Cavitation characteristics and suppression of pilot stage two-dimensional valve

Yonghua Zhao
    Jiaxing Vocational & Technical College

Tong Xing (✉️ 787077286@qq.com)
    College of Mechanical Engineering, Zhejiang University of Technology

Chuan Ding
    College of Mechanical Engineering, Zhejiang University of Technology

Heyuan Wang
    College of Mechanical Engineering, Zhejiang University of Technology

Jian Ruan
    College of Mechanical Engineering, Zhejiang University of Technology

Research Article

Keywords: 2D valve, computational fluid dynamics (CFD), cavitation, suppression

Posted Date: July 22nd, 2022

DOI: https://doi.org/10.21203/rs.3.rs-1879291/v1

License: This work is licensed under a Creative Commons Attribution 4.0 International License.
Read Full License
Abstract

The two-dimensional (2D) valve integrates the spool pilot and power stages to realize the valve's fast operation and high-frequency response. It has the advantages of simple structure, stable performance, and large power-to-weight ratio. In this paper, the cavitation flow in the pilot stage of the 2D valve is analyzed utilizing computational fluid dynamics. The effects of inlet and outlet pressure difference, outlet pressure, and valve opening velocity on cavitation are studied, and a U-shaped groove structure is designed to restrain cavitation. The results demonstrate that the greater the pressure difference, the higher is the cavitation intensity. Furthermore, the cavitation intensity increases with decrease in outlet pressure. The cavitation phenomenon occurs in the 2D valve pilot stage when the valve opening speed is high, and there is no cavitation in the flow channel with the decrease of the valve opening speed. After U-shaped grooves are opened on both sides of the valve sleeve groove, the cavitation number in the 2D valve pilot stage increases significantly compared with the original inclined groove structure under the same pressure difference. Additionally, the effect of restraining cavitation is pronounced with the increase of the width–depth ratio of U-shaped grooves, and the cavitation number increases.

Introduction

Hydraulic valves are control elements used to regulate the pressure, flow, and direction of liquid in hydraulic systems. The two-dimensional (2D) valve integrates the pilot and power stages on two degrees of freedom of a spool, and the opening of the rotary slide valve in the pilot stage has a high-pressure gain. The electro-mechanical converter only needs to output a small angular displacement to cause a sharp change in pressure, which is easy to realize the fast operation and high-frequency response of the valve. 2D valves have the advantages of small size, simple structure, stable performance, ideal dynamic characteristics, strong anti-pollution ability, low leakage flow, and large power-weight ratio. The rotation of the 2D valve spool causes the throttle of the valve to open and close frequently, and the liquid pressure drops sharply. When the local pressure of the liquid is lower than its saturated vapor pressure, the original “gas core” in the liquid grows into a bubble, and the bubble collapses at high pressure. Cavitation subsequently occurs, which is a key cause of pressure fluctuations, vibration, and noise in 2D valves.

Scholars have researched the occurrence, strength, and suppression of cavitations in the valve using numerical simulations or laboratory analyses based on the valve structural parameters, valve port opening, and the boundary conditions imposed before and after the valve port. In the throttle valve, cavitation occurs first near the throttle orifice, and then the annular bubble group causes structural damage downstream of the throttle valve. The cavitation appears at the front of the baffle and tip of the inner and outer nozzle wall in the electro-hydraulic servo valve. Saito et al. investigated the high-pressure cover valve and concluded that the cavitation bubble first emerges near the valve port at a small opening and collapses after a short distance. Lee et al. performed the numerical simulation of the reversing globe valve and established that the cavitation strength in the valve decreased with increase of the spool tail and wrist lengths. For the rotary valve, the cavitation strength increases with decrease in the
inlet diameter. To restrain cavitation in the valve, the groove depth needs to be properly selected\textsuperscript{7}. The use of an inverted circle, appropriate sealing cone angle, and inverted cone hole results in enhanced anti-cavitation performance for the control valve\textsuperscript{8}. An increase in the inlet pressure leads to greater cavitation strength in the valve\textsuperscript{9–12}. Cavitation mainly occurs near the cutoff valve seat for small openings and downstream of the valve disk for large openings and high flow rates\textsuperscript{13}. The cavitation area in front of the pressure relief valve disk and the cavitation strength in the pilot globe valve decreases with increase in the valve opening\textsuperscript{14,15}. Du Xuewen et al.\textsuperscript{16} experimentally studied the influence of the structural characteristics of the throttle groove on the pressure distribution, cavitation, and noise characteristics in the valve. They concluded that U-shaped grooves could better restrain the precipitation and growth of cavitation than the V-shaped grooves. Zhang Xiaokang et al.\textsuperscript{17} also used ANSYS FLUENT software to simulate and verify the effect of the current-limiting orifice plate on suppressing cavitation of compound regulating valve.

