pipeline system of some regulating valves, which then causes serious erosion of the pipeline system. It is necessary to analyze the causes of the erosion. Due to the complexity of cavitation, most of the research methods about cavitation depend on experiments. However, most experimental studies based on engineering applications can only obtain the erosion wear results, while analyzing the underlying causes proves challenging. The experimental approach to study the causes of the erosion require specialized test with high costs, and is limited to specific fluid types. Changing the fluid medium need additional considerations regarding cost and safety factors. Furthermore, current wear prediction research primarily focuses on solid particle erosion in valves, with relatively few studies addressing wear caused by gas-liquid multiphase flows. With the frequent occurrence of cavitation in engineering, numerical simulation can be applied to analyze cavitation.

In this study, the flow coefficient of a regulating valve is calculated by numerical simulation, and compared with the measured data. Then, for the erosion damage in the pipeline system, the cavitation numerical simulation is used to analyze the erosion damage of a bottom plate in the pipeline system. A method for predicting the amount of erosion is proposed.

2 Flow characteristic analysis of the regulating valve

The flow characteristic is a fundamental property of regulating valves, and the analysis of flow characteristics should be prioritized. The most representative parameter for flow characteristics is the flow coefficient calculation of regulating valves. This coefficient is intrinsically related to the internal structure of the valve and serves as a crucial parameter guiding structural design and optimization. After completing the flow coefficient calculation, subsequent analyses should address special conditions such as cavitation phenomena.

2.1 Fluid domain modeling and numerical calculation

In this section, the flow coefficient of a regulating valve is calculated by numerical calculation. First, the model of the valve body is established using SolidWorks, as shown in Figure 1.

Different from the direct establishment of a solid part, the fluid is full of the entire valve body, so it is necessary to utilize other methods to model the fluid domain. If the fluid domain is modeled directly, the modeling process is complex, and is easy to make mistakes. On the basis of establishing the model of the

valve body in Figure 1, the fluid is filled with the whole valve body, and the fluid domain model is obtained through Boolean operations in the modeling module of ABAQUS software, as shown in Figure 2. The model of the fluid domain is symmetrical, so the semi-model is used for subsequent simulation.

