Experiment and numerical simulation of cavitation performance on a pressure-regulating valve with different openings

W S Qu1, L Tan1, S L Cao1,3, Y Xu1, J Huang2 and Q H Xu2
1 State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China
2 Zhuzhou Southern Valve CO.,LTD., Zhuzhou 412007, China
E-mail: caoshl@mail.tsinghua.edu.cn

Abstract. As a kind of widely used device in pipe system for pressure and flow rate regulating, the valve would experience cavitation in the case when a sharp pressure drop occurs, which will induce the energy loss, noise and vibration of pipeline system, and even operational accidents. The experiment on flow resistance coefficient of a DN600 pressure-regulating valve under operation conditions from 0% to 100% openings is conducted. Based on the RNG k-ε turbulence model and the Rayleigh-Plesset cavitation equation, a set of computational model is developed to simulate the turbulent flow in the valve under operational conditions from 0% to 100% openings. The computational results of flow resistance coefficient are compared to the experimental data. And the numerical simulation is employed to predict the cavitation performance of the valve at different inlet flow conditions. The transient cavitating flow is calculated to reveal the time evolution of cavitation in the valve.

1. Introduction
Valves are commonly used in a variety of industries and play an important role in the control of flow rate, especially in hydraulic engineering. In most cases, valve acts as a throttle with the velocity increasing and the pressure dropping, therefore it is also applied to regulate the pressure in the pipeline system. Like in some other hydraulic components, cavitation appears in valves when the local pressure drops below the saturation pressure. Phase transformation occurs and bubbles formed by vapour emerge, grow and finally collapse with the pressure rising again, thus leading to vibration, noise and erosion on the valve body as well as other piping equipment. Since a pressure-regulating valve has to work under various pressure conditions such as atmosphere or even lower as much as required, it seems difficult to avoid cavitation thoroughly. However it is still very necessary to figure out the mechanism of its development and take into consideration its influence in the design of valve system.

Many works have been carried out in the investigation of flow pattern and cavitation in a valve, including both experimental measurements and numerical calculations. Vaughan et al. [1] carried out a series of simulations, and studied a range of poppet valves with different geometries and under different flow rate to predict the flow patterns and force characteristics. Recently, Valdes et al. [2] simulated the cavitating flow through a ball check valve and performed experimental tests to validate the simulation results. Tabrizi [3] et al. modelled the flow pattern in a ball valve at different opening
angles by modifying the turbulent viscosity and validated the results with experiments and then discussed the pressure drop behind the valve and formation of the vortex flow downstream. Chern et al. [4] studied the cavitation flow inside a globe valve numerically and proposed a way to prevent cavitation in the valve body and at the downstream region by introducing a one-stage perforated cage. Gao et al. [5] presented a study to investigate the cavitating flow near the orifice of a poppet valve, and conducted a visualization experiment to obtain cavitating images. The results showed that to reduce the outlet area can effectively suppress cavitation. Jazi et al. [6] studied the acoustic waveform to detect the cavitation occurrence in a globe valve, and analysed the amplitude and frequency.

Many of the previous works focused on the traditional valve types such as ball valve, however, the piston valve with a perforated structure to regulate the pressure is rarely studies. Besides, the transient numerical simulation of cavitation flow in a valve has hardly been carried out.

The main aim of this study is to predict the cavitation levels under different operating conditions in a pressure-regulating valve by both steady and unsteady numerical simulation. A set of computational model is developed and validated by a series of experiments. The flow patterns, pressure distribution and vapour volume fraction inside the valve are analysed in order to explore the mechanism of the cavitation occurrence.

2. Description of the valve model and CFD method

2.1. Valve model and meshes

Figure 1 shows the schematic of valve in the present study. The fluid is supposed to flow into the valve body through a pipe with a diameter of 0.2 m. Then it is forced to jet toward the centre by a perforated cylinder structure (Figure 2) which will be named as the cage in the following passages. The internal structure as well as the operating principle of the valve is also revealed in Fig 1. The connecting rod system driven by a stepper motor pushes the cage forward and backward along a guide rail to realize different operation conditions.