This paper takes the 2D valve pilot stage structure as the research object based on the above research methods and conclusions. We use FLUENT software for numerical simulation, analyze the change of cavitation intensity in the valve under different boundary conditions, and study the methods to restrain cavitation and provide a basis for the optimal design of the 2D valve.

1. Working Principle Of 2d Valve

The working principle of the 2D valve is shown in Figure 1 (a). The electro-mechanical converter shown in Fig. 1 (b) converts the electrical signal of the controller into a mechanical signal and transmits it to the mechanical transmission mechanism. The rotation signal amplifies the driving moment proportionally through the transmission mechanism to drive the valve core to rotate. The rotation of the valve core changes the coincidence area between the valve sleeve groove and the high- and low-pressure port, and the sensitive cavity pressure at the left end of the valve core changes accordingly. The pressure variation in the sensitive chamber causes an imbalance at both ends of the spool, driving the axial movement to control the valve pressure and flow output.

2. Preprocessing Of Simulation

2.1. Establishment of runner model

The three-dimensional (3D) model of the three-size 2D valve is established using the UG 3D modeling software, as illustrated in Figure 2 (a). The flow channel model is generated by reverse modeling, and the 2D valve pilot stage valve port-channel structure has the characteristics of double channel center symmetry, as shown in Figure 2 (b). As the spool rotates to open the high-pressure throttle, the fluid flows through the channel of the primary spool middle hole to the model's entrance as shown in Figure 3 (b). The fluid continues to flow through the transit channel, high-pressure area, high-pressure throttle hole and into the valve sleeve chute, and then flows to the sensitive cavity. In this study, half of the fluid models are
selected as the analysis object, as shown in Figure 3 (b). The diameter of the inlet and transition channels is 2 and 1.2 mm, respectively, and the outlet area is about 4.5 mm$^2$.

A structure is designed to restrain cavitation by opening U grooves on both sides of the inclined groove of the valve sleeve. The structure and slot position of the slotted valve sleeve is displayed in Figure 3 (a), associated with the structural characteristics and functional requirements of the 2D valve. Furthermore, Figure 3 (c) presents the fluid model of the slotted valve sleeve.

**2.2. Grid generation and grid independence test**

The MESH software is used to divide the mesh. The slip surface is refined locally using tetrahedral elements, and the throttle$^{18}$ leads to improved accuracy, as displayed in Figure 4.

The number of grids obtained is 223568. In a calculation cycle, the average pressure on the symmetrical surface of the spool varies within 1%$^{17,26}$, which meets the grid independence requirements, as listed in Table 1.

<table>
<thead>
<tr>
<th>Grid type</th>
<th>Grid number</th>
<th>Pressure value (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>151682</td>
<td>4360387</td>
</tr>
<tr>
<td>Grid 2</td>
<td>223568</td>
<td>4519856</td>
</tr>
<tr>
<td>Grid 3</td>
<td>247714</td>
<td>4520104</td>
</tr>
</tbody>
</table>

**3. Calculation Model And Boundary Conditions**

The Mixture model in FLUENT software is selected for numerical calculation. Under the assumption of this model, the time-average velocity of cavitation and oil liquid phases in the flow field is equal to that of the oil liquid phase in a small space length scale; the slip velocity and volume force between gas and liquid phases can be ignored. The fluid and cavitation phases are studied as a unified fluid during cavitation flow. The governing equations are as follows$^{19–22}$:

(1) Continuity equation

$$\frac{\partial \rho_m}{\partial t} + \nabla \left( \rho_m \mathbf{v}_m \right) = 0$$
where \( \nu_m \) is the average velocity of mass, \( \nu_m = \sum_{k=1}^{n} \alpha_k \rho_k v_k \left/ \rho_m \right. \), \( \rho_m \) is the mixing density, \( \rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \) is the volume fraction of phase \( k \).

(2) Momentum equation of gas phase and liquid phase

The momentum equation of the mixed model is obtained as follows

\[
\frac{\partial (\rho_m \nu_m)}{\partial t} + \nabla (\rho_m \nu_m \nu_m) = -\nabla p + \nabla \left( \rho_m \left( \nabla \nu_m + \nabla \nu_m \right) \right) + \rho_m g + F + \nabla \left( \sum_{k=1}^{n} \alpha_k \rho_k v_{drk} v_{drk} \right)
\]

(2)

where \( n \) is the phase number, \( \nu_m \) is the volumetric force, \( \mu_m \) is the mixed viscosity, \( \mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \) is the drift velocity of the second phase \( k \).

(3) Volume fraction equation

It is assumed that the hydraulic oil is an incompressible fluid; below is the volume fraction equation

\[
\frac{\partial \alpha_k}{\partial t} + \nabla \left( \alpha_k \nu_m \right) = \frac{\rho_l}{\rho_m} \eta \frac{d\phi}{dt} + \frac{\rho_v}{\rho_m} \frac{d\rho_v}{dt}
\]

(3)

where \( \eta \) is the number of bubbles per unit volume of fluid, \( \phi \) is the other volume fraction, \( \rho_l \) is the liquid phase density, \( \rho_v \) is the gas phase density, \( \rho_m \) is the density of mixed, \( \rho_m = \alpha_k \rho_v + \left( 1 - \alpha_k \right) \rho_l \).

(4) Cavitation model

Schnerr-Sauer cavitation model is a kind of cavitation model based on the Rayleigh-Plesset equation. The high-order and surface tension terms are ignored in the derivation process, but compared with Singhal and ZGB cavitation model, this model does not introduce any empirical coefficient. Thus, the Schnerr-Sauer model is an ideal cavitation model\(^{23}\) which is displayed below.

\[
R_e = \frac{\rho_l \rho_v}{\rho_m} \alpha_v \left( 1 - \alpha_v \right) \frac{3}{R_B} \left( \frac{2}{3} \frac{P_v - P}{\rho_l} \right)^{1/2}, \text{ } P \leq
\]

(4)
\[ R_B = \left( \frac{\alpha_v}{1 - \alpha_v} \frac{3}{4\pi} \frac{1}{n_0} \right)^{1/3} \]

where \( R_B \) is the cavitation radius, \( P_v \) is the saturated vapor pressure of flu, \( d \), saturated steam pressure of hydraulic oil is 37100 Pa at 20°C, \( n_0 \) is the numerical density of cavitation per unit liquid vol, me \( n_0 = 10^{13} \).

(5) Turbulence model

The \( k - \varepsilon \) standard model is selected as the turbulence model to ensure stable and fast convergence. According to the conservation of mass and momentum, this is the transport equation of incompressible turbulent kinetic energy and turbulent energy dissipation rate \( \varepsilon \).