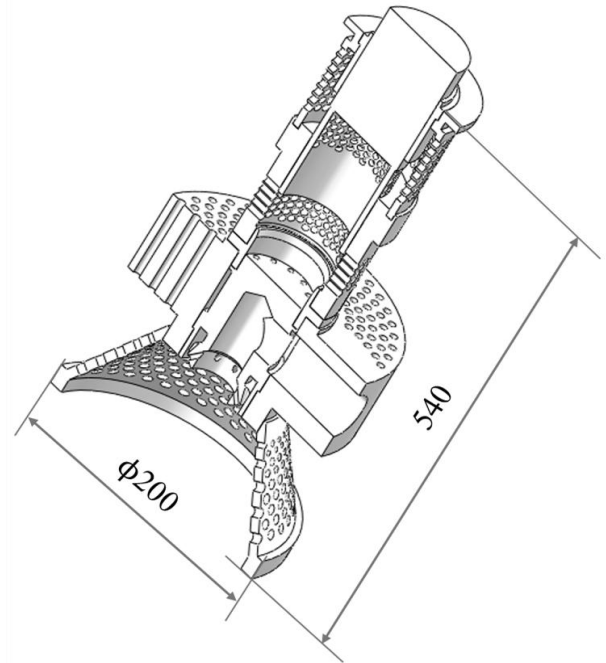


Fig. 1 The model of the valve

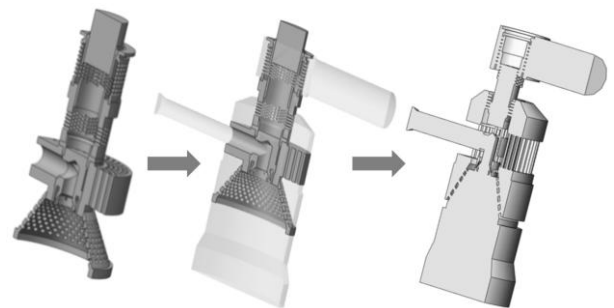


Fig. 2 The process of fluid domain modeling

Once the model of the fluid is established, the CFX software is used to calculate pressure and velocity fields in the domain. As recommend in JB/T 5296: Valves-Test method of flow coefficient and flow resistance coefficient, the parameters for calculating the flow coefficient are summarized in Table 1. Water at 20°C is selected as the medium, and the k- ϵ turbulence model is adopted. Boundary conditions include an inlet pressure of 0.035 MPa and an outlet pressure of 0 MPa. Symmetry boundary conditions are applied on the symmetric plane. The parameters that are used in the simulation are summarized in Table 1. The fluid domain model is meshed using Gambit software, as illustrated in Figure 3. In the irregular regions of the fluid domain, tetrahedral meshes are employed with appropriate mesh refinement.

In contrast, hexahedral meshes are adopted for the remaining regular regions, where the mesh is appropriately coarsened. The mesh element sizes varied approximately from 10 mm³ to 40 mm³.

Tab. 1 Parameters used in the simulation

Inlet pressure	Outlet pressure	Fluid temperature	Fluid density	Dynamic viscosity
0.035 Mpa	0 Mpa	20 °C	998.2 kg/m ³	1.0016 E-3kg/(m·s)

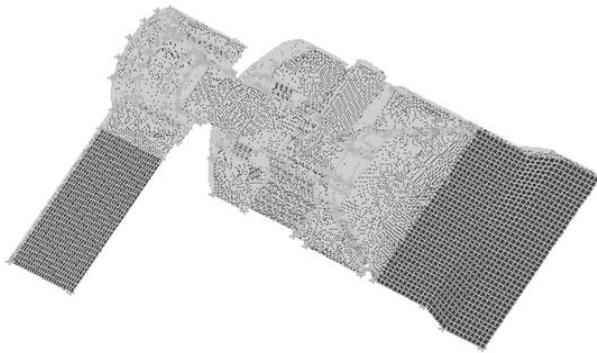


Fig. 3 Grid mesh of fluid domain

2.2 Flow characteristics analysis and validation

The flow coefficient refers to the volumetric or mass flow rate passing through the valve per unit of time under constant pressure. The opening degree of the valve is expressed by the position of the valve core inside the valve body, usually expressed by the percentage. When the valve is fully closed, the opening degree is 0%, and when the valve is fully open, the opening degree is 100%. The reason a valve can regulate pressure and flow rate is that it adjusts the valve opening degree by controlling the movement of the valve core. In order to analyze the flow characteristics, the valve opening degree is varied starting from 5% and 10%, then increasing by increments of 10% until 100%. Numerical simulations are conducted for each of these opening degrees. As depicted in Figure 4, the pressure and velocity contours in the domain are presented when the valve opening degree is 50%.

From Figure 4, it can be seen that the pressure in the domain of the regulating valve decreases when passing through components that have many small

After completion of the preprocessing, the case is submitted to the CFX solver for numerical computation.

holes inside the valve body. Overall, the pressure gradually decreases. From the velocity contour in Figure 4, it can be observed that the flow velocity increases when the fluid passes through the small hole structure. The tendency of pressure and flow rate in the fluid domain under other opening degrees is similar. The flow rate is obtained utilizing the CEL (CFX Expression Language) within CFX post-processing.

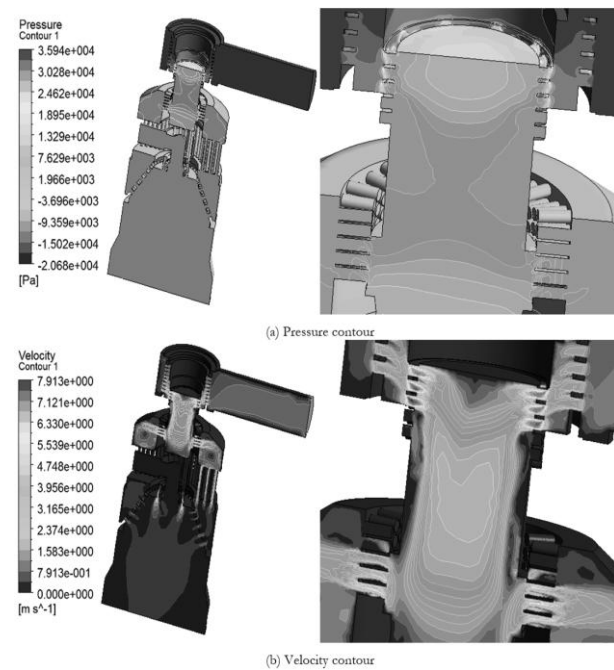


Fig. 4 Pressure and velocity contours in the domain

After obtaining the data of the flow field for all opening degrees, the flow coefficient can be calculated. The calculation method for the flow coefficient of the regulating valve is as follows:

$$Kv = 10Q \sqrt{\frac{\rho/\rho_0}{\Delta P}} (kg/s \cdot kPa^{0.5}) \tag{1}$$

Where:

Q ...The average flow rate. The average flow rate represents the average value of the inlet and outlet flow rates.

ΔP ...The pressure difference,

ρ ...The fluid density,

ρ_0 ...The standard fluid density.

Results are summarized in Table 2, which lists flow coefficients at various opening degrees.

Tab. 2 Flow coefficient for all opening degrees

Opening degree	5%	10%	20%	30%	40%	50%	60%	70%	80%	90%	100%
Q(kg/s)	1.51	3.93	9.76	14.3	19.4	23.1	26.9	30.2	32.6	34.4	35.7
Kv	9.19	23.9	59.4	87.3	118	141	164	184	199	209	218

The obtained flow coefficients are compared with the measured flow coefficients, as shown in Figure 5. The two flow coefficient curves are very close, which proves the correctness of the numerical simulation-based flow coefficient calculation method. Although they are very close, with the increase of the opening degree, the flow coefficient obtained by the simulation is slightly larger than the measured flow coefficient. Taking the measured flow coefficient as reference, the maximum error is 4.3%. It is speculated that there may be some leakage of the regulating valve in practical application, but the two are generally in good agreement.

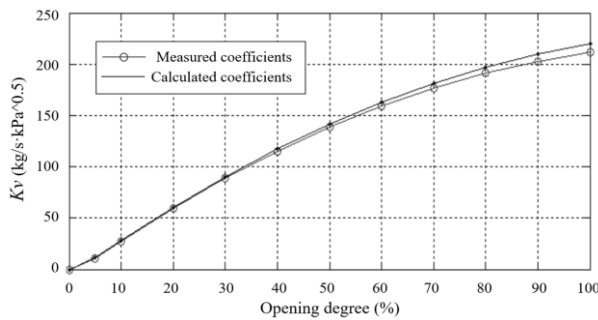


Fig. 5 Comparison between flow coefficients

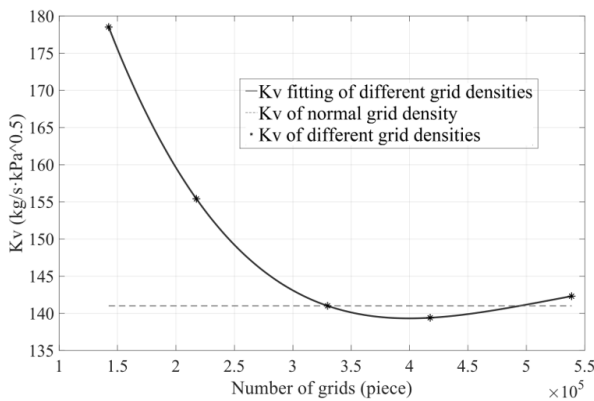


Fig. 6 Grid independence study

A grid independence study is performed at 50% valve opening degree using the computed flow coefficient as the reference. The mesh configurations included two coarse meshes, one normal density mesh, and two fine meshes. Figure 6 presents the correlation between number of grids and the resulting flow coefficient values. After comparative analysis with consideration of computational efficiency, the normal density mesh is ultimately selected for simulations.

3 Erosion analysis and prediction of the pipeline

After performing conventional flow coefficient calculations and implementing the regulating valve in practical engineering, cavitation phenomena were observed in the connected piping system of the valve. This finding indicates that, in addition to analyzing the flow characteristics of the regulating valve, dedicated research on this cavitation phenomenon is necessary. Such investigation will lead to a more comprehensive understanding of the performance characteristics of the valve, thereby facilitating both structural improvements of the regulating valve and its engineering applications.

