Numerical Simulation and mechanism investigations of cavitating noise around a two-dimension Valve pilot Stage

ZHAO Yonghuaw, b RUAN Jian, LU Qianqian

w College of Mechanical Engineering, Zhejiang University of Technology, Hangzhou, Zhejiang 310014, China

b Jiaxing Vocational & Technical College, Jiaxing, Zhejiang 314036, China

c Zhejiang University City College, Hangzhou, Zhejiang 310015, China

Abstract: The two-dimensional valve integrate the pilot stage and the power stage on one spool, which is easy to achieve fast valve operation and high frequency response. It has the advantages of simple structure, stable performance and high power to weight ratio. In this paper, the fluid flow in the pilot stage of a 2D valve is analyzed by computational fluid dynamics, and the influence of cavitation on the flow field in the valve is the main study object. The results show that the throttling action of the valve port forms a high-speed jet in the chute area of the valve sleeve, and a large area of cavitation cavitiation is generated at the same time. The velocity vortex and cavitation work together to intensify the pulsation of the fluid in the valve and become the main source of the noise. In the 2D valve pilot stage, the sound pressure level at the back of the chute of the valve sleeve reaches 175 dB, and the maximum sound pressure level at the middle and outlet reaches 168 dB. The sound pressure level at the back and middle of the chute decreases first and then increases, and the sound pressure level at the outlet of the chute decreases slowly. In the range of 0~5000 Hz, the noise energy is concentrated in the low frequency band of less than 1000 Hz, showing typical cavitation noise characteristics. The numerical results are in good agreement with the experimental results.

Keywords: Two-dimension valve; Large Eddy Simulation (LES); Cavitation noise; Vortex structures

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.\textsuperscript{1,2}

The observation is one of the methods to study cavitation. However, it is hard to observe the cavitation flow and its dynamic process in hydraulic valve because of the complex flow passage and narrow space in hydraulic valve. Even with the half-cut model and the medical probe to observe the cavitation flow in the cone and ball valves, they can only capture the overall cavitation flow distribution but cannot effectively capture the flow vacuoles.\textsuperscript{3,4} At present, many cavitation researches are based on numerical simulation analysis and then verified by measuring cavitation noise. In the numerical calculation, the Large Eddy Simulation (LES) combined with acoustic analogy is the most used simulation method.\textsuperscript{5-10} The application of LES in inducing noise, controlling surface pressure fluctuation, analyzing the speed statistically and the prediction of far-field noise agrees well with the experimental results.\textsuperscript{6-8} Meng Kunyu\textsuperscript{11} reviewed the prediction of hydrodynamic noise by LES combined with acoustic analogy theory, and calculated the pressure pulsation around the flow field with this method. Zhang Yun et al.\textsuperscript{12} verified the accuracy of numerical simulation results by using LES combined with Lighthill acoustic analogy through experiments. Another simulation method proposed by Deardorff,\textsuperscript{13} which is combined LES with RANS, can achieve a good simulation calculation for the flow noise with solid wall.

Using the calculation module provided by FLUENT, the accuracy of acoustic field prediction based on acoustic analogy theory can be guaranteed as long as the far field condition of acoustic radiation prediction is met. Many researchers have verified the rationality of the prediction results by experiments.\textsuperscript{14-20}

This paper takes 2D valve as the research object, using FLUENT to carry out numerical simulation to analyze the influence of cavitation on the flow field in the pilot structure of 2D valve, and studying on the change of sound field in the valve.

1. Working principle of two-dimensional valve

The working principle of the two-dimensional valve is shown in figure 1. The two overlapping areas formed by the high and low pressure grooves and the inclined grooves at the pilot stage of the valve act as a throttle, forming a half-bridge loop that drives the spool to move in a straight line through the rotational movement of the spool. In order to ensure the position feedback of the spool, the tangent of the side of the high-pressure throttle port and the low-pressure throttle port is inclined to a certain angle with the axis of the spool. When the spool moves, the oil flows out of the throttle port to form a space jet angle, and the resulting hydraulic force has axial, radial and circumferential components relative to the axis of the spool. The fluid flow at the port of the pilot stage valve directly affects the motion characteristics of the spool.\textsuperscript{21}
2. Pre-processing 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 mm².