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\delta_k}) \frac{\partial k}{\partial x_j} \right] + P - \rho \varepsilon
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\delta_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( {C_{\varepsilon 1}} P + \rho {C_{\varepsilon 2}} \varepsilon \right)
\]

\[
P = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}
\]

\[
\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}
\]

where \( \rho \) is the fluid density, \( u_i \) is the velocity component along the direct, \( \mu \) is the coefficient of kinematic viscosity of the medium, \( \mu_t \) is the turbulent viscosity, \( \rho \varepsilon \) is the dissipation term of turbulent kinetic energy, \( \delta_k \) is the Prandtl number corresponding to turbulent kinetic energy, \( \delta_\varepsilon \) is the Prandtl number corresponding to turbulent energy dissipation rate \( \varepsilon \), \( P \) is the turbulent kinetic energy generating term, \( C_{\varepsilon 1} = 1.44 \), \( C_{\varepsilon 2} = 1.92 \), \( C_\mu = 0.09 \), \( \delta_k = 1.0 \), \( \delta_\varepsilon = 1.3 \).

The numerical simulation defines the main phase as hydraulic oil, density 780 kg/m³, viscosity 0.0024 kg/m s, bulk elastic modulus 700 MPA, secondary phase air, density 1.225 kg/m³, viscosity 1.789 × 10⁻⁵ kg/m s, and the transition between main phase and secondary phase satisfies cavitation model. The model is defined as pressure inlet and outlet, and the slip grid model is adopted. The calculation time is 0.0005 s, and the transient calculation of fluid flow in the valve within 0.03 s.
4. Analysis Of Simulation Results

Based on the above numerical simulation settings, the model is simulated in FLUENT2020R2, and the cavitation characteristics of the flow field in the 2D valve pilot stage are analyzed. When the opening of the throttle port is 0.01 mm, the valve spool and the inclined groove sliding surface of the valve sleeve (such as the red surface shown in Figure 3 (b)) are the analysis surface.

4.1. Effects of pressure difference on the pilot stage

The possibility of cavitation grows with the increase of the pressure ratio. To study the effect of pressure difference on the cavitation in the pilot stage of the 2D valve, the outlet pressure is kept at 0.1MPa, and the inlet pressure is varied.

The cavitation number (σ) is widely used to characterize the cavitation intensity or inception. Hence, the cavitation number flowing through the throttle of the hydraulic valve is defined as follows [26]:

\[ \sigma = \frac{(p_1 - p_v)}{1/2 \rho v^2} \]

Here \( p_1 \) is the upstream pressure of the throttle, Pa; \( p_v \) is the saturation pressure, Pa; and \( v \) is the velocity of the fluid at the throttle, m/s.

Figure 5 shows the velocity cloud diagram of the slip surface of the spool and the valve sleeve chute channel where the 2D valve pilot stage throttle is located under three different pressure differences. The cloud pictures under different pressure differences are approximately equal. Consequently, the velocity at the throttle increases with the rise in pressure difference. If the pressure difference is too high, the pressure at the throttle may be as low as the saturation pressure, resulting in cavitation. Figure 6 exhibits the pressure distribution upstream and downstream of the throttle's surface and depicts the pressure distribution curve on the bottom of the chute. The pressure distribution is essentially equivalent for varying pressure difference values. The minimum pressure in the downstream area of the throttle is greater than the saturated vapor pressure when the pressure difference is 3.9 MPa, resulting in no cavitation. When the pressure difference is greater than 6.9 MPa, there is an apparent saturation and low-pressure zone in the downstream area of the throttle. A large low-pressure area appears when the pressure difference reaches 9.9 MPa. The pressure difference at the initial stage of cavitation is 5.9 MPa. Therefore, when the outlet pressure is 0.1 MPa, the inlet pressure should be less than 6 MPa to avoid cavitation.

The pressure distribution at the bottom of the inclined groove to determine the pressure value under various pressure differences is shown in Fig. 7. As the pressure difference increases, the throttle pressure decreases, and the pressure recovers downstream of the throttle. When the pressure difference is greater than 3.9 MPa, all pressures are higher than the saturation pressure. A small low-pressure region appears when the pressure difference is greater than 5.9 MPa, and a low pressure equal to the saturation pressure rises at the bottom of the inclined groove when the pressure difference is greater than 6.9MPa. Moreover,
if the pressure difference is greater than 9.9 MPa, the bottom of the chute is in a state of saturated pressure.