3.1 Cavitation modeling

In the actual working conditions of the regulating valve, unlike the analysis of its flow characteristics, the pressure and the temperature in the pipeline are usually very high. The internal structure of the regulating valve is so complex that cavitation phenomena may occur. Cavitation generates bubbles, which increase the overall flow velocity of the fluid. The fluid, carrying the bubbles, impacts the piping system, causing erosion.

The Rayleigh-Plesset bubble model is used for simulation, which outlines the basic laws of bubble expansion and contraction during cavitation:

$$R\ddot{R} + \frac{3}{2}\dot{R}^2 + \frac{2\sigma}{\rho R} = \frac{P_v - P}{\rho} \tag{2}$$

Where:

R ...The bubble radius,

ρ ...The density,

P_v ...The saturated vapor pressure,

P ...The local pressure,

σ ...The surface tension coefficient.

The second-order term in the equation describes

the oscillation of cavitation bubbles and the water surface tension on cavitation is negligible, so the equation can be simplified as follows:

$$\dot{R} = \sqrt{\frac{2p_v - p}{3\rho}} \text{ (m/s)} \tag{3}$$

The rate of change of the bubble volume is:

$$\dot{V} = \frac{d}{dt} \left(\frac{4}{3} \pi R^3 \right) = 4\pi R^2 \sqrt{\frac{2p_v - p}{3\rho}} \quad (m^3/s) \quad (4)$$

The rate of change of the bubble mass is:

$$\dot{m} = \rho_g \dot{V} = \rho_g \frac{d}{dt} \left(\frac{4}{3} \pi R^3 \right) = 4\pi R^2 \rho_g \sqrt{\frac{2p_v - p}{3\rho}} \quad (kg/s) \quad (5)$$

If there are N bubbles in a unit volume, the bubble volume fraction r_g can be expressed as:

$$r_g = VN = \frac{4}{3} \pi R^3 N \quad (6)$$

The cavitation rate per unit volume can be expressed as:

$$\dot{m}_f = \dot{m}N = N\rho_g \dot{V} = N\rho_g \frac{d}{dt} \left(\frac{4}{3} \pi R^3 \right) = \frac{3r_g \rho_g}{R} \sqrt{\frac{2p_v - p}{3\rho}} \quad (kg/(m^3 \cdot s)) \quad (7)$$

In a flow system, the tendency for cavitation to occur can be described using the cavitation number, defined as C_a :

$$C_a = \frac{p - p_v}{\frac{1}{2} \rho u^2} \quad (8)$$

Where:

p ...The inlet pressure in a flow system,

p_v ...The saturated vapor pressure.

It can be deduced that the smaller the cavitation number, the more likely cavitation will occur in the flow system. Cavitation must be distinguished from phase change phenomena caused by temperature variations. Cavitation is a phase change phenomenon caused by a decrease in pressure.

3.2 Simulation of cavitation in the pipeline system

Upon completion of the flow coefficient calculation for this regulating valve, the valve was subsequently deployed in high-temperature and high-pressure operating conditions. The region of the simulation is the connecting pipeline system of the regulating valve. The model of the fluid domain modeling is shown in Figure 7.

The inlet pressure is 1.96 MPa, the outlet pressure is 0.5 MPa, and the temperature is 212°C. In addition to the simulation settings in Section 2.1, the cavitation model needs to be enabled in Fluid Pair Models options, as shown in Figure 8. The pressure values are all absolute pressures. The cavitation simulation is initially performed within the fluid domain, bubble formation occurs in the liquid flow during cavitation, followed by bubble growth governed by the bubble model. This model characterizes both individual bubble dynamics and the collective behavior of bubble clusters per unit volume, thereby facilitating computation of the volume or mass fractions for liquid and gas phases in the fluid domain.