2.2 Grid division, grid independence test and monitoring point setting

The MESH software is used to divide the mesh. The slip surface is refined locally using tetrahedral elements, and the throttle leads to improved accuracy. Three monitoring points P1, P2, and P3 are arranged along center line of valve.
sleeve chute Z direction. The monitoring point P1 is located at the rear of the chute of the valve sleeve, P2 is located in the middle of the chute area (nearest to the throttle orifice), and P3 is located at the midpoint of the model outlet.

![Grid and monitoring points](image)

**Fig.3 Grid and monitoring points**

Under different number of grids, comparing the average pressure value of the spool symmetry plane upstream of the throttle orifice with the maximum gas volume fraction in downstream sleeve chute area, as shown in table 1, it can be found that the differences between the parameters under study are small when the number of grids reached 223568, and meet the requirements of the grid independence. Therefore, grid 4 was selected for numerical calculation.

<table>
<thead>
<tr>
<th>Grid type</th>
<th>Grid number</th>
<th>Maximum gas volume fraction</th>
<th>Pressure value (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid 1</td>
<td>50712</td>
<td>0.09965</td>
<td>4097114</td>
</tr>
<tr>
<td>Grid 2</td>
<td>113656</td>
<td>0.11583</td>
<td>4110983</td>
</tr>
<tr>
<td>Grid 3</td>
<td>151682</td>
<td>0.13561</td>
<td>4360387</td>
</tr>
<tr>
<td>Grid 4</td>
<td>223568</td>
<td>0.13688</td>
<td>4519856</td>
</tr>
<tr>
<td>Grid 5</td>
<td>247714</td>
<td>0.13690</td>
<td>4520104</td>
</tr>
</tbody>
</table>

3. Calculation model and boundary conditions

The FW-H equation in FLUENT uses the most general Lightill noise analogy method. The specific process is as follows: by using the time domain integral equation of FLUENT and a small number of surface integrals calculation, the LES model was solved to obtain the time precision solution of the flow field variables. Thus the noise information of the specified position is obtained. Sound pressure signals can be
obtained by fast Fourier transform (FFT). After processing by FLUENT, all sound pressure level (SPL) and noise data graph in energy spectrum range are obtained.\textsuperscript{25,26} FLUENT can satisfy the accuracy of the flow solution. By performing noise calculation simultaneously during the transient solution process, the FW-H noise model can predict the propagation of sound waves to free space well.\textsuperscript{27}

3.1 Large eddy simulation governing equation

Direct numerical simulation (DNS) calculates the noise by improving the resolution and refining the mesh elements. Theoretically, it can accurately calculate the interaction between fluid flow and sound wave, but it requires the grid to be smaller than the smallest vortex in the entire flow field, which requires a large amount of calculation and high calculation cost. LES is the most promising method in the field of fluid-induced noise,\textsuperscript{27} and it can effectively avoid the limitation of Reynolds number on calculation scale in DNS method. Therefore, the large eddy simulation equation is selected for transient calculation in this paper.

The governing equation of LES model is obtained by filtering in wave number space and physical space with the Navier-Stokes equation. The process of filtering is to remove the vortices smaller than the filter width or the given physical width, so as to obtain the governing equation of large eddy simulation as follows:

\[
\frac{\partial \rho}{\partial t} + u \frac{\partial \rho u_i}{\partial x_i} = 0
\]

\[
\frac{\partial}{\partial t} \left( \rho u_i \right) + \frac{\partial}{\partial x_j} \left( \rho u_i u_j + \rho \tau_{ij} \right) = \frac{\partial}{\partial x_j} \left( \rho \frac{\partial u_i}{\partial x_j} \right) - \frac{\partial p}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j}
\]

where \( \tau_{ij} \) is the subgrid stress, \( \tau_{ij} = \rho \bar{u}_i \bar{u}_j - \rho u_i u_j \).

3.2 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,\textsuperscript{22} which is displayed below.

\[
R_s = \frac{\rho \rho_p}{\rho_n} \alpha_s (1 - \alpha_s) \left( \frac{2 P - P}{3 \rho_p} \right)^{1/2}, P \leq P_c
\]

\[
R_s = \frac{\rho \rho_p}{\rho_n} \alpha_s (1 - \alpha_s) \left( \frac{2 P - P}{3 \rho_p} \right)^{1/2}, P \geq P_c
\]
where, \( R_\theta = \left( \frac{\alpha_s}{1-\alpha_s} \right) \left( \frac{1}{\pi n_0} \right)^{\frac{1}{3}} \), \( R_\theta \) is the cavitation radius, \( P_v \) is the saturated vapor pressure of fluid. saturated steam pressure of hydraulic oil is 37100Pa at 20°C, \( n_0 \) is the numerical density of cavitation per unit liquid volume, \( n_0 = 10^{13} \).  