Figure 8 shows the distribution of gas volume fraction on the research surface of the throttle port, which shows that the gas volume fraction changes with increase of pressure difference. There is a large area of gas upstream of the throttle if the pressure difference is 1.9 MPa, and the maximum volume fraction of gas at the throttle is 0.0033. As the pressure difference increases, the gas area at the throttle gradually grows, and the maximum value at 5.9 MPa is 0.021 and reaches 0.03 at 9.9 MPa, indicating that the pressure difference has a significant effect on the cavitation intensity in the pilot stage of the 2D valve.

4.2. Effect of outlet pressure on the pilot stage of 2D valve

The effect of outlet pressure on the cavitation in the pilot stage of the 2D valve is studied by changing the outlet pressure. We chose the same inlet and outlet pressure difference and pressure ratio, and the outlet pressure is 0.1, 1, and 5 MPa.

When the pressure difference is the same, the cavitation number under different outlet pressure is shown in Figure 8. Here the differential pressure is maintained at 5 MPa. It can be seen from Figure 9 that cavitation occurs when the outlet pressure is low, and there is no cavitation in the 2D valve pilot stage when the outlet pressure is high. The high outlet pressure ratio is 2, and the low outlet pressure ratio is 5; so it may be inferred that there is a relationship between cavitation and inlet and outlet pressure ratio.

Figure 10 shows the cavitation number under different outlet pressures when the pressure ratio is 10:00. The results show that the cavitation number with low outlet pressure is low at the same pressure ratio, indicating that the decrease of outlet pressure increases the cavitation intensity in the pilot stage of the 2D valve.

4.3. Effect of valve opening speed on 2D valve pilot stage

By changing the opening velocity of the valve port, the velocity is 0.1, 0.2, 0.3, 0.7, and 1 rad/s, respectively, and its effect on the cavitation in the 2D valve pilot stage is studied. Additionally, we chose the same import and export pressure of 6 and 0.1 MPa, respectively.

When the inlet and outlet pressures are equivalent, the cavitation number of the different valve opening speeds is shown in Figure 11. It can be seen from Figure 11 that cavitation occurs when the opening speed of the valve port is fast, and there may be no cavitation in the 2D valve pilot stage channel with the decrease of the opening speed. Therefore, it can be inferred that the cavitation is related to the opening speed of the valve port.

5. Inhibition Of The Occurrence Of Cavitation

U-shaped grooves are opened on both sides of the valve sleeve groove, restraining the cavitation in the 2D valve pilot stage. The influence of different U-groove width (W) on the cavitation phenomenon is studied
considering the processing technology requirements and the cavitation in the 2D valve pilot stage channel after the U-groove wall opening is analyzed and compared with the original structure. The ratio of width to depth is 1.4, 1.8, and 2.2, respectively.

The minimum pressure in the downstream channel of the throttle is used as a criterion to judge whether cavitation occurs. The pressure of import and export is 6 MPa and 0.1 MPa, respectively.

Figure 12 presents the pressure cloud diagram of the research surface where the 2D valve pilot stage throttle is located after the U-shaped groove is opened on the inclined groove wall. It can be found from Figure 12 that after opening the U-groove, the minimum pressure in the bottom area of the 2D valve pilot stage throttle is greater than the saturation pressure, but the structure of W/D = 1.4 and W/D = 2.2 has a local area close to the saturation pressure, so W/D = 1.8 is more likely to avoid cavitation.

The pressure distribution at the bottom of the inclined groove after opening the U-shaped groove shown in Figure 13 confirms that the opening of the U-shaped groove is beneficial to avoid cavitation in the 2D valve pilot stage.

Figure 14 shows the gas volume fraction at the orifice of the U-shaped slot. The gas volume distribution of the three width–depth ratio structures is equivalent compared to the original inclined groove structure under the same pressure difference in Figure 8. After U-shaped grooves are opened on both sides of the valve sleeve groove, the maximum gas volume fraction near the downstream of the throttle decreases by tens of times. Therefore, U-shaped grooves on both sides of the valve sleeve groove can effectively restrain the cavitation in the 2D valve pilot stage.