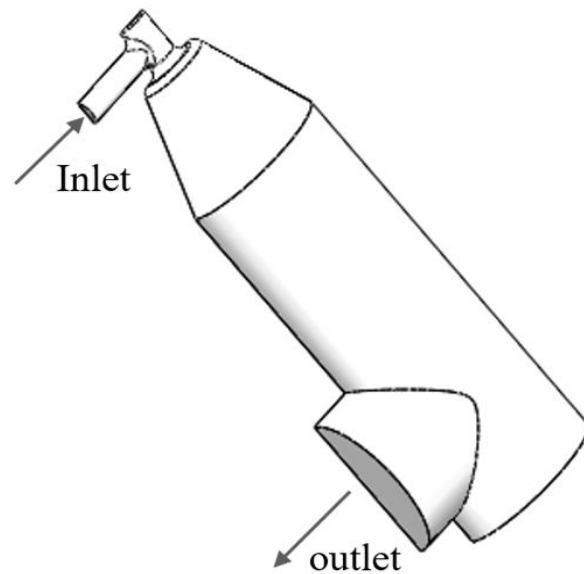


Fig. 7 The model of the fluid in the pipeline connecting the regulating valve

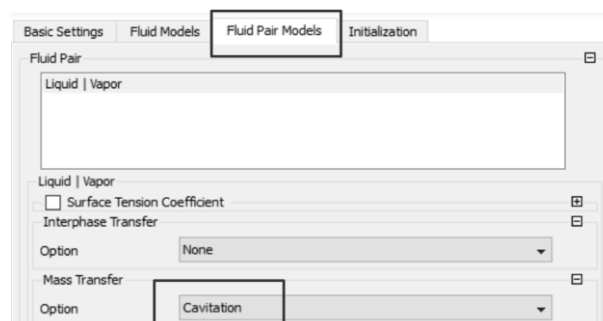
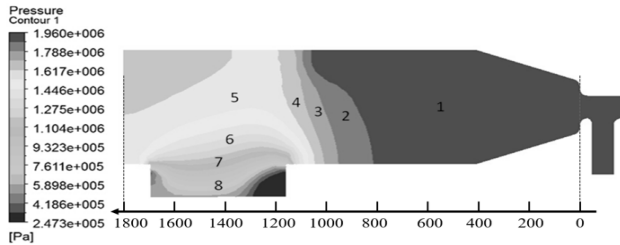


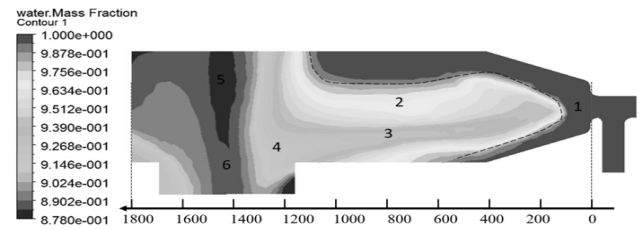
Fig. 8 Cavitation model selection in Fluid Pair Models options

As shown in Figure 9, the pressure contour and the mass fraction contour of the domain are presented. It can be observed that the pressure gradually decreases from the inlet to the outlet. It also can be observed that after the liquid water enters the inlet, as the pressure decreases to the saturated vapor pressure

at this temperature, cavitation begins to occur, and the mass fraction of liquid water gradually decreases to 87.8%. The internal fluid domain has transformed from a one-phase flow of liquid water to a gas-liquid two-phase flow, containing both liquid water and a large amount of bubbles.



(a) Pressure contour



(b) Mass fraction contour

Fig. 9 Pressure and the mass fraction contour of the domain

Figure 10 depicts the velocity contour. When the water is in the liquid state, its velocity is relatively low, with the velocity range being below 48.77 m/s. It can be observed from the mass fraction contour that water changes from liquid to gas, causing a sharp increase in velocity. The high-speed two-phase flow impacts the pipeline system, resulting in erosion.