The computational domain is composed of two parts. One part is the stable part under different operation conditions from 0% to 100% openings, and it is meshed in hexahedral mesh. The other part is the moving part under different operation conditions, and it contains 240 circumferential displaced holes on the cage, thus adopting the tetrahedral meshes due to the complicated structure. The mesh of both parts is shown in Fig 3. To figure out the influence of mesh number, a grid independency test is previously carried out. A variety of computational cells are utilized to simulate the flow in the valve with the opening of 40%. The result in table 1 shows that the three set of computational mesh makes little difference (error below 0.8%) in the pressure drop, especially when the cells are more than 2 million (error below 0.2%). Finally the mesh with 1,980,036 elements is chosen in the following calculation.
Figure 3. Computational domain and meshes (left) and meshes at the connecting holes (right)

Table 1. Result of mesh independency test.

| Mesh Number | Mass Flow Rate (kg/s) | P\text{in} (MPa) | P\text{out} (MPa) | ΔP (MPa) |
|-------------|-----------------------|-----------------|-----------------|---------|
| 953,046     | 57.444                | 0.52            | 0.2534          | 0.2666  |
| 1,980,036   | 57.444                | 0.52            | 0.2520          | 0.2680  |
| 2,948,484   | 57.444                | 0.52            | 0.2514          | 0.2680  |

2.2. Numerical methods

The basic governing equation of turbulent flow in the valve is the 3-dimentional Reynolds-averaged Navier-Stokes equation, which can be denoted as

\[ \frac{\partial}{\partial t} (\rho_m) + \nabla \cdot (\rho_m \mathbf{u}) = 0 \]  

(1)

\[ \frac{\partial}{\partial t} (\rho_m \mathbf{u}) + \nabla \cdot (\rho_m \mathbf{uu}) = -\nabla p + \nabla \cdot [(\mu_m + \mu_t)\nabla \mathbf{u}] + \frac{1}{3} \nabla \left[ (\mu_m + \mu_t) \nabla \cdot \mathbf{u} \right] \]  

(2)

where \( \rho_m \) is the mixture density; \( \mu_t \) is the turbulent viscosity; \( \mu_m \) is the kinetic viscosity averaged in accordance with the volume of the vapour and liquid phases; \( p \) is the pressure and \( \mathbf{u} \) is the velocity vector. The RNG \( k-\varepsilon \) turbulence model is applied because of its advantage in disposing the flow with a high strain rate and streamline curvature.

The cavitation model applied in the present study is the transport equation based on the Rayleigh-Plesset equation. Taking into account the evaporation and condensation process in the cavitation, the transport equation is given as

\[ \frac{\partial}{\partial t} (\alpha, \rho_v) + \nabla \cdot (\alpha, \rho_v \mathbf{u}) = m^e - m^c \]  

(3)

\[ m^e = C_{\text{vap}} \frac{3 \alpha, \rho_v}{R_b} \sqrt{\frac{2}{3}} \frac{\max(p^*-p,0)}{\rho_v} \]  

(4)

\[ m^c = C_{\text{cond}} \frac{3 \alpha, \rho_v}{R_b} \sqrt{\frac{2}{3}} \frac{\max(p-p^*,0)}{\rho_v} \]  

(5)

where \( m^e \) and \( m^c \) represent the rates of evaporation and condensation, respectively. Other coefficients appeared in the equation are also specified: \( C_{\text{vap}} = 50; C_{\text{cond}} = 0.01; \) \( \alpha_c \) is the vapour volume fraction; \( R_b \) is the bubble radius and set as \( 10^{-6} \) m; \( p_v \) is the saturation pressure of water at test temperature of 3574 Pa.

In consideration of the influence of turbulent pressure fluctuation to the saturation pressure, the corrected expression of saturation pressure is

\[ p_v' = p_v + 0.195 \rho_m k \]  

(6)
The simulation of the turbulence flow and cavitation in the valve is carried out by solving the equations above with the CFD software ANSYS CFX. The pressure at inlet and mass flow at outlet are specified in accordance with the experimental measurement. The no-slip boundary conditions are imposed at the solid walls of the valve. Besides, in the unsteady calculation the time step is set as $10^{-4}$ s and the total time step number is 2000.

3. Results and analysis
Both steady and unsteady flows of cavitation in the valve are simulated in this study. The pressure drop results are obtained and compared with the experimental data, and then the flow patterns under five different opening conditions are analysed. On the basis of steady results, the development of cavitation is predicted in the unsteady simulation.

3.1. Experiment validation
A hydraulic test system in Fig.4 has been developed for the experiment, with a flowmeter and two pressure sensor installed to measure the flow rate and pressure at both sides of the valve. The pressure upstream is fixed by using a surge tank in the experiment, and the flow rate is controlled by an extra valve downstream. To validate the model veracity and the calculation accuracy, experiments are conducted under five different opening conditions: 20%, 40%, 60%, 80% and 100% of valve opening. Comparison has been made between the computational and experimental results. The concept of resistance coefficient is defined as

$$K = \frac{2\Delta p}{\rho v^2} \quad (7)$$

where $\Delta p$ is the pressure difference between inlet and outlet ports; $\rho$ is the fluid density; $v$ is the cross-sectional velocity of the fluid and $K$ is the flow resistance coefficient.