3.3 Noise model

In FLUENT, the FW-H equation proposed by Ffowcs Williams and Hawkings is used to simulate the generation and propagation of sound. Lightill acoustic analogy model is adopted in this equation, and the specific form is as follows:

\[
\frac{1}{c_0^2} \frac{\partial^2}{\partial t^2} \left[ PH(f) \right] = \frac{\partial}{\partial t}\left[ \rho_0 \mu \nabla f \delta (f) \right] - \frac{\partial}{\partial x_j} \left[ \rho_0 n_0 \nabla f \delta (f) \right] + \frac{\partial^2}{\partial x_i \partial x_j} \left[ \tau_{ij} H(f) \right] \quad (5)
\]

where \( \tau_{ij} \) is the Lighthill stress tensor, \( \delta (f) \) is the Kronecker function, \( H(f) \) is the Heaviside function, \( \delta (f) \) is the Dirac delta function, \( c_0 \) is the local sound speed, \( P \) is the sound pressure.

The three items to the right of medium in Equation (5) represent monopole, dipole and quadrupole sound sources respectively.

3.4 Boundary condition

The flow field and sound field of 2D valve pilot stage were numerically simulated by FLUENT. The cavitation model of mixed multiphase flow mode and large eddy simulation model were used. Pressure velocity coupling algorithm, the first order upwind scheme for calculation. The main phase is defined as hydraulic oil with a density of 780 kg/m³ and a viscosity of 0.0024 kg/m⋅s. The subphase is air, with a density of 1.225 kg/m³ and a viscosity of 1.789×10⁻⁵ kg/m⋅s. The conversion between the primary phase and the subphase satisfies the Cavitation model. The numerical simulation adopts pressure inlet, pressure outlet and non-slip standard wall function boundary conditions. The inlet pressure is 14MPa and the outlet pressure is 0.1MPa.

The Mixture model in Fluent software is selected for numerical calculation. The numerical simulation defines that the main phase is hydraulic oil with a density of 780 kg/m³, and viscosity 0.0024 kg/m⋅s, and the secondary phase is air with a density of 1.225 kg/m³, and a viscosity of 1.789×10⁻⁵ kg/m⋅s. The transition between the main phase and the secondary phase satisfies the cavitation model. The inlet of the model is defined as the pressure inlet, the pressure is 14mpa, the exit is the pressure outlet, and the back pressure is 0.1 MPa, using the slip grid model.

According to Nyquist sampling theorem:
\[ t = \frac{1}{2f} \]

Considering the calculation time and the accuracy of the calculation results, the analysis time step \( t \) was set as 0.0001s, and 2400 steps were calculated to obtain the spectrum with the frequency \( f \) ranging from 0 to 5000Hz.

4. Analysis of simulation results

Characteristic moments T1, T2 and T3 within an analysis period were selected to study the flow characteristics of each analysis surface, corresponding to the three moments when the orifice opening was 0.005mm, 0.01mm and 0.02mm, respectively. As shown in figure 4, The analysis plane are: Analysis plane 1 (section XOZ in the direction of fluid flow), Analysis plane 2 (section YOZ in the direction of fluid flow), Analysis plane 3 (section XOY through the axis of transition channel and perpendicular to the axis of inlet channel), Analysis plane 4 (2mm away from analysis plane 3), Analysis plane 5 (4mm away from analysis plane 3).

![Fig.4 The location of each analysis plane](image)