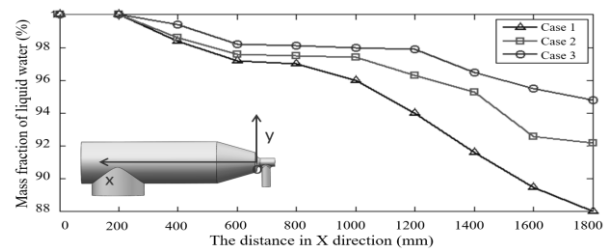


Fig. 11 The variation of mass fraction of liquid water under three cases

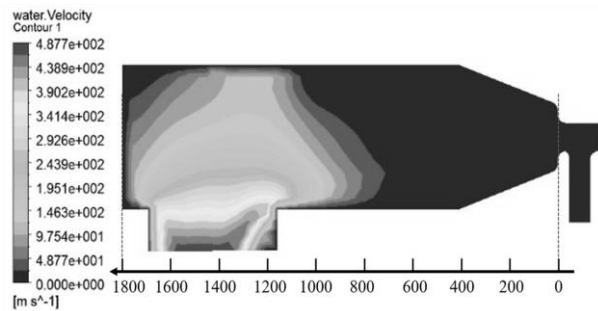


Fig. 10 Velocity contour

In addition, domains with cavitation are simulated under inlet pressures of 2.46 MPa and 2.96 MPa, with other simulation parameters unchanged. Figure 11 shows the variation of mass fraction of liquid water in the cavitation fluid domain at inlet pressures of 1.96 MPa, 2.46 MPa, and 2.96 MPa, corresponding to cases 1, 2, and 3, respectively. The graphical data indicates that cavitation occurs approximately 200 mm from the inlet, accompanied by a reduction in liquid water fraction from the initial 100% to 94.8%, 92.1%, and 88.1%, respectively. At this 200 mm location, the fluid domain pressure falls below the saturation vapor pressure at the given temperature, and the fluid transforms from pure liquid water to a gas-liquid two-phase flow. As cavitation occurs and develops, it can be concluded that the higher the inlet pressure, the greater the mass fraction of liquid water, which indicates that the higher the inlet pressure, the greater the resistance to cavitation development.

Figure 12 shows the variation of average outlet velocity under three cases. It can be observed that the lower the inlet pressure, the higher the average outlet velocity. Combined with the analysis of Figure 11, it can be concluded that the lower the pressure, the larger the proportion of gas in the gas-liquid two-phase flow, leading to a higher velocity of the flow. The outlet velocity begins to increase gradually at 1000 mm from the inlet, reaching up to 400 m/s in Case 1. Although Case 1 corresponds to the lowest inlet pressure (1.96 MPa) among the three cases and exhibits the smallest mass flow rate at the outlet, its maximum outlet velocity can be attributed to the highest gas fraction. The underlying mechanism is that in Case 1, the reduction magnitude of outlet flow rate is smaller than that of the average fluid density decrease, resulting in the highest outlet velocity compared with other cases.

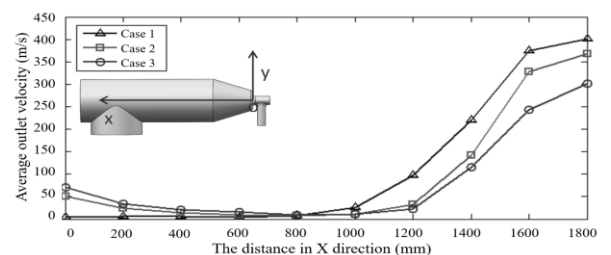


Fig. 12 The variation of average outlet velocity under three cases

4 Erosion prediction of the pipeline

Cavitation numerical simulation serves as both the foundation and data source for the erosion prediction model. Key parameters including flow rate and flow velocity are derived from cavitation simulation results. After analyzing the cavitation phenomena in the regulating valve pipeline system and identifying the causes of erosion, it becomes essential to conduct a quantitative assessment of the erosion of the pipeline.

Currently, there is considerable research on wear caused by solid particles, but research on erosion related to gas-liquid multiphase flow is still limited. By combining the simulation results obtained in Sections 2 and 3, the erosion prediction method in [13] is used to predict the mass loss amount caused by the erosion:

$$R = \frac{\dot{m}_l C(d) f(\alpha) v^{b(v)}}{A_{face}} \quad (\text{kg}/(\text{m}^2 \cdot \text{s})) \quad (9)$$

Where:

\dot{m}_l ...The average mass flow rate at the outlet, calculated using CEL functions in CFX post-processing.

$C(d)$...Function of the droplet diameter, with its value set to 1.8×10^{-9} as referenced in literature [14].