As presented in Fig 5, the experimental and numerical results agree well, except for the 20% opening, where the error may be caused by the complex flow pattern at small flow rate. The agreement in the tendencies of the two curves is good and it can be validated that the computation model has enough accuracy to be utilised in the following analysis.

3.2. Flow analysis under different opening conditions
Figure 6 depicts the pressure contours and flow patterns in the valve with different openings. Counter-rotating vortex pairs can be seen in every figure because the flow in this region is stopped and reversed by the wall. At small openings below 60%, a sharp pressure drop is caused when the fluid flows through the holes on the cage. So the pressure in the moving part is much lower than that in the stable part. With the increase of the opening, the pressure loss in the holes decreases, and the pressure in the moving part rebounds. For the opening above 80%, the flow resistance is further reduced and the pressure loss becomes unconspicuous because the fluid passes the wide channel rather than the holes on the cage.
Basically speaking, smaller valve opening leads to greater pressure decrease. Therefore, when the inlet pressure is low enough, the cavitation will occur. Figure 7 demonstrates the cavitation 3D structure by using the iso-surfaces with the vapour fraction of 0.1. Distributions of vapour differ when different inlet pressure conditions are given. Vapour cavities hardly appears with an inlet pressure higher than 0.4 MPa as indicated in Fig.7 (a). They emerge at the inlet of the holes with the inlet pressure around 0.39 MPa as shown in Fig.7 (b). When the set value approaches 0.37 MPa as shown in Fig.7 (c), vapour almost fills the hole walls, which can be defined as the cavitation inception because erosion on the valve body and influence on the downstream flow come to be significant. With a further drop in inlet pressure as shown in Fig.7 (d), cavitation becomes more serious, vapour expands and joins together in the flow passage. Cavitation shape grows more disordered and severe damage can be caused under this circumstance.

**Figure 6.** Flow pattern under different openings: (a) 20%; (b) 40%; (c) 60%; (d) 80% and (e) 100%.

**Figure 7.** Cavitation under different pressure conditions (40% opening): (a) $P_{in}=0.52$MPa; (b) $P_{in}=0.39$MPa; (c) $P_{in}=0.37$MPa and (d) $P_{in}=0.31$MPa.

### 3.3. Unsteady calculation of cavitation

The transient cavitation flow is also simulated based on the previously developed cavitation model in this study. The operating condition with an opening of 40% and inlet pressure of 0.31 MPa has been taken as an instance. The evolution of the cavitation shape inside the valve is shown in Fig.8, with a time interval of 0.05 s (i.e. 500 time steps) between two sequent captures. At first the cavities are relatively scattered, with several parts distributing in different regions along the flow passage. The largest cavity is situated at the outlet of the holes and forms a ring in the circumferential direction. This annular cavity gradually extends and unites downstream. The shape turns more ordered and the volume increases after that.
4. Conclusion
A series of numerical simulations of the turbulent flow through a pressure-regulating valve has been performed. The cavitation is studied by developing and applying a set of cavitation model based on the transport equation. A batch of experiments on the flow resistance coefficient has been conducted, measuring the pressure drop through the valve with the flow rates given. Two results have been compared and show good agreement in the tendency. The flow patterns are visualised by pressure contour and streamlines. The main flow is shown to jet towards the centre from the holes and leads to a sharp pressure drop at small opening conditions. Counter-rotating vortex pairs are observed because part of the flow is stopped and reversed by the wall. Cavitation vapour emerges at the inlet of the holes at first, then fills the region adjoining to the hole-walls when the inlet pressure decreases. With a further pressure drop, the cavities expand and join together. The transient cavitation flow is calculated then, and shows the time evolution of cavitation shape. The cavities are scattered and distributed in several parts. Afterwards the large annular cavity near the holes extends and unites downstream, which tends to fall off the solid wall and move downward. Based on the analysis above, the structure of the valve can be further improved to reduce cavitation and its accompanying problems.

References
[1] Vaughan N D, Johnston D N and Edge K A 1992 Proc. of the Institution of Mech. Engineers, Part C: J. of Mech. Eng. Sci. 206(2) 119-27
[2] Valdés J R, Rodriguez J M, Monge R, Peña J C and Pütz T 2014 Energy Conversion and Management 78 776-86
[3] Tabrizi A S 2014 J. Of Eng. Thermophysics 23(1) 27-38
[4] Chern M J, Hsu P H, Cheng Y J, Tseng P Y and Hu C M 2013 J. of Energy Eng. 139(1) 25-34
[5] Gao H, Lin W and Tsukiji 2006 Pro. of Institution of Mech. Engineers Part G-J. of Aerospace Eng. 220(G4) 253-65
[6] Jazi A M and Rahimzadeh H 2009 Ultrasonics 49(6-7) 577-82