4.1 Fluid field analysis

The fluid characteristics on analysis plane 3 are shown in Figure 5. The fluid pressure in the upstream transition section in the throttle orifice is stable. After passing the throttle orifice, the fluid pressure drops rapidly and forms a negative pressure area in the middle of the downstream valve sleeve chute. The speed is shown on the cloud map. Due to the throttling effect of the orifice, the high speed jet flow is formed in the lower reaches. Compare the three moments, the maximum flow rate of the throttle port at T3 reaches 167.8m/s. It can be seen from the gas volume fraction diagram that a large area of cavitation air mass is formed in the negative pressure region corresponding to the pressure cloud diagram in the middle of the chute.
In the valve sleeve chute, the high speed fluid and low speed fluid shear to form a vortex, and form a typical screw type rotational flow. It can be seen from the flow characteristics of the analysis plane in FIG. 6 that a fixed wall guide vortex is formed on the side of the orifice along the tangential direction of the fluid flow direction (as shown in FIG. 6(a) and (b)). According to the flow pattern shown in analysis plane 1, in addition to a large guide vortex near the outlet, there are also small vortexes generated near the orifice and behind the chute, and the number of small vortexes tends to increase with the increase of valve opening. The existence of these vortexes affects the consistency of the main flow direction of the fluid. FIG. 6(c), (d) and (e) show that there is a large scale vortex with obvious characteristic in the transverse direction of the fluid flow in the chute, and the secondary flow of this vortex on the transverse section is very significant. The strong eddy flow greatly weakens the
potential energy of the main flow. At the same time, due to the asymmetry of the distribution of the orifice, the shear flow effect along the two sides of the chute is inconsistent, the whirlpools of different sizes are formed. This difference increases the complexity of fluid flow in the chute.

According to the above flow field analysis, due to the throttling action of the orifice, high-speed jet and large area cavitation are formed in the chute area of the valve sleeve. The velocity vortex and cavitation caused by the high speed jet act together to intensify the fluid pulsation in the valve sleeve chute area. Large scale fluid vortex and cavitation are the main source of noise in the valve.  

(a)  

(b)  

(c)  

(d)
4.2 Sound field analysis

Figure 7(a)–(d) shows the frequency domain characteristics of the monitoring points P1, P2 and P3. The maximum sound pressure level at P1 is 175 dB, while that at P2 and P3 are both 168dB. The change of sound pressure level at P1 and P2 is consistent. They both decrease slowly at first, then increase gradually, and finally rise from 120dB to 135dB. The sound pressure level at P3 showed a slow decreasing trend and finally dropped to 110dB. The power spectrum density of the three monitoring points shows that P1 has an obvious secondary frequency peak at 700Hz, and no obvious sound pressure frequency appeared after 1000Hz. P2 has a concentrated and uniform frequency peak near 1000Hz, and no obvious sound pressure frequency in the subsequent frequency band. The sound pressure frequency of P3 does not change after 700Hz. It can be seen from the sound pressure level and density spectrum that the cavitation noise at the back of the valve sleeve chute reflects a higher degree of cavitation than the other two positions. The development of cavitation in the middle of the chute is more sustained. And the lowest noise sound pressure level is found at the outlet of the model. All these show that cavitation has a great influence on the flow of fluid in the valve.

In FIG. 8, Based on the same test conditions, the frequency spectrum density of the monitoring point obtained by numerical calculation is normalized and compared with the noise spectrum of the two-dimensional valve sensitive cavity measured in reference 30. The results show that the spectral density distribution of the monitoring point P3 agrees well with the experimental results in the range of 0~1000Hz.
5. Concluding

In this paper, the flow of fluid in the pilot stage of a two-dimensional valve is studied by numerical simulation, the influence of cavitation on the flow field is analyzed, and the following conclusions are drawn as follows:
Due to the throttling effect of the orifice, a high-speed jet and a large area of cavitation are formed in the chute area of the valve sleeve. The speed vortex and cavitation work together to aggravate the fluid pulsation in the chute of the valve sleeve and become the main source of noise in the valve.

In the 2d valve pilot stage, the sound pressure level at the rear of the valve sleeve chute is the highest, up to 175 dB. The maximum sound pressure level in the middle and outlet of the chute is 168 dB. The sound pressure level in the rear and middle of the chute firstly decreased and then increased, and finally rose to 135 dB within 0~5000 Hz. The sound pressure level at the outlet of the chute decreases slowly, and the final value is 110 dB.

The energy of the noise is concentrated in the low frequency band within 1000 Hz, showing typical cavitation noise characteristics. The numerical calculation can simulate the noise of 2D valve flow field, and agrees well with the experimental result.

Acknowledgements
This work was financially supported by the National Natural Science Foundation of China (Project No. 51675482), and National Key Research and Development Project (Project No.2019YFB2005201).

Data Availability Statements: All data sets for the paper's conclusions are available to the reader.

Reference


24 Zhu Mingming, Huang Biao, Wang Guoyu, etc. Numerical simulation of noise caused by