α ...The impact angle of the droplet with the surface,

$f(\alpha)$...Function of the impact angle. According to the erosion model in [15], when the impact angle is a right angle,

$f(\alpha)$...Is taken as 0.4.

$b(v)$...Function of the impact velocity. According to the literature [16], ignoring chemical damage,

$b(v)$...Is taken as 0.8.

The time is set to 110 hours, and the average outlet velocity is obtained from the aforementioned simulation in Section 3.2. By substituting all the values into equation (9), the mass loss amount caused by the erosion is able to be obtained:

$$R_t = \dot{m}_l C(d) f(\alpha) v^{b(v)} t = 10.88 \text{ kg} \quad (10)$$

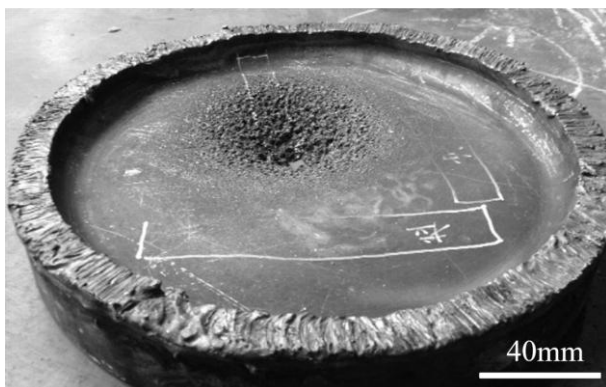


Fig. 13 The erosion condition of the bottom plate

The erosion condition of the bottom plate of the piping system, with a thickness of 50 mm, is shown in Figure 13. The erosion area approximately forms a conical shape, with the conical surface covered densely with numerous pits. The mass loss, as measured, is 6.378 kg. Considering the complexity of the cavitation phenomenon, the proposed method has a good agreement between the prediction of mass loss and the measured mass loss and can predict the erosion caused by cavitation in engineering.

5 Conclusions

The regulating valve serves as a critical component in hydraulic control systems, where accurately determining the flow coefficient is essential for ensuring normal operation. Current methods for calculating flow coefficients rely on theoretical estimations or empirical formulas, which exhibit reasonable accuracy for simple fluid domains but demonstrate limitations when applied to complex configurations. Many regulating valves operate under harsh conditions, particularly in high-temperature and high-pressure pipeline systems, where cavitation frequently occurs. In addition, experimental studies in engineering can only obtain the erosion wear results, identifying the root causes remains challenging. Furthermore, experimental approach to investigate erosion requires specialized test with high costs. Several conclusions are drawn as follows:

- (1) A CFD simulation-based method is proposed to calculate flow coefficients across all valve opening degrees. Compared to measured values, the maximum error is 4.3%.
- (2) Cavitation simulations are performed in the fluid domain of the pipeline system connected to the regulating valve, revealing the underlying causes of severe erosion on the bottom plate. Under inlet pressures of 1.96 MPa, 2.46 MPa, and 2.96 MPa, the liquid water volume fraction decreased to 94.8%, 92.1%, and 88.1%, respectively. At an inlet pressure of 1.96 MPa, the maximum exit velocity of the liquid flow reached 400 m/s.
- (3) A novel method for predicting erosion amount in the bottom plate is presented. The proposed model estimates an erosion amount of 10.88 kg for the regulating valve after 110 hours of continuous operation. Correctness of the method is validated through comparison with actual erosion measurements.

- (4) The simulation-based coefficients calculation and the erosion prediction method presented in this study can assist in improving the structural design of the regulating valve and its pipeline system, and shorten the design cycle of the regulating valve.

Erosion in regulating valves represents a complex engineering issue. Future research should prioritize:

- (1) In-depth mechanistic studies of erosion processes
- (2) Comprehensive evaluation of turbulence model effects on numerical simulation accuracy
- (3) Development of dedicated experimental platforms for analyzing both flow characteristics and erosion patterns.
- (4) For the erosion prediction modeling, the parameters require further optimization to achieve more precise parameter quantification, thereby obtaining accurate erosion amount.

Acknowledgement

This research was funded by the Key Natural Science Research Project of Chengdu Aeronautic Polytechnic University in 2024 (ZZX0624087).