Figure 15 compares the cavitation number between the original inclined groove structure and U-shaped groove structure. The results show that the cavitation number of the 2D valve pilot stage is significantly increased after U-shaped grooves are opened on both sides of valve sleeve grooves, indicating that U-shaped grooves on both sides of inclined grooves can restrain the occurrence of cavitation in the flow channel of the 2D valve pilot stage. Furthermore, as the width–depth ratio of U-shaped grooves increases, the cavitation number in the pilot stage of the 2D valve rises.

**Conclusion**

In this paper, the cavitation flow in the pilot stage of the 2D valve is examined through computational fluid dynamics. The effects of inlet and outlet pressure difference, outlet pressure, and valve opening velocity on cavitation are analyzed. The inhibitory effect of U-shaped grooves on both sides of the inclined groove of the valve sleeve on the cavitation of the 2D valve pilot stage was studied. The results show that the greater the pressure difference, higher is the cavitation intensity is. Additionally, the cavitation intensity increases with the decrease of outlet pressure. The cavitation in the 2D valve pilot stage is also related to the valve opening speed. The cavitation phenomenon happens when the valve opening speed is high, and there may be no cavitation in the flow channel with the decrease of the opening speed. After opening U-shaped grooves on both sides of the valve sleeve grooves, the maximum
gas volume fraction near the downstream of the throttle decreased by tens of times compared with the original inclined groove structure under the same pressure difference. Thus, the opening of U-shaped grooves on both sides of the valve sleeve grooves can effectively restrain the occurrence of cavitation in the 2D valve pilot stage. The cavitation number in the pilot stage of the 2D valve increases as the width–depth ratio of U-shaped grooves increases.

Declarations

All co-authors agree with the contents of the manuscript and there are no conflicts of interest to declare. We certify that the submission is original work and is not under review at any other publication. I had full access to all of the data in this study and I take complete responsibility for the integrity of the data and the accuracy of the data analysis.

Acknowledgements

This work was financially supported by National Key Research and Development Project (Project No.2019YFB2005201), the National Natural Science Foundation of China (Project No. 51675482), and the Scientific Research Project of Jiaxing Vocational And Technical College(Project No.JZYJ202212).

Data Availability Statements

The data that support the findings of this study are available from the corresponding author, Xing, upon reasonable request.

Author contributions statement

Zhao: Software, Data analysis, Writing-Original draft preparation and Editing. Xing: Conceptualization, Methodology. Ding: Validation. Wang: Software, Investigation. Ruan: Supervision. All authors reviewed the manuscript.

References


Figures
Figure 1

Working principle and structure diagram of 2D valve

Figure 2

(a) 3D model of 2D valve

(b) Fluid model
2D valve model and its fluid model

(a) The slot in the valve sleeve  (b) Fluid model before slotting  (c) Slotted fluid model

Figure 3

Slotted valve sleeve and fluid model

Figure 4
2D valve runner grid

Figure 5
Velocity cloud map at the throttle

Figure 6
Pressure cloud diagram at the throttle
Figure 7

Pressure distribution on the bottom of the chute

Figure 8
Gas volume fraction at the throttle

Figure 9

Cavitation number of different outlet pressures under the same pressure difference
Figure 10

Cavitation number of different outlet pressures at the same pressure ratio
Figure 11

Cavitation number of different spool speed at the same inlet and outlet pressure

Figure 12
Pressure cloud diagram of U-slot throttle

**Figure 13**
Pressure distribution on the bottom of U-shaped grooves

![Pressure distribution on the bottom of U-shaped grooves](image)

**Figure 14**
Gas volume fraction of throttle with U-shaped slot

<table>
<thead>
<tr>
<th>W/D</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.4</td>
<td><img src="image" alt="Image" /></td>
</tr>
<tr>
<td>1.8</td>
<td><img src="image" alt="Image" /></td>
</tr>
<tr>
<td>2.2</td>
<td><img src="image" alt="Image" /></td>
</tr>
</tbody>
</table>
Figure 15

Cavitation numbers of different structures