References

- [1] BAO, Y., WANG, H. (2022). Numerical study on flow and heat transfer characteristics of a novel Tesla valve with improved evaluation method. In: *Int J Heat Mass Transf*, Vol.187, pp. 122540. ISSN 1879-2189
- [2] ROUTRAY, A., SINGH, R., CEPOVA, L., SANDEEP, V., et al. (2026). Design and Life of a Ball Valve as per the ASME BPVC Section VIII by the Elastic Stress Analysis Method. In: *Manufacturing Technology*, Vol.26(1), pp. 78–87. ISSN 2787-9402
- [3] KUMAR, S., TEWARI, V. K., BHARTI, C. K., RANJAN, A. (2021). Modeling, simulation and experimental validation of flow rate of electro-hydraulic hitch control valve of agricultural tractor. In: *Flow Meas Instrum*, Vol.82, pp. 102070. ISSN 0955-5986
- [4] LIN, Z., LI, J., JIN, Z., QIAN, J. (2021). Fluid dynamic analysis of liquefied natural gas flow through a cryogenic ball valve in liquefied natural gas receiving stations. In: *Energy*, Vol.226, pp. 120376. ISSN 1873-6785
- [5] YANG, B. (2022). Modeling Compressor Pressure Ratio vs. Mass Flow Rate Including Choke Line and Surge Line Using Scaling Theorem. In: *SAE Technical Paper*. ISSN 0148-7191
- [6] PAWEL, S. J., MANSUR, L. K. (2008). Cavitation-erosion resistance of 316LN stainless steel in mercury containing metallic solutes. In: *J Nucl Mater*, Vol.377, pp. 174–181. ISSN 0022-3115
- [7] HATTORI, S., TAKINAMI, M. (2010). Comparison of cavitation erosion rate with liquid impingement erosion rate. In: *Wear*, Vol.269, pp. 310–316. ISSN 0043-1648
- [8] SUN, Z., WANG, X. (2023). Function Modeling and Simulation Analysis of Aero-engine VB. In: *Manufacturing Technology*, Vol.23(4), pp. 538–544. ISSN 2787-9402
- [9] FOLDEN, T. S., ASCHMONEIT, F. J. (2023). A classification and review of cavitation models with an emphasis on physical aspects of cavitation. In: *Phys Fluids*, Vol.35, pp. 081301. ISSN 1070-6631
- [10] ZHANG, J., GAO, R., WU, Y. (2025). Research on Optimization Design and Processing Technology of Engine Intake System Based on NX and Fluent. In: *Manufacturing Technology*, Vol.25(5), pp. 711–719. ISSN 2787-9402
- [11] REINKE, P., RIENAECKER, A., SCHMIDT, M., BECKMANN, T. (2023). Digital High-Speed Photography of Cavitation in Journal Bearings. In: *SAE Technical Paper*. ISSN 0148-7191
- [12] WANG, B., SU, H., ZHANG, B. (2021). Hydrodynamic cavitation as a promising route for wastewater treatment - A review. In: *Chem Eng J*, Vol.412, pp. 128685. ISSN 1385-8947
- [13] ZHANG, S., DAM, J. K., NORKJER, S. (2014). Erosion of wind turbine blade coatings - Design and analysis of jet-based laboratory equipment for performance evaluation. In: *Prog Org Coat*, Vol.78, pp. 103–115. ISSN 0300-9440
- [14] LI, R., MORI, M., NINOKATA, H. (2012). A calculation methodology proposed for liquid droplet impingement erosion. In: *Nucl Eng Des*, Vol.242, pp. 157–163. ISSN 0029-5493
- [15] EDWARDS, J. K., MCLAURY, B. S., SHIRAZI, S. A. (2000). Evaluation of alternative pipe bend fittings in erosive service. In: *Proceedings of the 2000 ASME Fluids Engineering Division*, pp. 959-966. ISSN 1093-4928
- [16] HAN, L., LI, X., XUE, F. (2019). Biocorrosion behavior of micro-arc-oxidized AZ31 magnesium alloy in different simulated dynamic physiological environments. In: *Surf Coat Technol*, Vol.361, pp. 240–248. ISSN 0257-8972